ALTAIR

ONLY FORWARD

Altair newFASANT 2025

User Guide

Updated: 12/14/2024

Intellectual Property Rights Notice

Copyright © 1986-2024 Altair Engineering Inc. All Rights Reserved.

This Intellectual Property Rights Notice is exemplary, and therefore not exhaustive, of the intellectual property rights held by Altair Engineering Inc. or its affiliates. Software, other products, and materials of Altair Engineering Inc. or its affiliates are protected under laws of the United States and laws of other jurisdictions.

In addition to intellectual property rights indicated herein, such software, other products, and materials of Altair Engineering Inc. or its affiliates may be further protected by patents, additional copyrights, additional trademarks, trade secrets, and additional other intellectual property rights. For avoidance of doubt, copyright notice does not imply publication. Copyrights in the below are held by Altair Engineering Inc. or its affiliates. Additionally, all non-Altair marks are the property of their respective owners. If you have any questions regarding trademarks or registrations, please contact marketing and legal.

This Intellectual Property Rights Notice does not give you any right to any product, such as software, or underlying intellectual property rights of Altair Engineering Inc. or its affiliates. Usage, for example, of software of Altair Engineering Inc. or its affiliates is governed by and dependent on a valid license agreement.

Altair® HyperWorks®, a Design & Simulation Platform

Altair® AcuSolve® ©1997-2024

Altair® Activate® ©1989-2024

Altair® Automated Reporting Director™ ©2008-2022

Altair® Battery Damage Identifier™ ©2019-2024

Altair® CFD™ ©1990-2024

Altair Compose® ©2007-2024

Altair® ConnectMe™ ©2014-2024

Altair® DesignAI™ ©2022-2024

Altair® DSim™ ©2024

Altair® EDEM™ ©2005-2024

Altair® EEvision™ ©2018-2024

Altair® ElectroFlo™ ©1992-2024

Altair Embed® ©1989-2024

Altair Embed® SE ©1989-2024

Altair Embed®/Digital Power Designer ©2012-2024

Altair Embed®/eDrives ©2012-2024

Altair Embed® Viewer©1996-2024

Altair® e-Motor Director™©2019-2024

Altair® ESAComp® ©1992-2024

Altair® expertAI™ ©2020-2024

Altair® Feko® ©1999-2024

Altair® FlightStream® ©2017-2024

Altair® Flow Simulator™ ©2016-2024

Altair® Flux® ©1983-2024

Altair® FluxMotor® ©2017-2024

Altair® GateVision PRO™ ©2002-2024

Altair® Geomechanics Director™ ©2011-2022

Altair® HyperCrash® ©2001-2023

Altair® HyperGraph® ©1995-2024

Altair® HyperLife® ©1990-2024

Altair® HyperMesh® ©1990-2024

Altair® HyperMesh® CFD ©1990-2024

Altair® HyperMesh ® NVH ©1990-2024

Altair® HyperSpice™ ©2017-2024

Altair® HyperStudy® ©1999-2024

Altair® HyperView® ©1999-2024

Altair® HyperView Player® ©2022-2024

Altair® HyperWorks® ©1990-2024

Altair® HyperWorks® Design Explorer ©1990-2024

Altair® HyperXtrude® ©1999-2024

Altair® Impact Simulation Director™ ©2010-2022

Altair® Inspire™ ©2009-2024

Altair® Inspire™ Cast ©2011-2024

Altair® Inspire™ Extrude Metal ©1996-2024

Altair® Inspire™ Extrude Polymer ©1996-2024

Altair® Inspire™ Form ©1998-2024

Altair® Inspire™ Mold ©2009-2024

Altair® Inspire™ PolyFoam ©2009-2024

Altair® Inspire™ Print3D ©2021-2024

Altair® Inspire™ Render ©1993-2024

Altair® Inspire™ Studio ©1993-2024

Altair® Material Data Center™ ©2019-2024

Altair® Material Modeler™ ©2019-2024

Altair® Model Mesher Director™ ©2010-2024

Altair® MotionSolve® ©2002-2024

Altair® MotionView® ©1993-2024

Altair® Multi-Disciplinary Optimization Director™ ©2012-2024

Altair® Multiscale Designer® ©2011-2024

Altair® newFASANT™ ©2010-2020

Altair® nanoFluidX® ©2013-2024

Altair® NLVIEW® ©2018-2024

Altair® NVH Director™ ©2010-2024

Altair® NVH Full Vehicle™ ©2022-2024

Altair® NVH Standard™ ©2022-2024

Altair® OmniV™ ©2015-2024

Altair® OptiStruct® ©1996-2024

Altair® physicsAI™ ©2021-2024

Altair® PollEx™ ©2003-2024

Altair® PollEx™ for ECAD ©2003-2024

Altair® PSIM™ ©1994-2024

Altair® Pulse™ ©2020-2024

Altair® Radioss® ©1986-2024

Altair® romAI™ ©2022-2024

Altair® RTLvision PRO™ ©2002-2024

Altair® S-CALC™ ©1995-2024

Altair® S-CONCRETE™ ©1995-2024

Altair® S-FRAME® ©1995-2024

Altair® S-FOUNDATION™ ©1995-2024

Altair® S-LINE™ ©1995-2024

Altair® S-PAD™ ©1995-2024

Altair® S-STEEL™ ©1995-2024

Altair® S-TIMBER™ ©1995-2024

Altair® S-VIEW™ ©1995-2024

Altair® SEAM® ©1985-2024

Altair® shapeAI™ ©2021-2024

Altair® signalAI™ ©2020-2024

Altair® Silicon Debug Tools™ ©2018-2024

Altair® SimLab® ©2004-2024

Altair® SimLab® ST ©2019-2024

Altair® SimSolid® ©2015-2024

Altair® SpiceVision PRO™ ©2002-2024

Altair® Squeak and Rattle Director™ ©2012-2024

Altair® StarVision PRO™ ©2002-2024

Altair® Structural Office™ ©2022-2024

Altair® Sulis™ ©2018-2024

Altair® Twin Activate® ©1989-2024

Altair® UDE™ ©2015-2024

Altair® ultraFluidX® ©2010-2024

Altair® Virtual Gauge Director™ ©2012-2024

Altair® Virtual Wind Tunnel™ ©2012-2024

Altair® Weight Analytics™ ©2013-2022

Altair® Weld Certification Director™ ©2014-2024

Altair® WinProp™ ©2000-2024

Altair® WRAP™ ©1998-2024

Altair® HPCWorks®, a HPC & Cloud Platform

Altair® Allocator™ ©1995-2024

Altair® Access™ ©2008-2024

Altair® Accelerator™ ©1995-2024

Altair® Accelerator™ Plus ©1995-2024

Altair® Breeze™ ©2022-2024

Altair® Cassini™ ©2015-2024

Altair® Control™ ©2008-2024

Altair® Desktop Software Usage Analytics™ (DSUA) ©2022-2024

Altair® FlowTracer™ ©1995-2024

Altair® Grid Engine® ©2001, 2011-2024

Altair® InsightPro™ ©2023-2024

Altair® Hero™ ©1995-2024

Altair® Liquid Scheduling™ ©2023-2024

Altair® Mistral™ ©2022-2024

Altair® Monitor™ ©1995-2024

Altair® NavOps® ©2022-2024

Altair® PBS Professional® ©1994-2024

Altair® PBS Works™ ©2022-2024

Altair® Simulation Cloud Suite (SCS) ©2024

Altair® Software Asset Optimization (SAO) ©2007-2024

Altair® Unlimited™ ©2022-2024

Altair® Unlimited Data Analytics Appliance™ ©2022-2024

Altair® Unlimited Virtual Appliance™ ©2022-2024

Altair® RapidMiner®, a Data Analytics & AI Platform

Altair® AI Hub ©2023-2024

Altair® AI Edge™ ©2023-2024

Altair® AI Cloud ©2022-2024

Altair® AI Studio ©2023-2024

Altair® Analytics Workbench™ ©2002-2024

Altair® Graph Lakehouse™ ©2013-2024

Altair® Graph Studio™ ©2007-2024

Altair® Knowledge Hub™ ©2017-2024

Altair® Knowledge Studio® ©1994-2024

Altair® Knowledge Studio® for Apache Spark ©1994-2024

Altair® Knowledge Seeker™ ©1994-2024

Altair® IoT Studio™ ©2002-2024

Altair® Monarch® ©1996-2024

Altair® Monarch® Classic ©1996-2024

Altair® Monarch® Complete™ ©1996-2024

Altair® Monarch® Data Prep Studio ©2015-2024

Altair® Monarch Server™ ©1996-2024

Altair® Panopticon™ ©2004-2024

Altair® Panopticon™ BI ©2011-2024

Altair® SLC™ ©2002-2024

Altair® SLC Hub™ ©2002-2024

Altair® SmartWorks™ ©2002-2024

Altair® RapidMiner® ©2001-2024

Altair One® ©1994-2024

Altair® CoPilot™ ©2023-2024

Altair® License Utility™ ©2010-2024

Altair® TheaRender® ©2010-2024

OpenMatrix™ ©2007-2024

OpenPBS® ©1994-2024

OpenRadioss™ ©1986-2024

October 7, 2024

Technical Support

Altair's support resources include engaging learning materials, vibrant community forums, intuitive online help resources, how-to guides, and a user-friendly support portal.

Visit Customer Support to learn more about our support offerings.

Contents

Intellectual Property Rights Notice ii
Technical Support viii

1 Utilities 23

1.1 Remote Server 24
1.2 Batch Meshing and Execution of Projects 27
1.3 High Performance Computing (HPC)/Queueing 29

2 GUI 31

2.1 GUI Overview 32
2.1.1 Console 32
2.1.2 Tree 33
2.1.3 ToolBar 43
2.1.4 StatusBar 45
2.1.5 Multi-Tab 46
2.2 File Menu 47
2.2.1 New (Ctrl+N) 47
2.2.2 Wizard 47
2.2.3 Open (Ctrl+O) 49
2.2.4 Recent Projects 50
2.2.5 Close 50
2.2.6 Save (Ctrl+S) 51
2.2.7 Save As (Ctrl+S) 51
2.2.8 Project Folder 52
2.2.9 Exit 52
2.3 View Menu 53
2.3.1 Reset View (Ctrl+R) 53
2.3.2 Zoom In 53
2.3.3 Zoom Out 53
2.3.4 Zoom Selected 53
2.3.5 Rotate 53
2.3.6 Pan 54
2.3.7 Select 54
2.3.8 Axis 54
2.3.9 Reference Plane 54
2.3.10 Set View 58
2.3.11 Render Mode 60
2.4 Edit Menu 62
2.4.1 Undo (Ctrl+Z) 62
2.4.2 Redo (Ctrl+Y) 62

2.4.3 History....62
2.4.4 Delete....63
2.4.5 Delete All....63
2.4.6 Copy (Ctrl+C)....63
2.4.7 Move (Ctrl+X)....64
2.4.8 Duplicate....65
2.4.9 Group....66
2.4.10 Explode....67
2.4.11 Geometric Operations....68
2.4.12 View Control Points....89
2.4.13 Invert Normals....91
2.4.14 Layers....92
2.4.15 Object Properties....93
2.5 Geometry Menu....95
2.5.1 Importing and Exporting....95
2.5.2 Primitives....97
2.5.3 Tools....187
2.5.4 Parameters....205
2.6 Tools....210
2.6.1 DIA Files....210
2.6.2 Diagram Correlation....227
2.6.3 Plot Viewer....227
2.6.4 Calculator....230
2.6.5 User Functions....230
2.6.6 Script....252
2.6.7 Batch....256
2.6.8 Remote....257
2.7 Help Menu....258
2.7.1 About....258
2.8 Annex 1: Graphics Advanced Options....259
2.8.1 Properties....259
2.8.2 Copy....265
2.8.3 Save As....265
2.8.4 Print....266
2.8.5 Zoom In....267
2.8.6 Zoom Out....267
2.8.7 Auto Range....268
2.9 Annex 2: Inverting Normals....269

3 Post-Process....275

3.1 Post-Process Menu....276
3.2 Post-Process Window....277
3.3 Trace Window....279
3.3.1 Trace Options Panel....279
3.3.2 Trace Representation Panel....280
3.4 Trace Results Output....283

3.5 Examples ... 287
3.5.1 Example 1: Table Trace ... 287
3.5.2 Example 2: Plot Trace ... 289
3.5.3 Example 3: Diagram 2D Trace ... 290
3.5.4 Example 4: Diagram 3D Trace ... 292

4 Antennas ... 294

4.1 Plane Wave ... 295
4.2 Dipole ... 297
4.2.1 Dipole Antenna ... 297
4.2.2 Dipole Array ... 301
4.2.3 Attach Dipole to Object ... 304
4.3 Pattern File ... 305
4.3.1 Pattern File Antenna ... 305
4.3.2 Pattern File Array ... 307
4.3.3 Attach Pattern File to Object ... 309
4.4 Equivalent Sources ... 311
4.4.1 Multipole Antenna ... 311
4.4.2 Insight ... 312
4.4.3 Define Near Field from Data ... 314
4.4.4 Import Feko Near Field (.EFE, .HFE) ... 316
4.5 Coaxial Feed ... 318
4.5.1 Add Feed Point ... 318
4.6 Waveguides ... 319
4.6.1 Add Waveguide Port ... 319
4.7 Primitive Antenna ... 321
4.7.1 Horns ... 321
4.7.2 Coaxial Feed ... 326
4.7.3 Slot Antennas ... 339
4.7.4 Patch Antennas ... 349
4.8 Radome ... 358
4.8.1 Define Interface Radome ... 358
4.8.2 Define Volumetric Radome ... 359
4.8.3 Define Radome by Material Surface ... 360
4.8.4 FSS Primitives ... 365
4.9 Reflectors ... 366
4.9.1 Simple Reflector ... 366
4.9.2 Cassegrain Reflector ... 368
4.9.3 Gregorian Reflector ... 371
4.9.4 Equivalent Antenna ... 374
4.10 Reflectarrays ... 377
4.10.1 New Layout ... 377
4.11 Import Pattern File ... 380
4.12 Edit Antenna ... 381
4.13 Delete Antenna ... 382
4.14 Antenna Size ... 383

4.15 Antenna Feeding ... 384

5 Materials ... 385

5.1 Materials Menu (MOM Module) ... 386
5.1.1 Add ... 386
5.1.2 Edit ... 391
5.1.3 Delete ... 391
5.1.4 Assign ... 391
5.1.5 Material Properties ... 392
5.2 Materials Menu (MONCROS Module) ... 394
5.2.1 Add ... 394
5.2.2 Edit ... 397
5.2.3 Delete ... 398
5.2.4 Assign ... 398
5.3 Materials Menu (PO Module) ... 400
5.3.1 Add ... 400
5.3.2 Edit ... 404
5.3.3 Delete ... 405
5.3.4 Assign ... 405
5.3.5 Material Properties ... 406
5.4 Materials Menu (GTD and GTD-PO Modules) ... 408
5.4.1 Add ... 408
5.4.2 Edit ... 412
5.4.3 Delete ... 412
5.4.4 Assign ... 413
5.4.5 Material Properties ... 414
5.5 Materials Menu (Periodical Structures Module) ... 415
5.5.1 Add ... 415
5.5.2 Edit ... 416
5.5.3 Delete ... 416
5.6 Materials Menu (US Module) ... 418
5.6.1 Add ... 418
5.6.2 Edit ... 421
5.6.3 Delete ... 421
5.6.4 Assign ... 422
5.6.5 Properties ... 423
5.7 APPENDIX A: Color Selection ... 424

6 Command Line ... 427

6.1 Using the Command Line ... 428
6.2 List of Commands Ordered by Functionality ... 429
6.3 Command Reference ... 433
6.3.1 analyzeCurvature ... 437
6.3.2 analyzeEdges ... 438

6.3.3 analyzeNakedBorders ... 439
6.3.4 angle ... 440
6.3.5 arc ... 441
6.3.6 area ... 443
6.3.7 array ... 444
6.3.8 arrayOnSurface ... 446
6.3.9 axis ... 451
6.3.10 bentDipoles ... 452
6.3.11 booleanDifference ... 453
6.3.12 booleanInside ... 456
6.3.13 booleanIntersection ... 458
6.3.14 booleanOutside ... 461
6.3.15 booleanSplit ... 464
6.3.16 booleanUnion ... 467
6.3.17 booleanXOR ... 469
6.3.18 box ... 472
6.3.19 capacitance ... 474
6.3.20 circle ... 475
6.3.21 circularHole ... 476
6.3.22 cone ... 477
6.3.23 connect2Surfaces ... 479
6.3.24 connectCurves ... 482
6.3.25 coons ... 483
6.3.26 copy ... 486
6.3.27 cross ... 488
6.3.28 crossHole ... 490
6.3.29 curve ... 491
6.3.30 curveFunction ... 493
6.3.31 curveInterpolation ... 495
6.3.32 cylinder ... 497
6.3.33 delete ... 499
6.3.34 disk ... 500
6.3.35 distance ... 501
6.3.36 duplicate ... 502
6.3.37 ellipse ... 503
6.3.38 ellipsoid ... 505
6.3.39 exit ... 507
6.3.40 explode ... 508
6.3.41 export ... 509
6.3.42 extendHole ... 510
6.3.43 extrude ... 513
6.3.44 extrudeNormal ... 515
6.3.45 extrudeSurface ... 517
6.3.46 flare ... 519
6.3.47 group ... 521
6.3.48 help ... 522
6.3.49 hexagon ... 523

6.3.50 hyperbola 524
6.3.51 hyperboloid 525
6.3.52 import 526
6.3.53 interdigitalLines 527
6.3.54 invertNormals 528
6.3.55 length 530
6.3.56 line 531
6.3.57 localTransform 532
6.3.58 meshFromSurfaces 535
6.3.59 meshToSurfaces 536
6.3.60 move 537
6.3.61 ogive 539
6.3.62 openRing 542
6.3.63 parabola 543
6.3.64 paraboloid 544
6.3.65 perpendicular 545
6.3.66 pipe 548
6.3.67 planarSurface 550
6.3.68 plane 552
6.3.69 point 554
6.3.70 pointCloud 555
6.3.71 pointsFromCurve 557
6.3.72 pointsFromCurveByDistance 558
6.3.73 pointsFromSurface 559
6.3.74 polygon 561
6.3.75 project 563
6.3.76 rebuildCurve 566
6.3.77 rebuildSurface 567
6.3.78 rectangle 570
6.3.79 rectangularHole 572
6.3.80 rename 573
6.3.81 revolve 574
6.3.82 rhomboid 577
6.3.83 ring 579
6.3.84 rotate 580
6.3.85 scale 582
6.3.86 scale1D 584
6.3.87 scale2D 587
6.3.88 scale3D 590
6.3.89 scaleNonUniform 593
6.3.90 script 596
6.3.91 segment 597
6.3.92 semicircle 598
6.3.93 set 599
6.3.94 skinned 601
6.3.95 sphere 603
6.3.96 spiral 605

6.3.97 spiralCross ... 606
6.3.98 splitByProjection ... 607
6.3.99 splitCurves ... 610
6.3.100 splitRing ... 612
6.3.101 splitSurfaceByIsoparam ... 613
6.3.102 splitSurfaceByIsoparamGrid ... 615
6.3.103 splitSurfaces ... 616
6.3.104 stripCross ... 618
6.3.105 stripHexagon ... 619
6.3.106 stripTrifilar ... 620
6.3.107 surface ... 621
6.3.108 surfaceCornerPoints ... 623
6.3.109 surfaceEdges ... 624
6.3.110 surfaceEllipse ... 626
6.3.111 surfaceFromEdges ... 627
6.3.112 surfaceFunction ... 630
6.3.113 surfaceInterpolation ... 632
6.3.114 surfaceIsoparamCurve ... 634
6.3.115 surfaceSpiral ... 636
6.3.116 sweep ... 637
6.3.117 symmetric ... 638
6.3.118 tangent ... 641
6.3.119 tCross ... 642
6.3.120 threeDipoles ... 643
6.3.121 threeDipolesUnion ... 644
6.3.122 torus ... 645
6.3.123 trifilar ... 647
6.3.124 trim ... 648
6.3.125 unset ... 651
6.3.126 untrim ... 652
6.4 Appendix I: List of Functions ... 654

7 GTD ... 658

7.1 File Menu ... 659
7.2 Edit Menu ... 660
7.3 View Menu ... 661
7.4 Geometry Menu ... 662
7.5 Materials Menu ... 663
7.6 Simulation Menu ... 664
7.6.1 Parameters ... 664
7.6.2 Creeping ... 667
7.6.3 Doppler ... 667
7.6.4 HPC Parameters ... 669
7.7 Antennas Menu ... 670
7.7.1 Dipole ... 670
7.7.2 Pattern File ... 674

7.7.3 Multipole Antenna....677
7.7.4 Import DIA File....678
7.7.5 Edit Antenna....678
7.7.6 Delete Antenna....679
7.7.7 Antenna Display Size....679
7.7.8 Antenna Feeding....680
7.8 Output Menu....682
7.8.1 Observation Directions....682
7.8.2 Observation Points....683
7.8.3 Coupling....692
7.9 Meshing Menu....699
7.9.1 Ignore Edges....699
7.9.2 Create Mesh....701
7.10 Calculate Menu....702
7.11 Show Results Menu....704
7.11.1 Common Options....704
7.11.2 Far Field....706
7.11.3 Near Field....711
7.11.4 Doppler Spectrum....721
7.11.5 Coupling....724
7.11.6 MIMO Results....730
7.11.7 Post-Process....735
7.12 Tools Menu....736
7.13 Help Menu....737
7.14 Training Examples....738
7.14.1 Example 1: Getting Started....738
7.14.2 Example 2: Dynamic Simulation....745
7.14.3 Example 3: Radiation Pattern File Simulation....751
7.14.4 MIMO Examples....757
7.14.5 Using Multipole in GTD Module....769

8 PO....777

8.1 File Menu....778
8.2 Edit Menu....779
8.3 View Menu....780
8.4 Geometry Menu....781
8.5 Materials Menu....782
8.6 Simulation Menu....783
8.6.1 Parameters Menu....783
8.6.2 Doppler....784
8.6.3 Delete Edges....786
8.6.4 Duplicate Normals....788
8.7 RCS Menu....789
8.7.1 Parameters....789
8.8 Output Menu (Alt +O)....790
8.8.1 Observation Directions....790

8.9 Meshing Menu ... 792
8.9.1 Create Mesh ... 792
8.9.2 Visualize Existing Mesh ... 794
8.9.3 View Log ... 795
8.10 Calculate Menu (Alt + C) ... 796
8.10.1 Execute ... 796
8.11 Show Results Menu ... 798
8.11.1 Far field ... 798
8.11.2 Post-Process ... 803
8.12 Tools Menu ... 804
8.13 Help Menu ... 805
8.14 Training Examples ... 806
8.14.1 Example 1 ... 806
8.14.2 Example 2 ... 817
8.14.3 Example 3 ... 825
8.14.4 Example 4: Monostatic RCS of a Ship ... 834

9 GTD-PO ... 839

9.1 File Menu ... 840
9.2 Edit Menu ... 841
9.3 View Menu ... 842
9.4 Geometry Menu ... 843
9.5 Materials Menu ... 844
9.6 Simulation Menu ... 845
9.6.1 Parameters ... 845
9.6.2 Creeping ... 848
9.6.3 Doppler ... 848
9.7 RCS Menu ... 851
9.7.1 Parameters ... 851
9.8 Source Menu ... 852
9.8.1 Source Parameters ... 852
9.8.2 Multipole Source ... 853
9.8.3 Pattern File ... 854
9.8.4 Import Pattern Files ... 856
9.9 Output Menu (Alt +O) ... 857
9.9.1 Observation Directions ... 857
9.9.2 Coupling ... 859
9.10 Meshing Menu ... 861
9.10.1 Create Mesh ... 861
9.10.2 Visualize Existing Mesh ... 864
9.10.3 Manual Surface Classification ... 865
9.10.4 Manual Divisions ... 866
9.11 Calculate Menu (Alt + C) ... 867
9.11.1 Execute ... 867
9.12 Show Results Menu ... 869
9.12.1 RCS ... 869

9.12.2 Field....874
9.12.3 Ray Tracing....879
9.12.4 Coupling....881
9.12.5 Doppler....883
9.13 Tools Menu....893
9.14 Help Menu....894
9.15 Training Examples....895
9.15.1 Example 1: Monostatic RCS of a Sphere and a Box....895
9.15.2 Example 2: Monostatic RCS of a Ship....906
9.15.3 Example 3: Bistatic RCS of a Box....913
9.15.4 Example 4: Coupling....919
9.15.5 Example 5: Doppler Spectrum....923
9.15.6 Example 6: Monostatic RCS of a Truck....929
9.15.7 Example 7: FMCW Analysis....933

10 MONCROS....945

10.1 File Menu....946
10.2 Edit Menu....947
10.3 View Menu....948
10.4 Geometry Menu....949
10.5 Materials Menu....950
10.6 Simulation Menu....951
10.6.1 Parameters....951
10.6.2 HPC Parameters....951
10.7 Solver Menu....952
10.7.1 Parameters....952
10.7.2 Advanced Options....958
10.8 RCS Menu....961
10.9 Output Menu....963
10.9.1 Observation Directions....963
10.9.2 Observation Points....965
10.10 Meshing Menu....972
10.10.1 Create Mesh....972
10.10.2 Visualize Existing Mesh....978
10.10.3 Visualize Mesh Log....979
10.11 Calculate Menu....981
10.11.1 Execute....981
10.12 Show Results Menu....983
10.12.1 Far Field....983
10.12.2 NearField....988
10.12.3 View Currents....995
10.12.4 View Charges....996
10.13 Tools Menu....998
10.14 Help Menu....999
10.15 Training Examples....1000
10.15.1 Example 1....1000

10.15.2 Example 2 ... 1010
10.15.3 Example 3 ... 1023
10.16 Annex 1: Post-Processing Currents ... 1029

11 MOM ... 1036

11.1 File Menu ... 1037
11.2 Edit Menu ... 1038
11.3 View Menu ... 1039
11.4 Geometry Menu ... 1040
11.5 Materials Menu ... 1041
11.6 Simulation Menu ... 1042
11.6.1 Parameters ... 1042
11.6.2 HPC Parameters ... 1044
11.7 Solver Parameters Menu ... 1045
11.7.1 Parameters ... 1045
11.7.2 Advanced Options ... 1052
11.8 Source ... 1055
11.9 Output Menu ... 1056
11.9.1 Passive Antenna ... 1056
11.9.2 Observation Directions ... 1057
11.9.3 Observation Points ... 1058
11.10 Meshing Menu ... 1068
11.10.1 Create Mesh ... 1068
11.10.2 Visualize Existing Mesh ... 1075
11.10.3 View Log ... 1075
11.11 Calculate Menu ... 1076
11.11.1 Execute ... 1076
11.12 Show Results ... 1078
11.12.1 Far Field ... 1078
11.12.2 Radiation Pattern ... 1086
11.12.3 View Currents ... 1096
11.12.4 View Charges ... 1097
11.12.5 Near Field ... 1098
11.12.6 View Radiation Map ... 1109
11.12.7 View Delay Process ... 1111
11.12.8 Export Multipole File ... 1114
11.12.9 Antenna Impedance ... 1115
11.12.10 S-Parameters ... 1118
11.12.11 Insertion Losses ... 1122
11.12.12 Boresight Error ... 1123
11.12.13 Post-Process ... 1123
11.12.14 Amplitude-Phase Point Results ... 1124
11.13 Tools Menu ... 1132
11.14 Help Menu ... 1133
11.15 Training Examples ... 1134
11.15.1 Example 1: Common Simulation ... 1134

11.15.2 Example 2: Common Parametric Simulation with Frequency Sweep 1145
11.15.3 Example 3: Radiation Map Simulation with Frequency Sweep 1164
11.15.4 Example 4: Delay Process Simulation with Frequency Sweep 1177
11.15.5 Example 5: Multipole Examples 1189
11.15.6 Example 6: Common Simulation with a Coaxial Feed Source 1209
11.15.7 Example 7: Basic Reflectarray Simulation 1222
11.15.8 Example 8: S-Parameters 1233
11.15.9 Example 9: Parameters, Frequency Sweep, Far and Near Field 1243
11.15.10 Examples with Different Pointing Algorithms 1257
11.15.11 Corrugated Horn Examples 1323
11.15.12 Design of Waveguides 1357
11.15.13 Reflectarrays 1374
11.15.14 Radomes 1384
11.16 Test Cases 1439
11.16.1 Test Case A 1439
11.16.2 Test Case B 1449

12 Periodical Structures 1464

12.1 File Menu 1465
12.2 Edit Menu 1466
12.3 View Menu 1467
12.4 Geometry Menu 1468
12.5 Materials Menu 1469
12.6 Cell Menu 1470
12.6.1 Define Cell 1470
12.6.2 FSS Primitives 1471
12.6.3 Import 1471
12.6.4 Export 1471
12.7 Simulation Menu 1472
12.7.1 Simulation Parameters 1472
12.8 Solver Menu 1473
12.8.1 Solver Parameters 1473
12.9 Meshing Menu 1475
12.9.1 Create Mesh 1475
12.9.2 Visualize Existing Mesh 1480
12.9.3 Visualize Mesh Log 1481
12.10 Calculate Menu (Alt + C) 1482
12.10.1 Execute 1482
12.11 Show Results Menu 1484
12.11.1 View Cuts by Frequency 1484
12.11.2 View Cuts by Step 1485
12.11.3 View Text Files 1485
12.11.4 Export Database 1485
12.11.5 Export Feko TR File 1486
12.11.6 View Database 1487
12.12 Tools Menu 1488

12.13 Help Menu 1489
12.14 Training Examples 1490
12.14.1 Example 1: Analysis of a Cell with Disk Geometry 1490
12.14.2 Example 2: Analysis of a Cell with Spiral Geometry and Skew Angle 1495
12.15 Annex 1: Creating a Reflectarray Database 1501
12.16 Annex 2: Analysis of a Reflectarray Database Creation 1505

13 Parabolic Equation 1513

13.1 File Menu 1514
13.2 Edit Menu 1515
13.3 View Menu 1516
13.4 Terrain Menu 1517
13.4.1 Import Terrain 1517
13.4.2 Delete Terrain 1519
13.5 Profiles Menu 1520
13.5.1 Extract Topographic Profiles 1520
13.5.2 Topographic Profile Editor 1521
13.6 Simulation Cases Menu 1524
13.6.1 Simulation Case Editor 1524
13.7 Calculate Menu 1527
13.8 Results Menu 1528
13.8.1 Visualizing Results 1528
13.9 Annex 1: PE Module Commands 1531
13.9.1 importTerrain command 1531
13.9.2 deleteTerrain command 1532
13.9.3 profileBetweenPoints command 1532
13.9.4 radialProfiles command 1532
13.10 Training Examples 1534
13.10.1 Example 1: Propagation Factor on an Existing Terrain 1534

14 Ultrasound 1540

14.1 File Menu 1541
14.2 Edit Menu 1542
14.3 View Menu 1543
14.4 Geometry Menu 1544
14.5 Materials Menu 1545
14.6 Simulation Menu 1546
14.6.1 Parameters 1546
14.6.2 Doppler 1549
14.7 Source Menu 1551
14.7.1 Ultrasound Pattern 1551
14.7.2 Import DUS File 1553
14.7.3 Edit Source 1553
14.7.4 Delete Source 1553

14.7.5 DUS File Editor....1554
14.7.6 Source Display Size....1555
14.8 Output Menu....1556
14.8.1 Observation Points....1556
14.8.2 Coupling....1565
14.9 Meshing Menu....1567
14.9.1 Create Visibility Matrix....1567
14.10 Calculate Menu....1568
14.11 Show Results Menu....1569
14.11.1 View Ray Tracing....1569
14.11.2 View Ray Tracing Text File....1570
14.11.3 View Text Files....1571
14.11.4 Near Field....1572
14.11.5 Coupling....1582
14.11.6 Doppler Spectrum....1584
14.12 Tools Menu....1587
14.13 Help Menu....1588
14.14 Training Examples....1589
14.14.1 Example 1: Getting Started....1589
14.15 Annex 1: DUS File Format....1598
14.15.1 REV DUS Files....1598
14.15.2 3D DUS Files....1599
14.16 Annex 2: Closed-Loop Simulation Support in the US Module....1600
14.16.1 Simulation Workflow....1600
14.16.2 Command Reference....1602

15 List of Acronyms and Abbreviations....1611

15.1 Acronyms....1612

Index....1613

Utilities

1

The newFASANT utilities allows you to set up a remote server, use the batch utility to mesh and execute a project or upload a project to a queueing system.

This chapter covers the following:

- 1.1 Remote Server (p. 24)
- 1.2 Batch Meshing and Execution of Projects (p. 27)
- 1.3 High Performance Computing (HPC)/Queueing (p. 29)

1.1 Remote Server

A remote newFASANT server can be set up to execute a project remotely.

By default, newFASANT is run locally on your own machine, meaning that the kernels used is provided with the software installed and must have an active license for them.

With the Client-Server mode, you can define a new remote server where newFASANT software must be installed and must have an active license for the installed kernels. It requires setting up a connection to the server indicating its network route (IP address) and the port to which it will connect. Once the connection is established, the client sends data to the server, processing it and returning the results to the client.

To get a newFASANT server up and running, you must launch the newFASANT executable within the installation alongside the `-server` argument. A port number can be specified after the `-server` argument, by default, it will use 3107 (no additional arguments are needed).

For example:

```
C:\Program Files\Altair\2021\feko\bin>newfasant.exe -server
```

To create a remote connection, you only need to set up the connection profile. Click **Tools** > **Remote**.

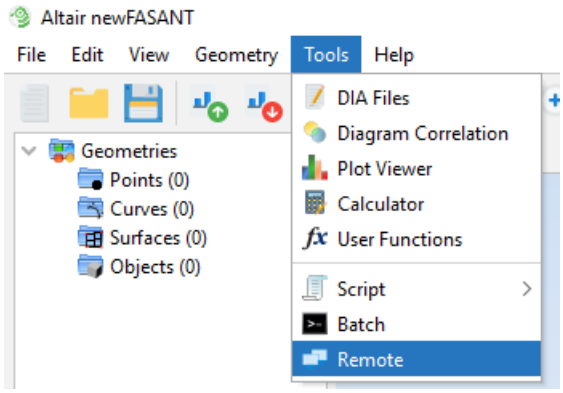


Figure 1: ***Remote*** *menu*

And the following panel is displayed:

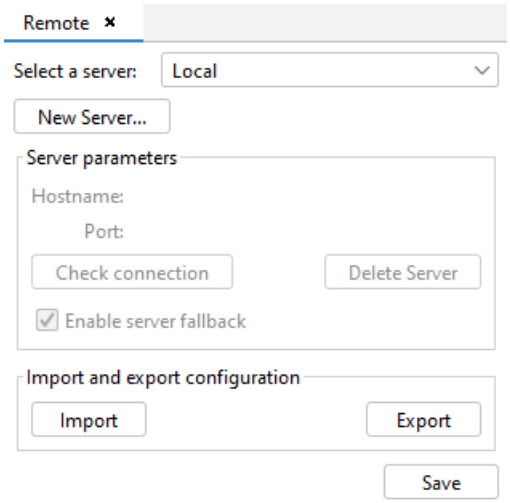


*Figure 2: **Remote** panel*

Click **New Server** to open the panel to create the connection with the new server.

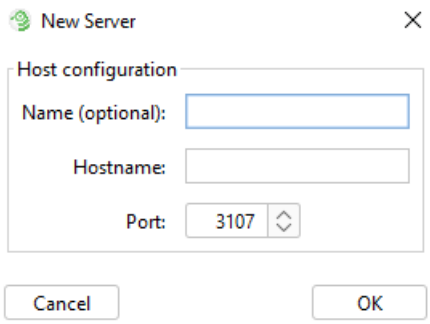


*Figure 3: **New Server** panel*

- **Name (optional)**: Identification for the new server to show on the server list in the main tab.
- **Hostname**: IP address of the server.
- **Port**: Listening connections port.

Then click **OK** to select the new remote server.

Complimenting this form, return to the **Remote** panel where the specified server can be selected in the **Select a server** drop-down list. By default, **Local**, means that the current machine is used as both client and server.

To ensure that the selected profile is the correct one and that the connection is available, click **Check Connection**. In case the check is successful, the following dialog appears:

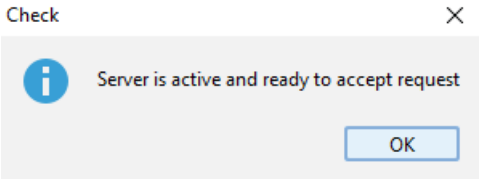


Figure 4: Success message

If the server is configured incorrectly or has a problem, the next message will appear:

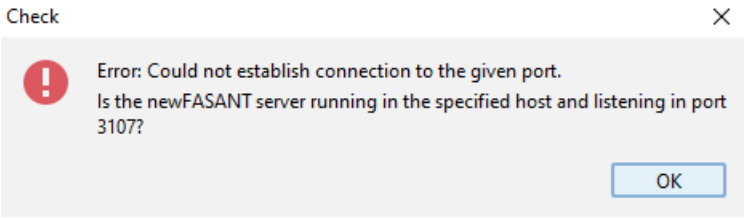


Figure 5: Failure warning

- **Delete Server**: Allows you to delete any server in the list of stored servers.
- **Enable server fallback**: This option checks all the servers on the list to see if it is active.
- **Import**: This option loads the configuration information about the servers from a selected file.
- **Export**: This option saves the configuration information about the servers to a selected file.

Save all changes by clicking **Save**.

1.2 Batch Meshing and Execution of Projects

newFASANT supports two methods of meshing and running simulations from the command line or terminal, allowing you use the mesher and kernels without launching the newFASANT GUI.

Using the Batch Utility

To specify the number of processors, threads and the different MPI options, you must set up the following environment variables:

- *NF_PROCS*
  Overwrites the number of MPI tasks stored in the project. In Linux, it works similarly but also can be filled with `--use-job-scheduler` to call feko_parallel with this parameter.

- *NF_THREADS*
  Overwrites the number of OpenMP threads stored in the project. It works similarly in Linux.

- *NF_MPI_OPTIONS*
  Defines additional options for MPI. In Windows, these options are supplied as an input parameter for MPIexec. Linux works in a similar way, but instead of supplying them to MPIexec it goes to feko_parallel[1].

Launch the newFASANT batch utility using the command:

```
newfasant <"project file path"> [options]
```

project file path The path to the project file inside “”.

The command line options are:

-mesh Mesh the project.

For example:

```
newfasant "/home/newfasant/MyProject.nfp" -mesh
```

-calculate Execute the project.

For example:

```
newfasant "/home/newfasant/MyProject.nfp" -calculate
```

-update Mesh and execute the project.

For example:

```
newfasant "/home/newfasant/MyProject.nfp" -update
```

1. See the Feko User Guide for more information: **Running the Parallel Version**.

For more information on how to use the batch utility:

```
newfasant_batch.exe -h
```

Using Runfeko

Another way to launch newFASANT from a command line or terminal is by using the runfeko utility. The syntax for launching newFASANT through runfeko is:

```
runfeko FILENAME [OPTIONS]
```

FILENAME	The path of the `.nfp` project (with or without extension).
--version	print the version information and then exit
-np x	start the parallel newFASANT version with x processes (special case "-np all" uses all available from the machines file)
--use-job-scheduler	run the parallel newFASANT Solver within a queuing system (see documentation which are supported) and obtain the number of parallel processes as well as the host list directly from the respective job scheduler
--feko-options ...	all following options will be passed to newFASANT
--num-threads x	x is the number of threads to be used. This option is only supported for OMP and HYBRID solvers

Examples:

To use 4 processes and 4 threads for MPI to solve a project located at "c:/Temp/horn.nfp":

```
runfeko C:/Temp/horn -np 4 --feko-option --num-threads 4
```

To use the PBS resources and 4 OMP threads to solve a project named "horn.nfp":

```
runfeko horn.nfp --use-job-scheduler --feko-options --num-threads 4
```

1.3 High Performance Computing (HPC)/Queueing

newFASANT offers the possibility to work with a queueing system such as Altair PBS Professional, LSF or Slurm.

This feature allows you to upload your projects to a cloud queue, and the system decides in which node of the cluster the simulation will run.

To enable it, go to **Simulation** > **HPC Parameters** and the following panel appears:

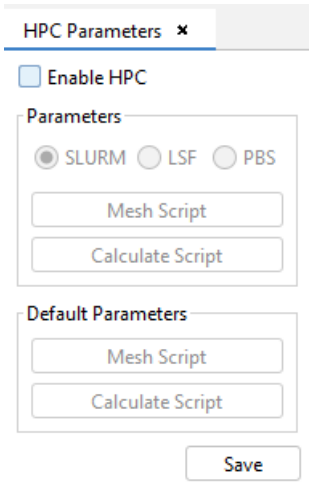


*Figure 6: **HPC Parameters** panel*

By selecting the **Enable HPC** check box, the rest of the options are enabled. Select the queueing system (SLURM, LSF, PBS).

The script that is launched can be modified under **Parameters**.

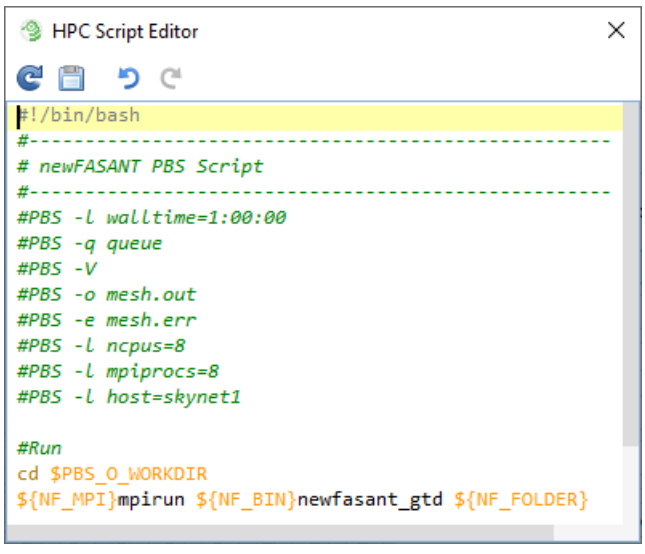


Figure 7: HPC Script Editor

In all HPC scripts, there are three common parameters:

NF_BIN
The directory that contains all executables.

NF_INPUT
The input file for the kernel.

NF_MPI
The path to the MPI executable.

The above parameters are automatically set up by the GUI. If you want to revert modifications to these scripts, it can be done under**Default Parameters**, which contains the original, unmodified scripts. To save the changes, click **Save**.

GUI

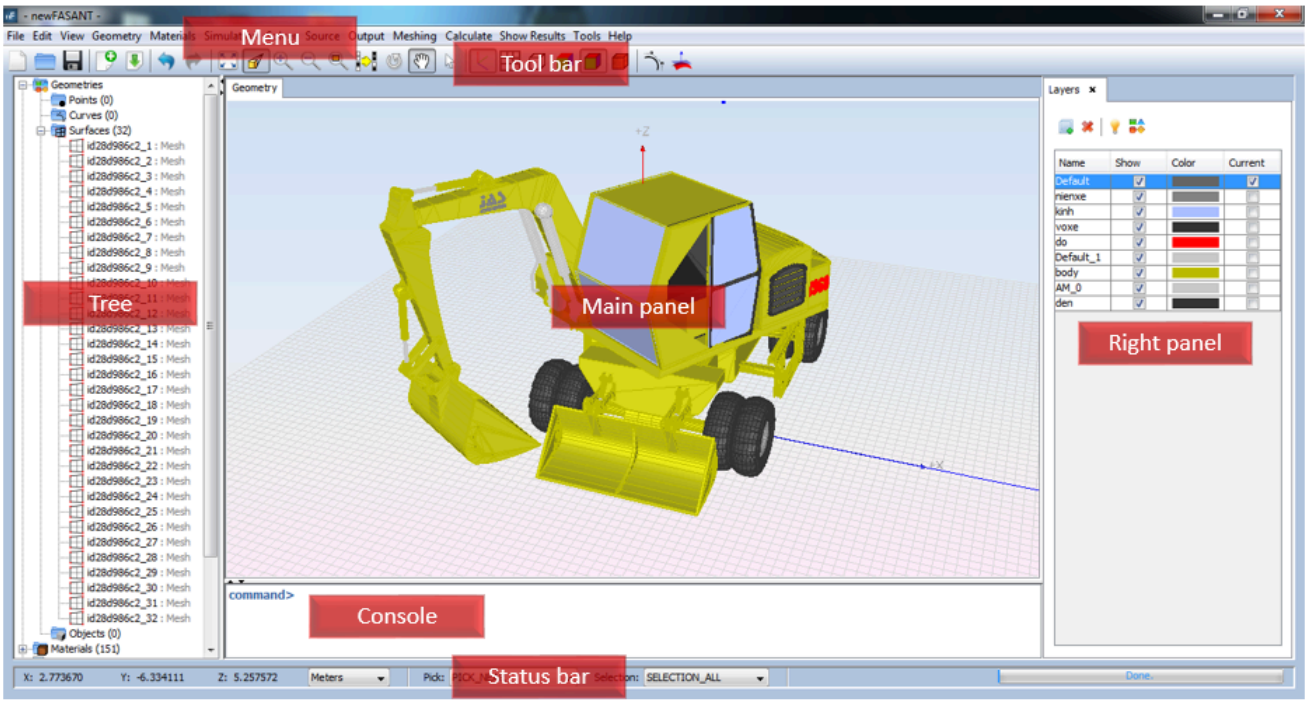
2

An overview of the newFASANT graphical user interface is given.

This chapter covers the following:

- 2.1 GUI Overview (p. 32)
- 2.2 File Menu (p. 47)
- 2.3 View Menu (p. 53)
- 2.4 Edit Menu (p. 62)
- 2.5 Geometry Menu (p. 95)
- 2.6 Tools (p. 210)
- 2.7 Help Menu (p. 258)
- 2.8 Annex 1: Graphics Advanced Options (p. 259)
- 2.9 Annex 2: Inverting Normals (p. 269)

2.1 GUI Overview



- **Main panel:** Multi-tab panel with the geometry model, mesh and different results.
- **Right panel:** Multi-tab panel where different windows are displayed.
- **Menu:** Includes all options and geometry commands.
- **Console:** Its used to create or edit geometry.
- **Tree:** Provides quick access to geometry and simulation parameters.
- **Tool Bar:** Contains the common application functions.
- **Status Bar:** Contains information and geometry options.

2.1.1 Console

The newFASANT command console is a command-line shell which supports both interactive and batch-file commands to manage geometry. The command line supports geometric construction with the mouse. All commands are accessible from menu.

```
command> box
First corner of base [x y z]: 0.0 0.0 0.0
Base size [width depth]: 1.0 1.0
Height [double]: 1.0
command>
```

Figure 8: Console panel

The basic usage of the command line is explain on 'help' command:

```
command> help
Help Usage: Type 'help <command>' or '<command> -h' to show help about a specific command.
Commands:
  - Type a command and press ENTER to execute it.
  - Press ESC to abort a command.
  - Press TAB to complete the written pattern name with the coincident commands.
  - Use the mouse to pick points or draw the geometry directly.
  - Press Ctrl+Left in a command step for coming back to the previous one.
History:
  - Use Up/Down arrows to show the commands used recently.
  - Use Ctrl+Z or Ctrl+Y for Undo/Redo.
  - Go to Edit->History to modify previous operations or delete specific commands.
Ctrl+L: Clear the console panel.
```

Figure 9: 'help' command

You can view the Command Line to see a description of each command.

2.1.2 Tree

The left side of the main window features the **tree panel**, where the **tree** can be found

The purpose of the tree is to provide a simple and global view of the parameters of the current project, as well as an alternative way to modify these parameters.

The following figure shows the tree for a Moncros project:

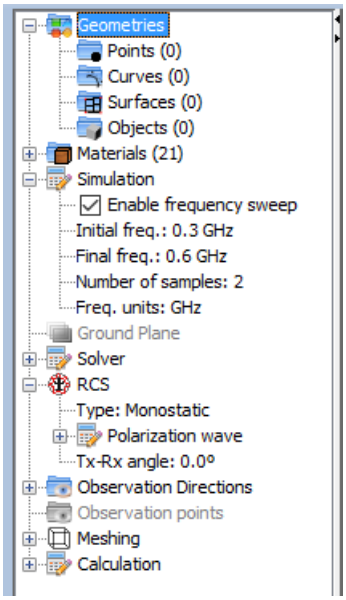


Figure 10: Example tree of a Moncros project

The tree is composed of **nodes**, which are laid out in a hierarchical structure. This means that some nodes can have children nodes that represent related project parameters. A node that owns at least one children is called a **parent node**, and its children can be hidden (also known as **collapsing** the parent node) or shown (also known by **expanding** the parent node) by clicking the "-" (minus) sign to the left of the node to hide its children or the "+" (plus) sign to the left of the node to show its

children. Additionally, it is possible to double-click on the node to toggle its state between collapsed and expanded.

Some nodes provide functions that are not visible at first glance, as they may be provided through a **contextual menu** of the node. Showing the contextual menu for a node is as simple as right-clicking that node. Note that most nodes don't have a contextual menu.

Regarding to the contents of the tree, there's a common node that is displayed in every project (module independent) the **geometry node**. Additionally, every module provides its own set of nodes that represent the parameters of a project from that module. However, the user interface of the tree is the same across all modules and the different ways of editing project parameters is explained in the following subsections.

Managing Geometry Objects

The geometry tree node is fixed to the top of the tree panel and it's always present. This node displays all the geometry objects of the current project, as can be seen in the following figure:

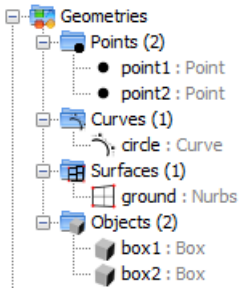


Figure 11: Geometry node displaying some existing objects

The objects are grouped regarding to its type (points, curves, surfaces or 3D objects) and, for each object, a name and a more specific type (if available) are shown. It is possible to change the name of an object by selecting it and pressing enter, or by triple-clicking on it. Doing any of these actions will put the node in "editing state", in which the new name for the object can be written:

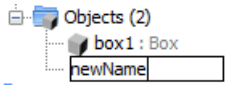


Figure 12: Changing the name of an object

When the new name for the object has been entered, the Enter key needs to be pressed in order to exit the editing state and actually assign the new name to the object.

It is easy to see which object a node corresponds to by clicking on it on the tree. This will select the object globally and the user will be able to see the selected object in the geometry panel, as it will be coloured in yellow.

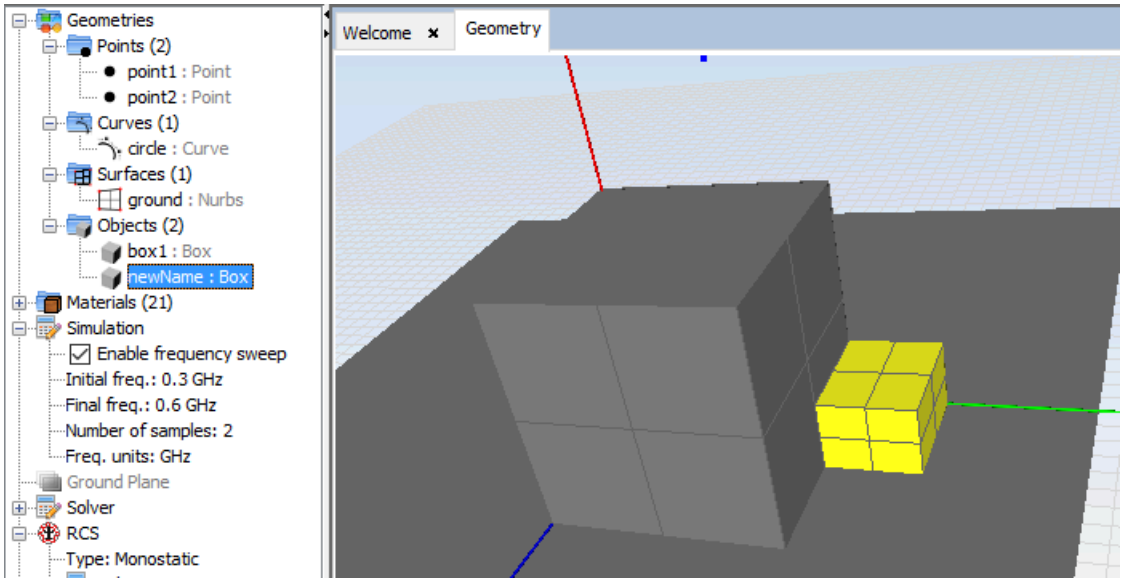


Figure 13: Viewing a selected object in the geometry panel

The user can also perform several other operations on the objects of the geometry besides renaming them. For this, they first need to right-click on the node. This action will cause a popup menu to show, giving the user the option to select one of various geometric operations:

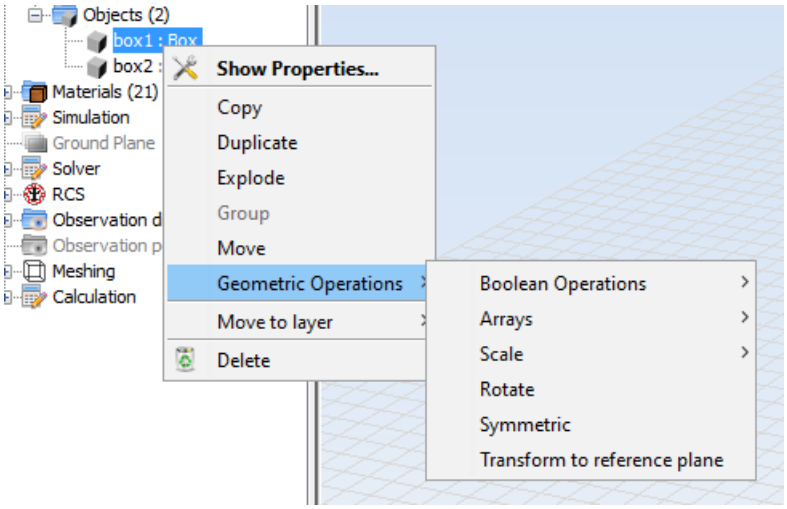


Figure 14: Contextual menu showing the different operations that can be performed on an object

Selecting any of these options will call the corresponding command on the selected objects. For a detailed reference about how these operations work, consult Edit Menu.

For operations that can work with more than one object at a time (like, for example, the **copy** or **move** operations), it is possible to select several objects by holding the **Ctrl** key and clicking on each node. Additionally, it is possible to select a consecutive set of nodes on the tree by selecting the first one and then selecting the last one while holding the **Shift** key. Selecting a operation on the contextual menu while there are several selected objects will run the operation on all the selected objects.

It is also possible to select one or more objects and change the layer they are on. To do this, call the context menu and put the mouse over the “Move to Layer” option. A list of all the layers will be revealed and the user may click one of them to move the selected objects to that layer.

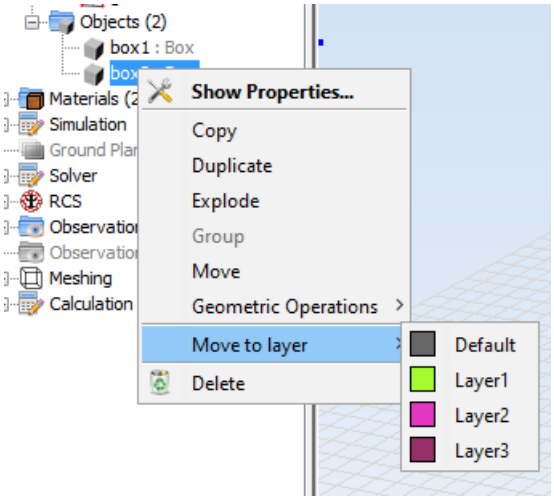


Figure 15: Contextual menu showing the "Move to Layer" option

Nodes on a hidden layer are grayed-out in the tree, and cannot be selected by clicking in their parent node. For example, in the next figure, if the user clicks on the “Objects (2)” node, only the “box1” object will be selected. This is by design, as it prevents the user from doing operations (accidentally) in objects that are not visible to them. However, if the user clicks on the grayed-out node directly, it will still be selected.

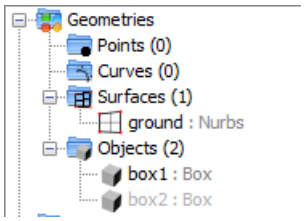


Figure 16: Example of a node on a hidden layer

Finally, it is possible to delete an object or set of objects by selecting them on the tree and pressing the **Delete** key on the keyboard (as well as selecting the “Delete” option in the contextual menu).

Editing Project Parameters

As mentioned in previous sections, each module provides each own tree structure to modify the parameters of a project. However, the ways the user can interact with the tree are very similar across all modules.

The nodes below the geometry node are the nodes that are specific to the project and they usually give the user the option to visualize every parameter of the project in a hierarchical and concise way. Additionally, each parameter can be modified in one of the following ways:

- **On/off parameters** Some parameters can only be in one of two states enabled or disabled. These parameters are represented in the tree with a checked or unchecked box, respectively, and its state can be toggled between the two states by clicking on them, or by pressing the Enter key while the node is selected.

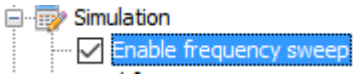


Figure 17: Tree node with a check box

- **Numerical parameters** Some other parameters have a numerical (integer or real) domain. In these cases, the parameter can be modified by triple-clicking on the node, or by pressing Enter while the node is selected. This will put the node in "edit state", where the new value for the parameter can be inserted. Once the new value has been written, the user needs to press Enter to confirm this value. If the input value is not valid for the parameter, a error dialog will be shown.

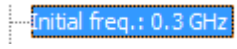


Figure 18: Tree node that represents a numerical value (view state)

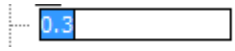


Figure 19: Tree node that represents a numerical value (edit state)

- **Multiple choice parameters** In some cases, the possible values for a parameter belong to a small set of values. To modify these values from the tree, the user can triple-click on a node to display a dropdown menu with every possible value for the parameter, and they can select one of these values to assign it.

  Alternatively, the user can press the Enter key while the node is selected to put the node in "editing mode" and press the up and down arrow keys to switch between the possible values. Once the value is selected, the user can press the Enter key to confirm the selection.

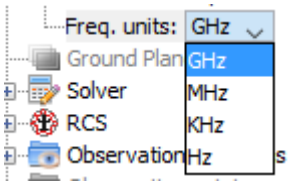


Figure 20: Editing tree node with a predefined set of values

- **Drag and drop** In some cases, it is possible to drag a node from the tree and dropping it over another node to carry out an action. Examples of this feature include material assignation to surfaces and 3D objects (in which the user has to drag a material node and drop it over a geometry object), material assignation to a ground plane in the Moncros module, or even adding Doppler and Creeping to objects in the PO and GTD modules, respectively. The different uses for this feature will be explained in the following section, " **Drag and drop**".

These types of nodes are usually the leaf nodes of the tree. In addition to this, there can be grayed out nodes in the tree that represent groups of configuration parameters that are not available at the

moment due to diverse reasons (for example, because these parameters wouldn't apply to the currently selected simulation type, or because the functionality is not enabled):

Figure 21: Example of a disabled tree node

Usually, double-clicking these nodes will show a message dialog describing the reasons of why the node is disabled and, if possible, asking the user if they want to enable the functionality associated with the node:

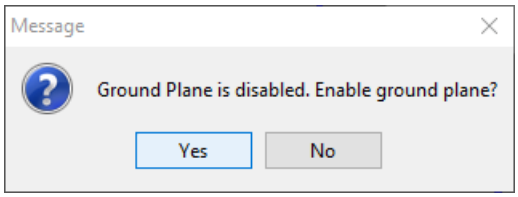


Figure 22: Dialog window asking the user to activate the functionality

If the user answers affirmatively to this question, the functionality will be enabled. In consequence, the tree node will be activated and it will show the corresponding configuration options:

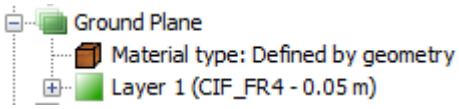


Figure 23: Example of an enabled node

Drag and Drop

Some tasks can be simplified using the "Drag and Drop" feature of the tree. This feature allows the user to perform certain actions by clicking and dragging a node of the tree into another node of the tree.

The uses of this feature are detailed below:

Material assignation to a surface or object

In every module, it is possible to assign a material to a surface or 3D object by just clicking and dragging a material node into a geometry node in the tree.

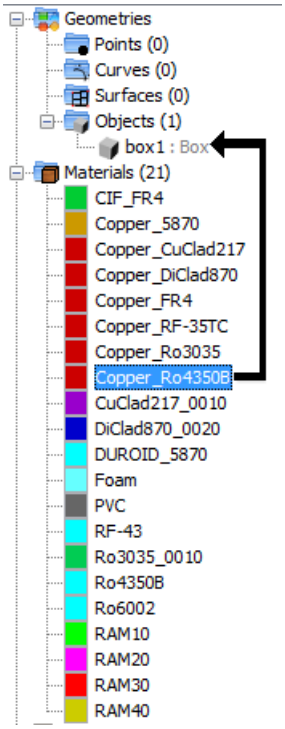


Figure 24: Drag and drop process of assigning a material to an object

Doing this will assign a layer to the object with a default thickness. If the user wants to further configure the layers of an object, they need to use the “Assign” menu option of the “Materials” menu, available in most modules.

Material assignation to a ground plane layer

In the PO and Moncros modules, it is possible to assign a material to a ground plane defined by geometry. To do this, the user needs to drag a material node and drop it into the node that represents the ground plane layer. The user can modify the thickness of each layer from the tree.

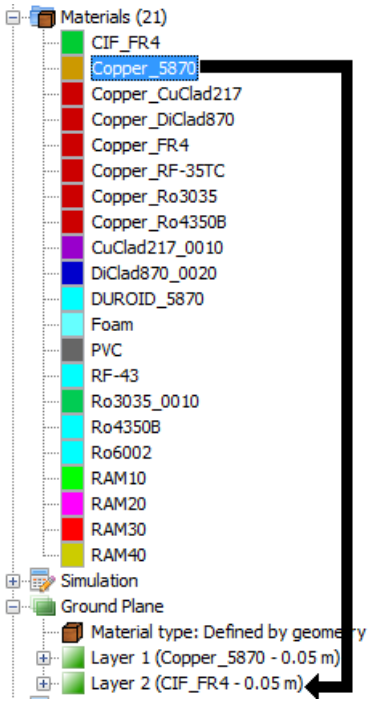


Figure 25: Drag and drop process of assigning a material to a ground plane layer

Creation of observation points from a curve or surface

In the Moncros, GTD and MOM modules, the user is able to extract points from a curve or surface to create a set of observation points used for the analysis of the near field. To do this using the tree, the user needs to drag a curve or surface node into the "Observation points" node. Doing this will show a dialog prompting the user to specify the number of points that will be sampled from the selected curve or surface.

Please, note that:

- In order to be able to add a set of observation points in Moncros, bistatic RCS needs to be enabled.
- To add a surface of observation points, it is possible that the user needs to explode the object that represents the surface, so that it appears in the "Surfaces" folder of the tree.

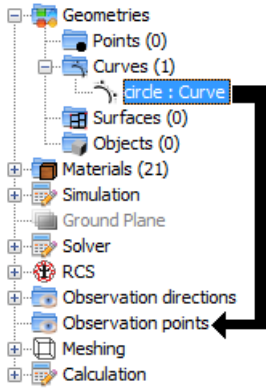


Figure 26: Drag and drop process of adding a set of observation points from a curve

If the user drags a curve node into the "Observation points" node, the following dialog will appear:

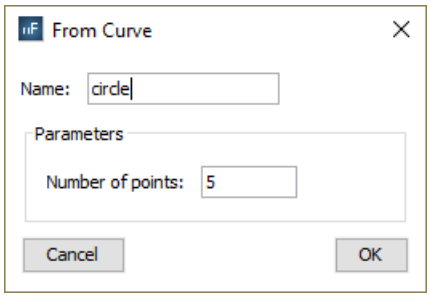


Figure 27: Dialog asking the user how many point samples to extract from the curve

In the other hand, the following dialog will appear if the user does the same with a surface node. In this case, it is necessary to specify the number of points to sample in each direction of the parametric surface.

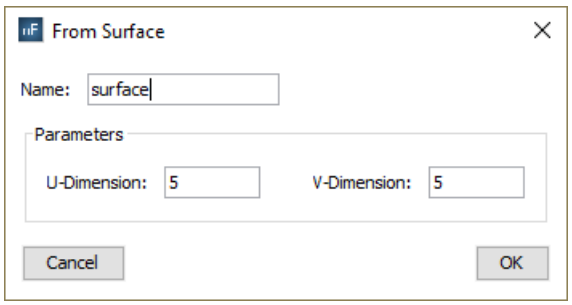


Figure 28: Dialog asking the user how many point samples to extract from the surface, for each parametric direction

When the user presses the "OK" button, the observation points will be added.

Duplicate normals of an object in the PO module

In the PO module, the user can duplicate the normals of an individual surface or all the surfaces of an object by clicking and dragging the object or surface and dropping it in the "Duplicated normals" node.

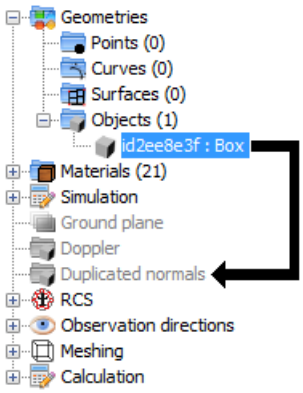


Figure 29: Drag and drop process of adding a set of observation points from a curve

The list of objects and surfaces with duplicated normals can be found under the "Duplicated normals" node.

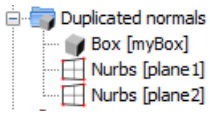


Figure 30: Tree node that shows the list of objects with duplicated normals

In order to cancel the normals duplication of a surface or object, the user needs to right-click the corresponding node and select the "Don't duplicate normals" option of the popup menu.

Set creeping to an object or surface

In the GTD module, it is possible to add creeping behaviour to an object or surface by just clicking and dragging a object node and dropping it into the "Creeping objects" node. Please nota that the "Creeping" simulation option must be enabled before doing this.

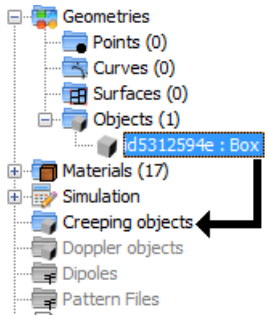


Figure 31: Drag and drop process of adding creeping to an object

The children nodes of the "Creeping objects" node represent the objects that have creeping enabled at the moment. To remove creeping from any of these objects, the user needs to right-click the node and select the "Remove Creeping" option.

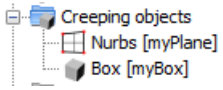


Figure 32: Tree node that shows the list of objects with creeping

Set Doppler parameters to an object

In the PO and GTD modules, the user is able to add doppler parameters to an object. This task can also be done with drag and drop by clicking and dragging the object node and dropping it into the "Doppler" node.

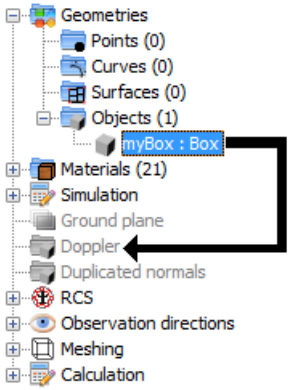


Figure 33: Drag and drop process of adding doppler to an object

The children nodes of the "Doppler" node will represent all the objects with doppler in the current project. This node will also allow the user to modify the doppler parameters for each object (i.e the translation parameters and the rotation parameters, as well as the global parameter "frequency bin").

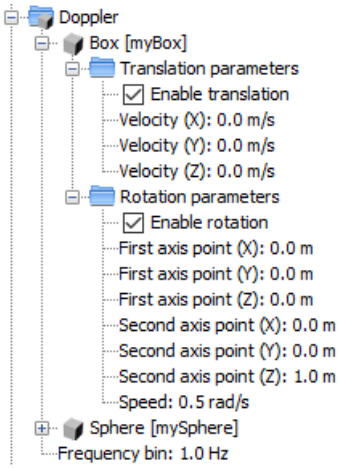


Figure 34: Tree node that shows the list of objects with doppler

To delete doppler of an object, the user needs to right-click the corresponding child node of the "Doppler" node and select the "Delete doppler" option from the contextual menu.

2.1.3 ToolBar

Figure 35: ToolBar Panel

The ToolBar panel contains the most common options, it is divided into 5 sections:

Project options:

- New project -> Creates a newFASANT project.

- Open project -> Open an existing file project.
- Save project -> Save the current project.

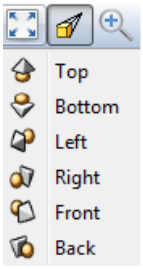
Geometry file options:

- Import file -> Import a geometry file.
- Export file -> Exports the geometry.

- Undo (CTRL+Z) -> The undo option is used to undo the previous change.
- Redo (Ctrl+Y) -> If the user has undone any changes it is possible to redo them with this option.

View options:

- Reset view (CTRL+R) -> This option resets the view settings of the Geometry view area to the default values. If you click with the right button can select the view.



- Perspective / Parallel projection -> Sets the perspective or parallel projection.
- Zoom in -> This option zooms in the current view of the Geometry.
- Zoom out -> This option zooms out the current view of the Geometry.
- Zoom selected -> This option zooms into the selected area of the Geometry.
- Invert selection -> This option inverts the geometry selection.
- Rotate -> This option allows to rotate camera with the left mouse button.
- Pan -> This option allows to translate camera with the left mouse button.
- Select -> This option allows to select objects with the left mouse button.
- Show axis -> This option toggles the visualization of the three axes in the view area.
- Reference plane -> Shows the reference plane panel.
- Lines -> Toggles render mode to lines.
- Shaded -> Toggles render mode to shaded.
- Shaded with lines -> Toggles render mode to shaded with lines.
- Material / Layer view -> Shows the color of the material or the layer.

- Control Points -> Shows the control points of a selected NURBS surface.

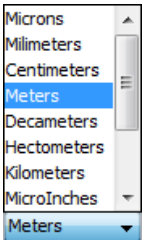
- Invert Normals -> View and invert normals of a selected objects. To invert normals click on the view area.

2.1.4 StatusBar

Figure 36: StatusBar panel

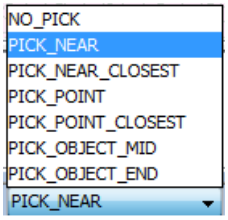
Current Point: Shows the mouse point selected. If pick is enabled you can view this point on the geometry panel with blue color.

Units: This combo allows to change the model units. If the panel contains geometry you can scale it to keep the size.



Pick: This option enables you to select a geometry point with the mouse. Contains seven pick modes.

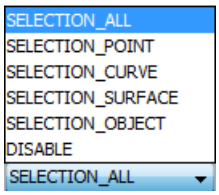
- **NO_PICK** The selected point is the transformation of the mouse position into 3d coordinates.
- **PICK_NEAR** or **PICK_NEAR_CLOSEST** The selected point is the nearest point of the geometry or reference plane. PICK_NEAR only gets the point if its near of the mouse.
- **PICK_POINT** or **PICK_POINT_CLOSEST** Returns the coordinates of the nearest point object. PICK_POINT only gets the point if its near of the mouse.
- **PICK_OBJECT_MID** If the object is an object3D selects the geometric center. If the object is a surface or a curve selects the mid point.
- **PICK_OBJECT_END** Returns the corners of the object that is closest to the mouse.



Selection: Contains the geometry selection options. This changes the mouse selection and rectangle selection.

- **SELECTION_ALL:** All objects can be selected (points, curves, surfaces and objects).
- **SELECTION_POINT:** Only points can be selected.

- **SELECTION_CURVE:** Only curves can be selected.
- **SELECTION_SURFACE:** Only surfaces can be selected.
- **SELECTION_OBJECT:** Only objects can be selected.
- **DISABLE:** Disable selection.



2.1.5 Multi-Tab

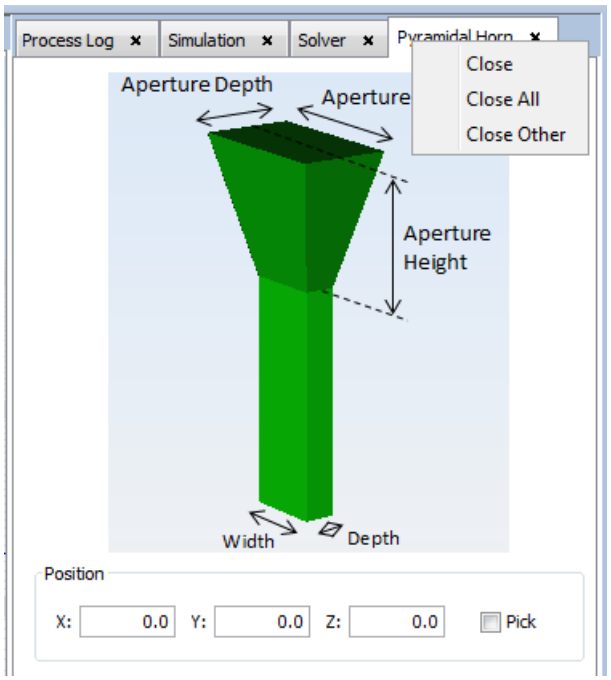


Figure 37: Right panel

Clicking on the right button in a tab a popup menu appears with the following options:

- **Close:** Close the selected tab.
- **Close All:** Close all tabs on the panel.
- **Close Other:** Close all tabs except the selected tab.

2.2 File Menu

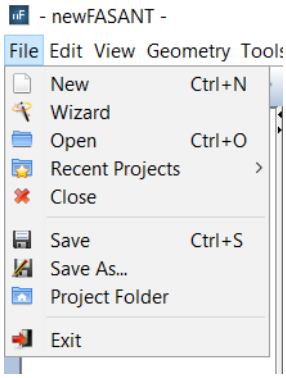


Figure 38: File menu

The options of the File Menu are explained below.

2.2.1 New (Ctrl+N)

This option allows the user to create a new project. The user have to select the desired module from the list that appears in the central area.

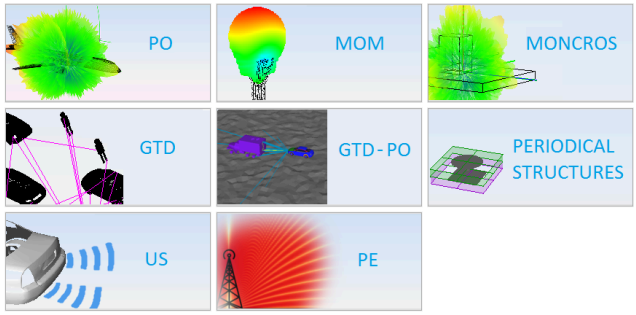


Figure 39: Method Type selection

The user may save the changes to the current working project before creating the new one. A dialog will ask the user to do that.

2.2.2 Wizard

Using the **project wizard** it is possible to create new projects using a template for a particular case. If the simulation conditions are known, they save time by setting the default values for a project, such as units, simulation frequency and number of processors to use during the meshing and simulation stages.

To create a new project using the template wizard, use the **New | Wizard** menu entry. The wizard window will open presenting the list of available project templates. Each template is based on a

particular solver and may use different parameters in order to optimize it for a particular kind of project. Select a template of your choice and press **Next** to continue to the next step.

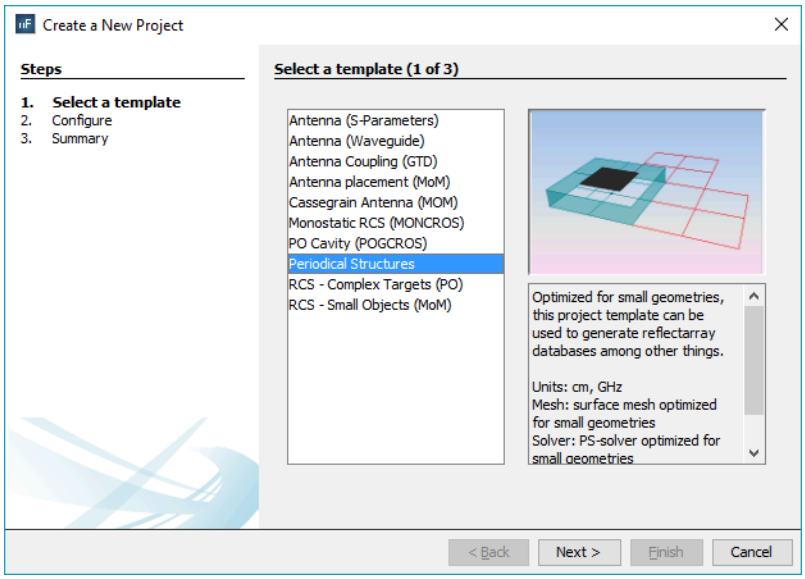


Figure 40: Step 1— Choosing a project template through the wizard.

The next step is to select what kind of project create based on this template. There are two possible choices:

- **Create a new empty project** use this option to create a brand new project using the optimized default parameters for the particular template selected in the previous step. It is still possible to modify some of the parameters using the controls below, such as the default units, the number of processors (a number that is set by default to the number of processors in your system), or the project simulation frequency.
- **Open a master project** using this option, a new project is bootstrapped from an example project, containing sample geometries and possibly other project features such as antennas or observation points. They can be used to test the meshing and simulation process, or they can be deleted once the project is created to replace them by your own geometries and project settings.

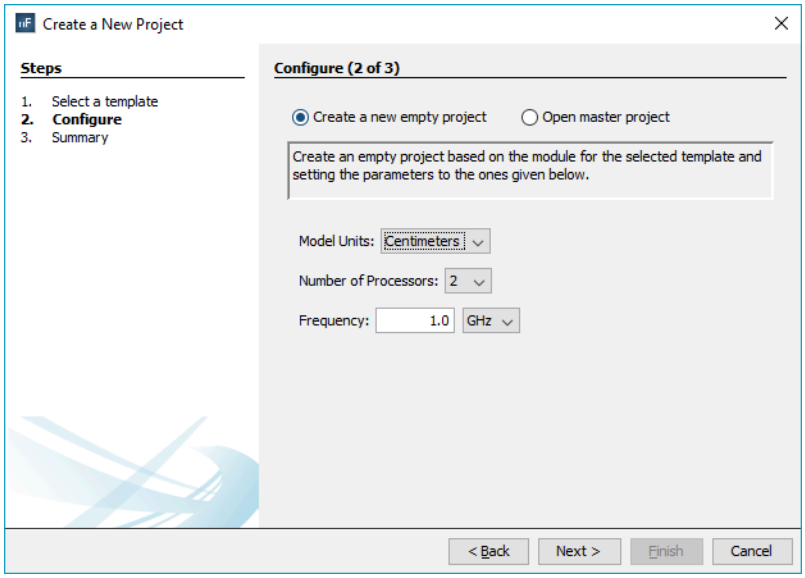


Figure 41: Step 2— Setting the new project parameters.

Once the selection is made, press **Next** to jump to the last step.

The last step presents a summary for the choices made in this project, including information about the template and the default values set by the template or changed by the user. Press **Finish** to confirm the selection and to create the project.

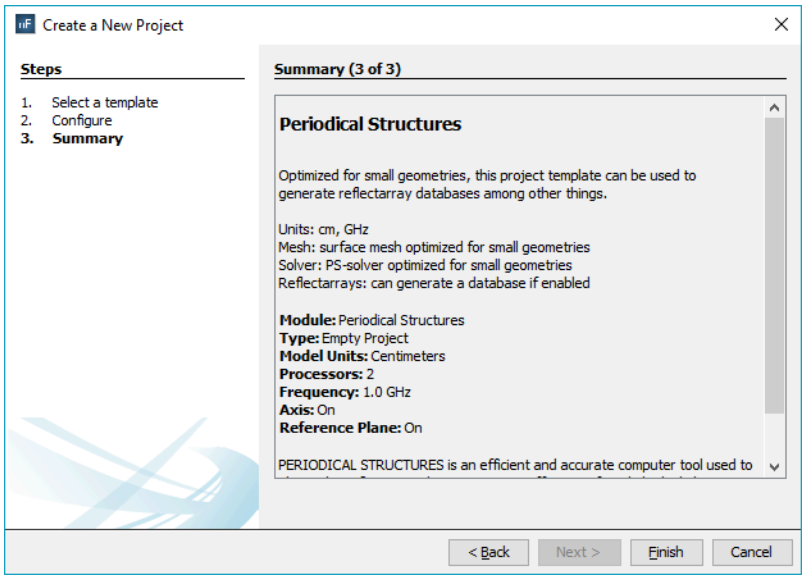


Figure 42: Step 3— Project summary.

2.2.3 Open (Ctrl+O)

This option allows the user to open an existing newFASANT project (`.nfp` file extension). A dialog asking whether the user wants to save the previous working project will be prompted.

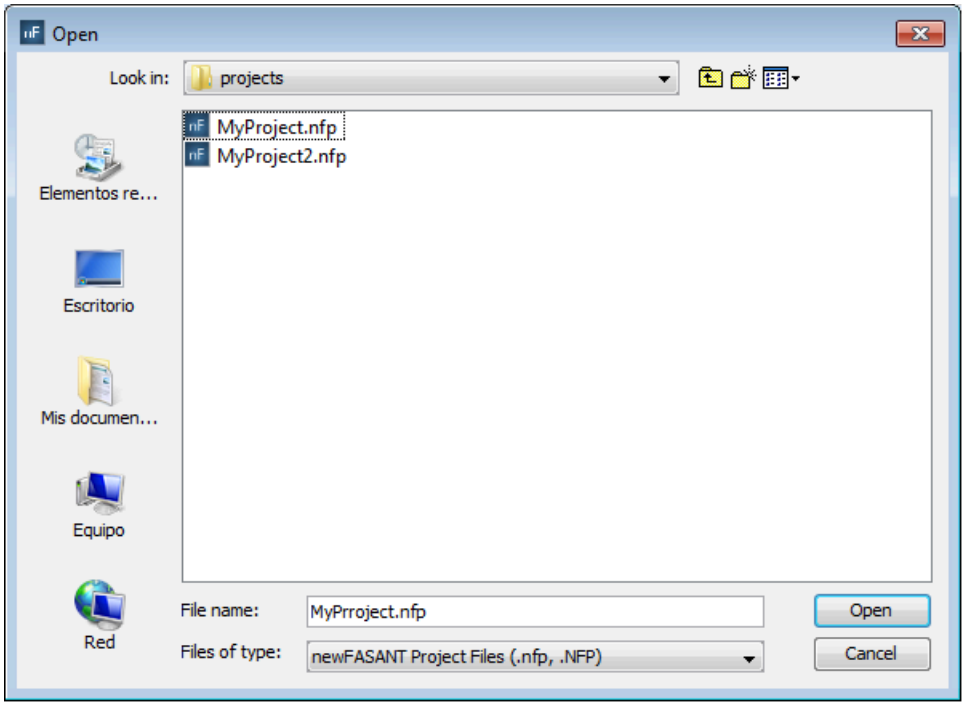


Figure 43: Open file dialog

2.2.4 Recent Projects

This menu lists the last projects recently used. The user can open the project immediately by clicking on any project of the menu list.

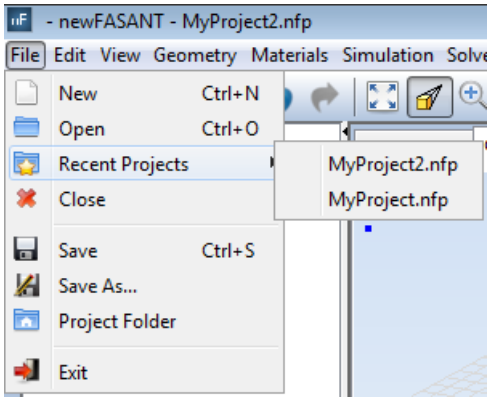


Figure 44: Open file dialog

2.2.5 Close

This option will close the current project. A dialog will offer the possibility to save the changes to the current project.

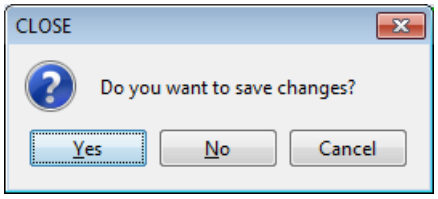


Figure 45: Dialog to save the changes

2.2.6 Save (Ctrl+S)

This option allows the user to save the current project. The first time that this option is selected, a save dialog will be shown asking the user for the location of the new newFASANT Project file (`.nfp`). The following times that the option is clicked on, the project will be saved in the previously chosen location.

2.2.7 Save As (Ctrl+S)

This option allows the user to save the current project in any location. When this option is selected, a save dialog will be shown requesting the location of the new newFASANT Project file (`.nfp`).

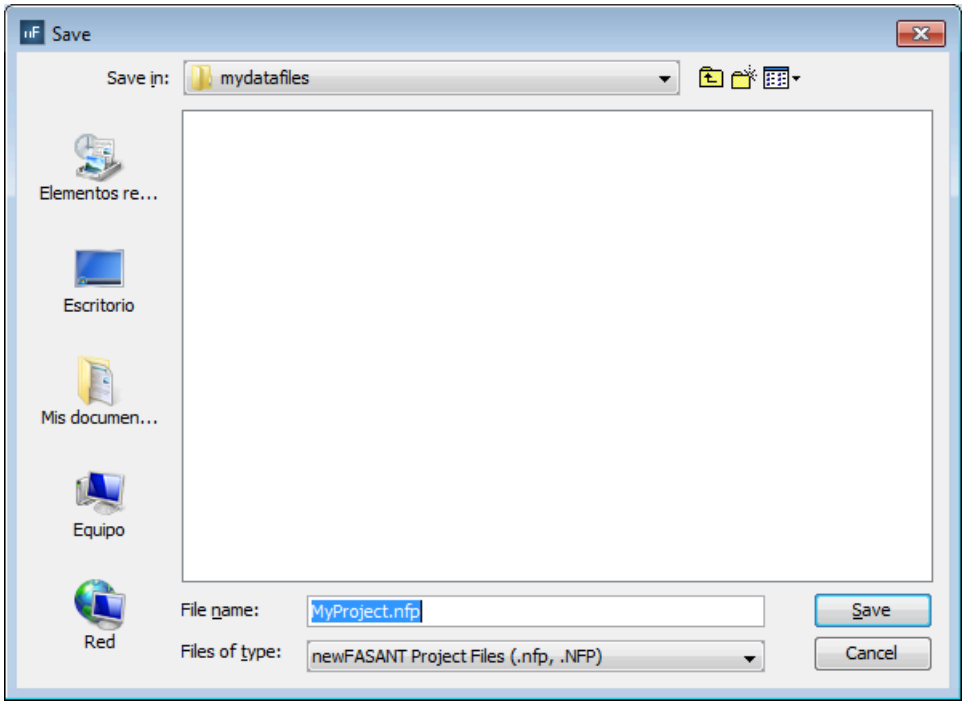


Figure 46: Save as dialog

2.2.8 Project Folder

This option shows the project location in your system. When opening a project, all the work performed will be stored temporally in this location.

A folder explorer window appears as shown in the following Figure:

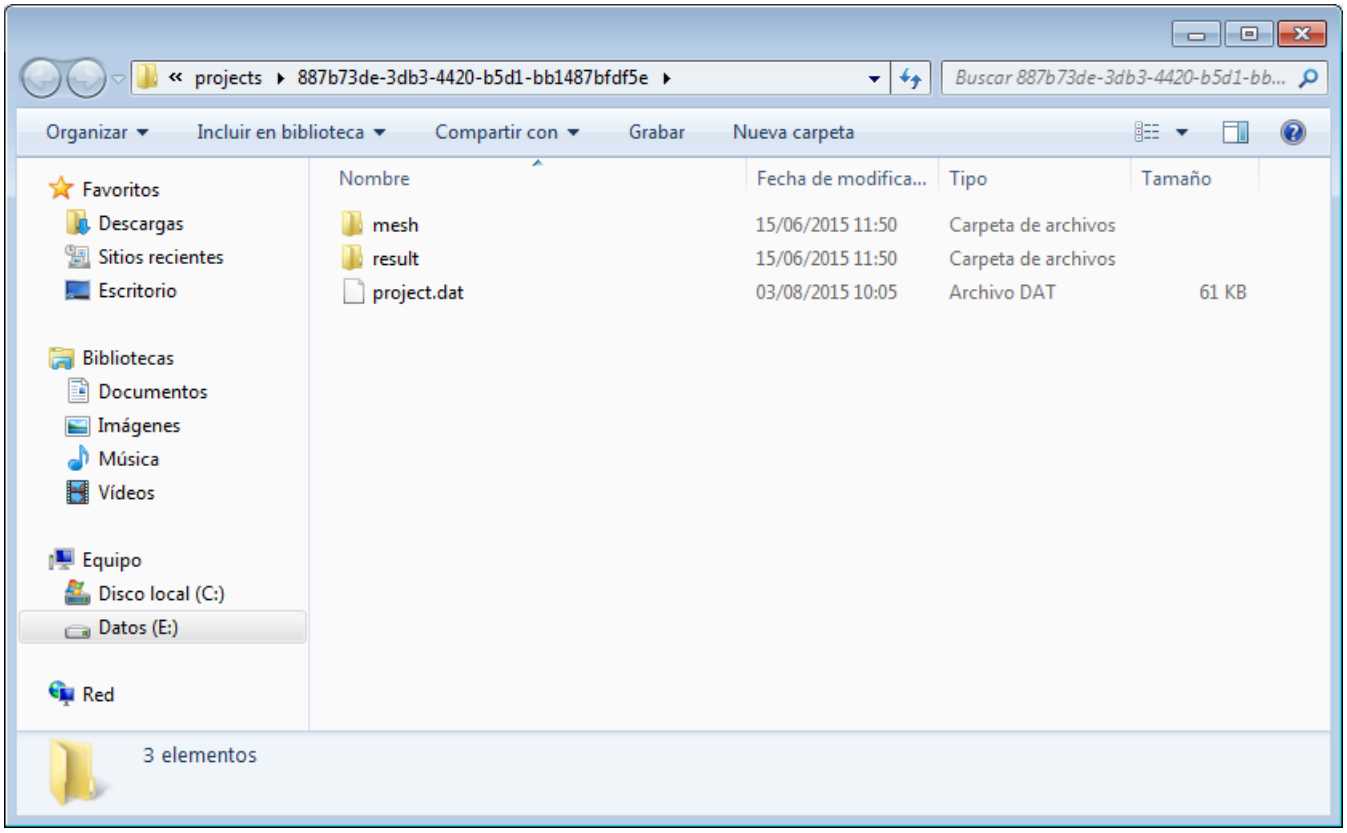


Figure 47: Project Folder window

2.2.9 Exit

This option will close the application. A dialog will offer the possibility to save the changes to the current project before exiting.

2.3 View Menu

These options allow the user to configure the properties of the main window area.

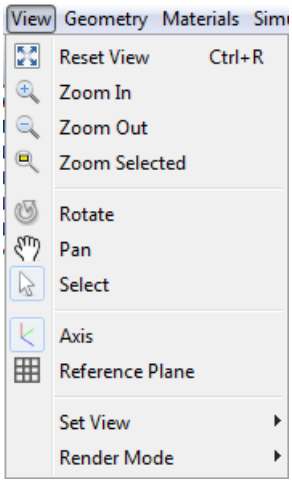


Figure 48: View menu

2.3.1 Reset View (Ctrl+R)

This option resets the view settings of the Geometry view area to the default values.

2.3.2 Zoom In

This option zooms in the current view of the Geometry view area.

2.3.3 Zoom Out

This option zooms out the current view of the Geometry view area.

2.3.4 Zoom Selected

This option zooms into the selected area of the Geometry view area.

2.3.5 Rotate

Use this option to rotate the current view at the geometry view area.

Remark Even if this tool is not selected, the user is also able to rotate the view by holding down the right mouse button while moving the cursor.

2.3.6 Pan

This option changes the current tool to the Pan tool. Using this tool, the user can move the field of view by using the cursor while holding down the left mouse button in the view area.

2.3.7 Select

This option changes the current mouse to the Selection tool. Using this tool, the user can select the different objects that appear in the view area by left-clicking them, or using the rectangular selection (hold on the left mouse button to draw a rectangle and select all the elements inside it). The user may also select several objects by holding down Ctrl and then left-clicking on the objects to select.

2.3.8 Axis

This option toggles the visualization of the three axes in the view area.

2.3.9 Reference Plane

This option shows the following panel on the right side of the window:

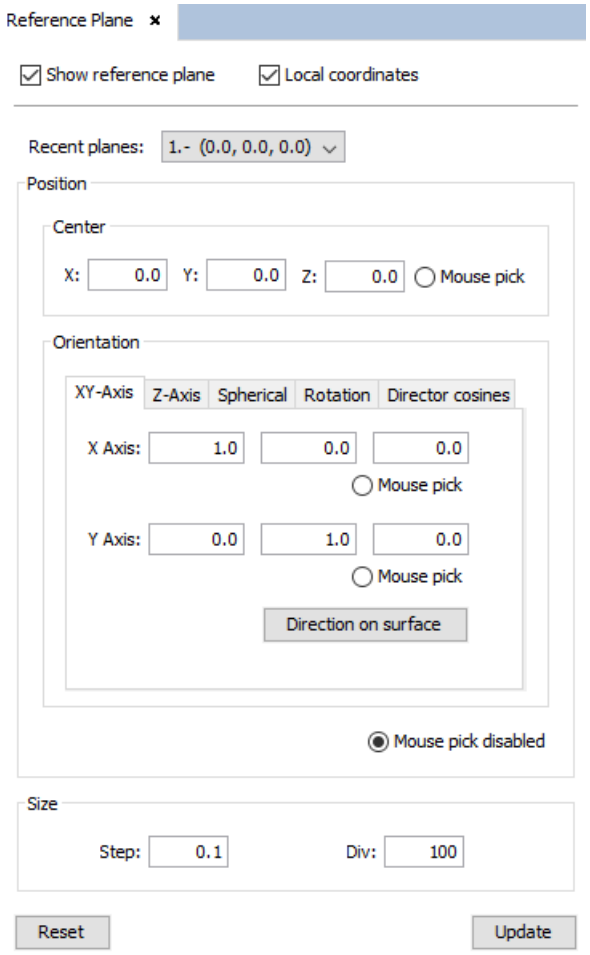


Figure 49: Reference plane panel

This panel allows the user to show a reference grid by selecting the center, orientation and resolution. It is especially useful for modeling geometry or placing elements, as it may be used as a **local coordinate system**.

All the options available in the Reference Plane panel are listed in the following:

The **Show reference plane** option paints or hides the current configuration of the reference plane.

When the **Local coordinates** check box is enabled, the local coordinate system defined by the reference plane is considered for new geometrical operations. Disable this option to work with the default Cartesian coordinate system.

All the previous different coordinate systems are listed in the **Recent planes** section. The reference plane is updated when an element of the list is selected, by using the current **Step** and **Div** parameters.

- The **position** section gathers the parameters required for the new local coordinate system.
- The options **Mouse pick** may be used to load the adjacent fields from the next clicked point on the geometry.
- The **Center** fields define the position where the reference plane is centered.

- The option **Mouse pick disabled** option deactivates the fields load from the mouse click events on the geometry.
- The **Orientation** subsection contains multiple ways for defining the coordinate system of the reference plane. The next options are available:
- ***XY-Axis***

The direction of the X and Y axes must be specified. Note that the directions are defined starting from the center position.

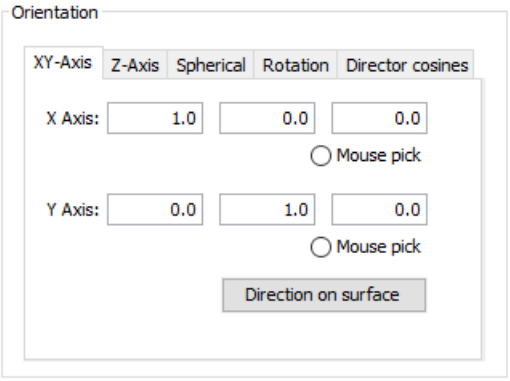


Figure 50: Orientation of reference plane with XY-axis

The **Direction on surface** button search the closest point to the specified **Center** on the selected surfaces. The positions parameters of the reference plane are directly loaded to generate a plane centered on the closest point on the surface that is tangent, i. e., the Z axis of the new reference plane is the normal vector of the surface at the center point.

Z-Axis

The reference plane is normal to the specified **Z axis**. Once the plane is defined, it also may be rotated by specifying the **Alpha rotation** angle, in degrees.

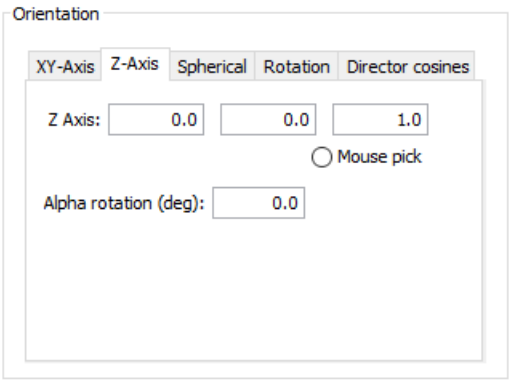


Figure 51: Orientation of reference plane with Z-axis

Spherical

The reference plane orientation is defined in spherical coordinates with the **Theta** and **Phi** angles, and also may be rotated with the **Alpha** rotation.

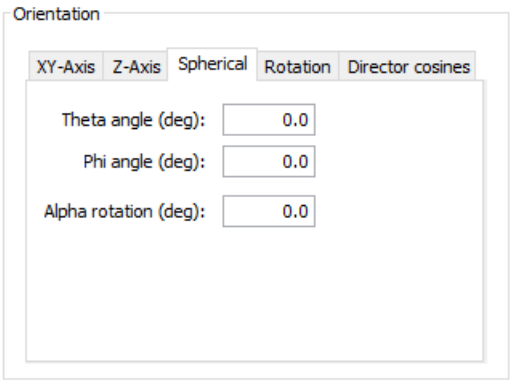


Figure 52: Orientation of reference plane with Spherical coordinate system

Rotation

The reference plane is defined by applying the **X**, **Y** and **Z rotation angles**, centered at the specified position.

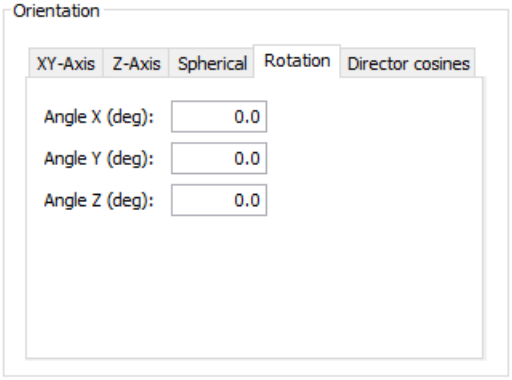


Figure 53: Orientation of reference plane with Rotation angles

Direction cosines

The reference plane's coordinate system is directly defined by its **rotation matrix**.

Figure 54: Orientation of reference plane with Director cosines

The **Size** section allows to change the resolution and size of the reference plane. Note that the reference plane is always a square grid, so the **Size** properties affects both directions of the plane.

- The **Step** parameter modifies the length of all divisions.
- The **Div** parameter changes the number of divisions.

The **Update button** repaints the new reference plane with the last updated parameters.

The **Reset button** restores all the default parameters of the Reference Panel, so it is placed at the XY plane and centered at the origin.

2.3.10 Set View

This submenu contains several options that set the user view to different perspectives.

- The perspective view is the real one: the camera rotates around the center.
- The parallel view sets a visualization to a plane view: the dimensions are projected always to a plane.

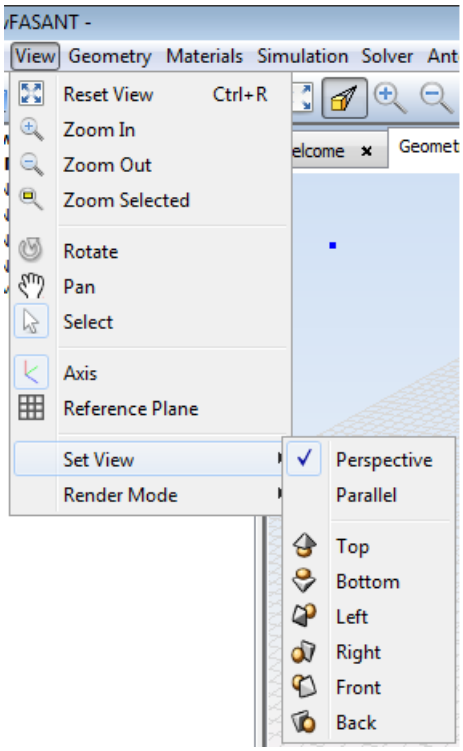
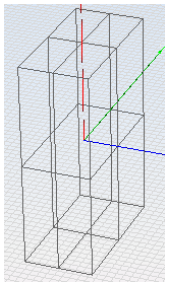
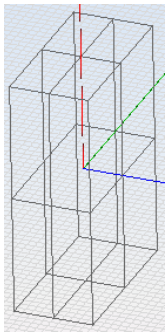
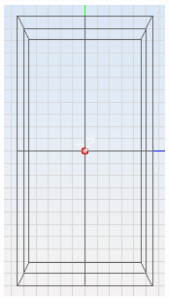
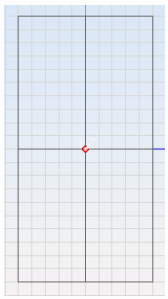
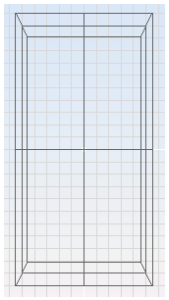
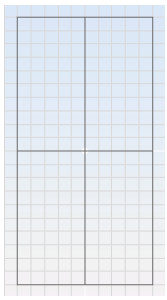
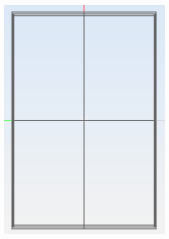
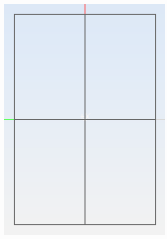


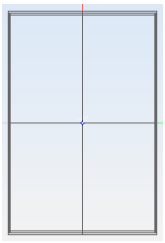
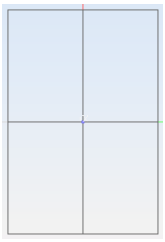
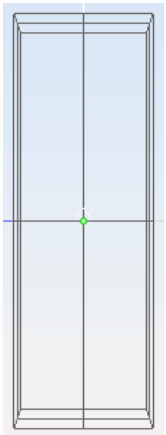
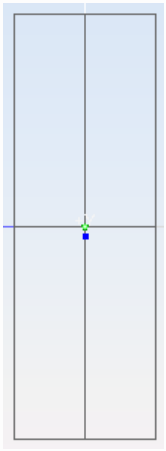
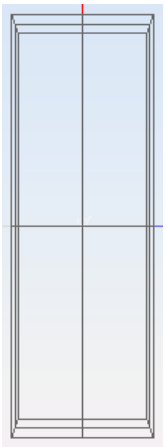
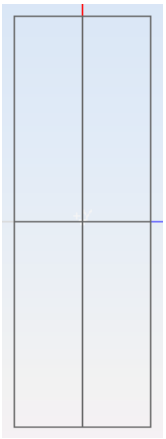
Figure 55: View modes

A box has been created with the following parameters to compare the different view possibilities.

```
command> box
First corner of base [x y z]: -0.5 -1 -1.5
Base size [width depth]: 1 2
Height [double]: 3
command>
```

The next table shows the comparison of all views of the box.

View	Perspective	Parallel
Reset view		
Top		
Bottom		
Left		

View	Perspective	Parallel
Right		
Front		
Back		

2.3.11 Render Mode

This submenu contains several options that affect how the geometry is rendered. The effects for each one of these options can be seen in the following figures:

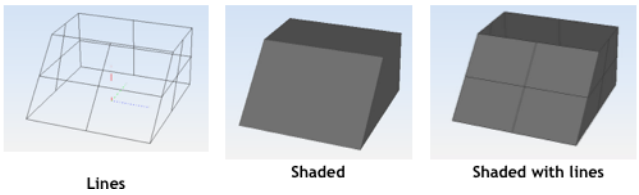


Figure 56: Render modes

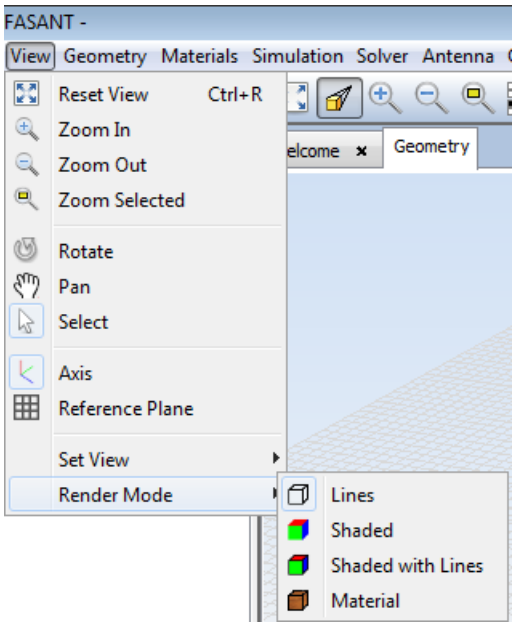
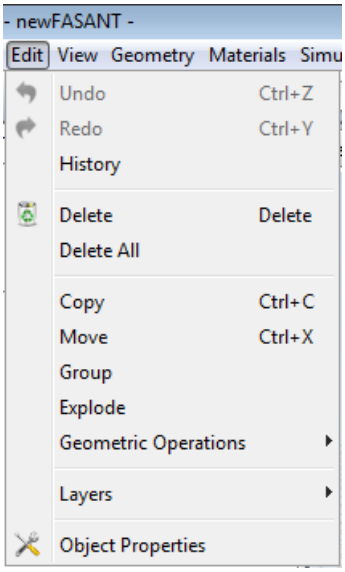


Figure 57: Submenu

In addition, the submenu also contains a Material Render Mode. This only modifies the figures color by the corresponding material color.

2.4 Edit Menu



2.4.1 Undo (Ctrl+Z)

The undo option is used to undo the previous change. The same functionality can be achieved using the button on the toolbar.

2.4.2 Redo (Ctrl+Y)

If the user has undone any changes it is possible to redo them with this option. The same functionality can be achieved using the button on the toolbar.

2.4.3 History

This option opens a new panel on the right side of the window. In this panel we can see and select the different objects in the geometry view. Also we can delete this objects history or all history. It is possible to modify the values of the parameters that define the objects by clicking on **Save** after changing the values.

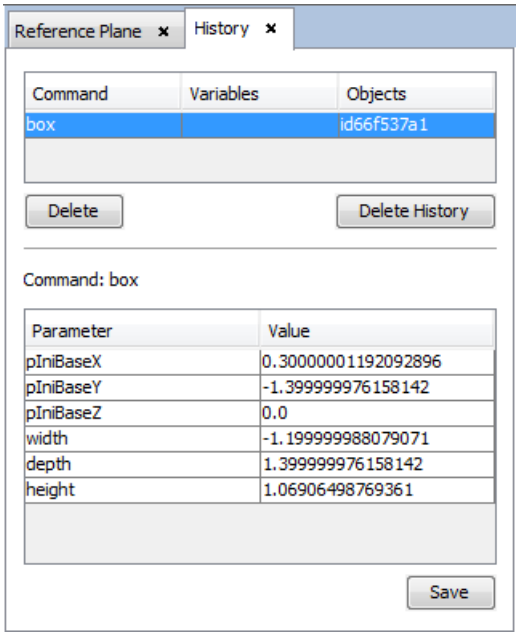


Figure 58: History panel

2.4.4 Delete

With this option the user can delete a set of objects on the screen by selecting them and pressing enter. This option is associated with the "delete" command, so this functionality can also be achieved from the command line.

2.4.5 Delete All

This option is used to remove all the objects of the geometry. It is associated with the "delete all" command of the command line.

2.4.6 Copy (Ctrl+C)

An object can be copied into another position. For example, given a box-shaped geometry, after clicking on the Edit → Copy button, different options to select the objects to be copied are available:

- Copy Selection multiple objects can be selected by pressing simultaneously "Ctrl + Left Mouse button".
- Copy All to select all existing objects in the geometry.

With this option the user can select the object to copy by entering the coordinates where it will be placed or by clicking directly on the geometry area and pressing enter.

For example, in the figure below we will make a copy of the box. To do this we click on copy, select the object and press enter. Now we introduce the points as seen in the figure and press enter. In this example the result is shown in next Figure.

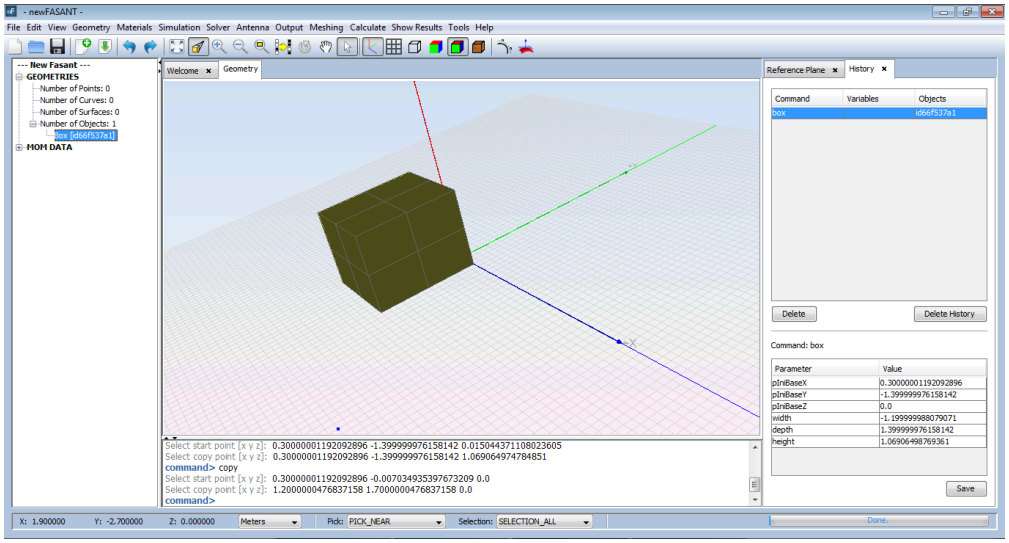


Figure 59: Initial state of the geometry

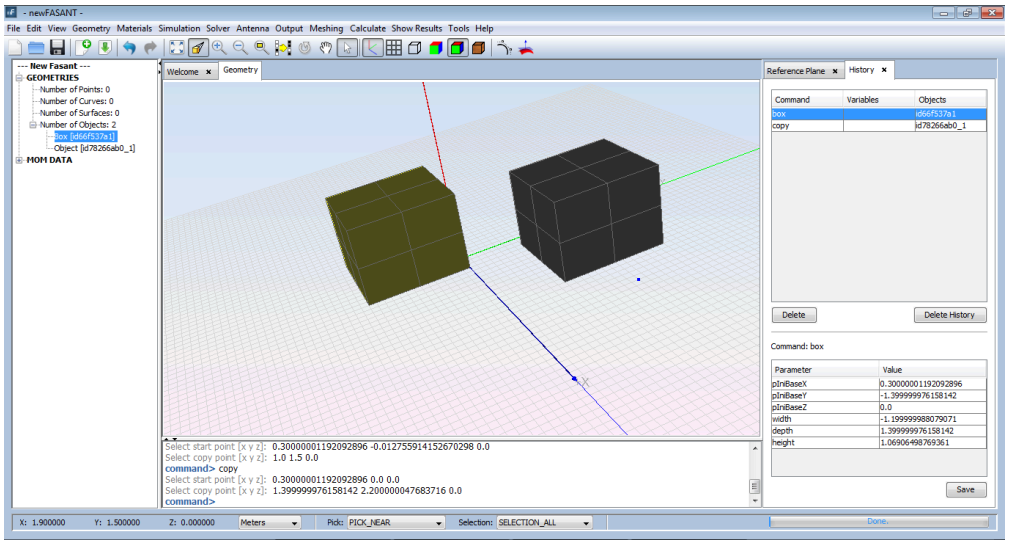


Figure 60: After copying

2.4.7 Move (Ctrl+X)

An object can be moved into another position. After clicking on the **Edit** → **Move** button, different options to select the objects to be moved are available:

- Move Selection multiple objects can be selected by pressing simultaneously “Ctrl + Left Mouse button”.
- Move All to select all existing objects in the geometry.

With this option the user can select the object to move by entering the coordinates where it should be placed or by clicking directly on the surface and pressing enter.

For example, if we want to move the object in the previous figure of the Copy section, we click on the object and select move. For the sake of illustration, we explode the box and then select only one of the exploded surfaces, after that, we introduce the origin and destination points. The result is shown in the following Figure.

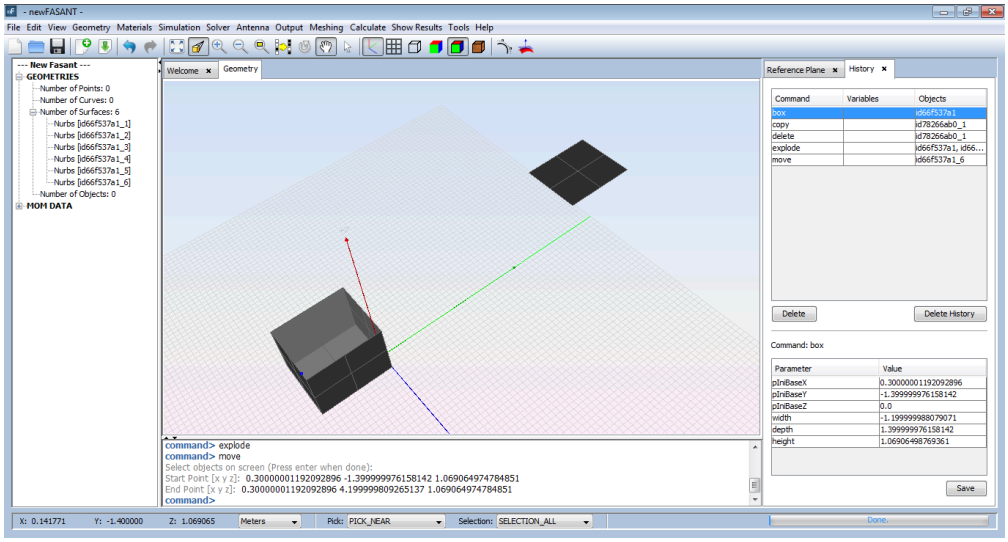


Figure 61: Moving a surface

2.4.8 Duplicate

This option allows to create a copy of the selected objects that is placed at the same position than the original ones. The figure below show an example for duplicating a plane.

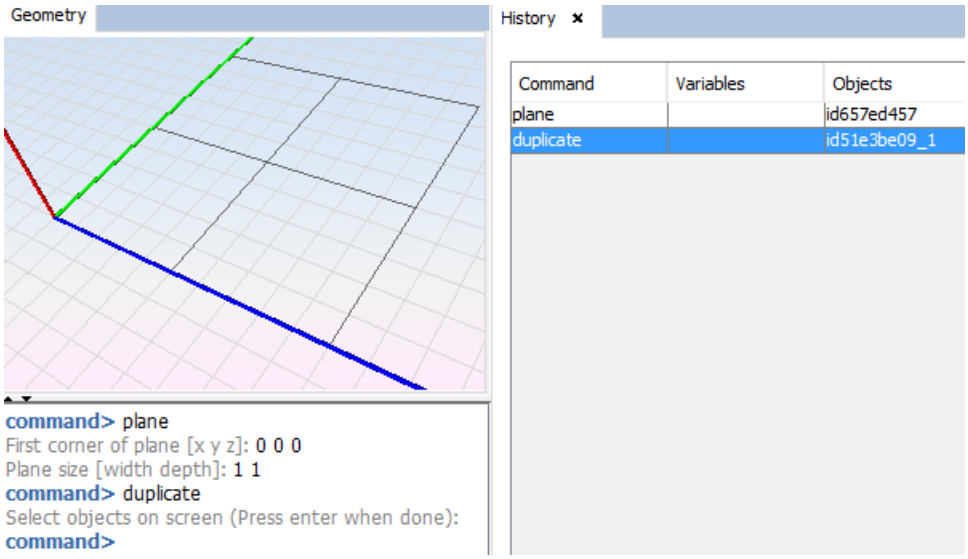


Figure 62: Duplicated plane

2.4.9 Group

Two or more objects can be grouped into an unique one. This operation turns the original objects into an imported object that can be used to perform different operations, but the properties of the original primitives disappear.

The user can select the surfaces and then click on **Group**.

For example, we will apply group surfaces on surfaces of the next Figure. We can see that in the tree on the left we have zero objects and six surfaces.

Then, select all and click on **Group**. We can see the result in the tree on the left of the subsequence. Now we can see that we have zero surfaces and an object.

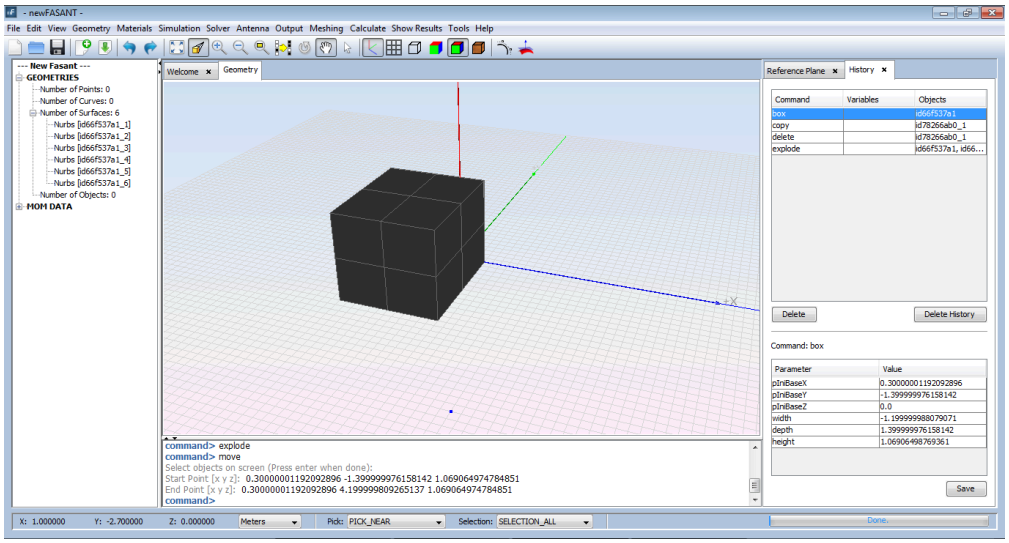


Figure 63: State of the geometry before grouping

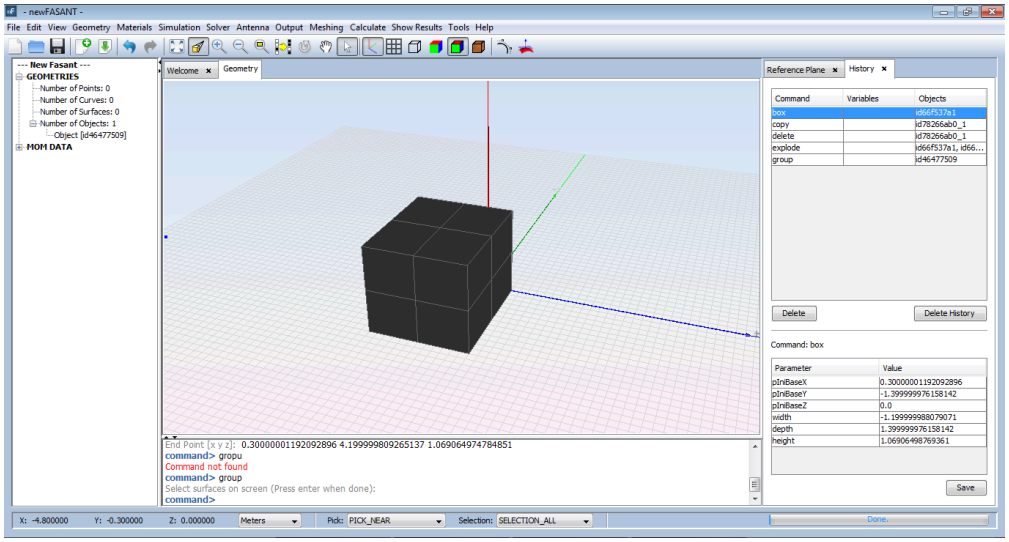


Figure 64: After grouping the surfaces

2.4.10 Explode

An object can be broken into its components. When the option Edit → Geometric Operations → Explode is selected, a number of options to select the objects to be exploded are available:

By clicking on them, multiple objects can be selected (pressing simultaneously “Ctrl + Left Mouse button)”.

Rectangle selection by drawing a rectangle and selecting the objects inside the rectangle.

For example, In the figure below, we have an object and zero surfaces. If we select the explode option and the whole object, we can see that we now have six surfaces and zero objects.

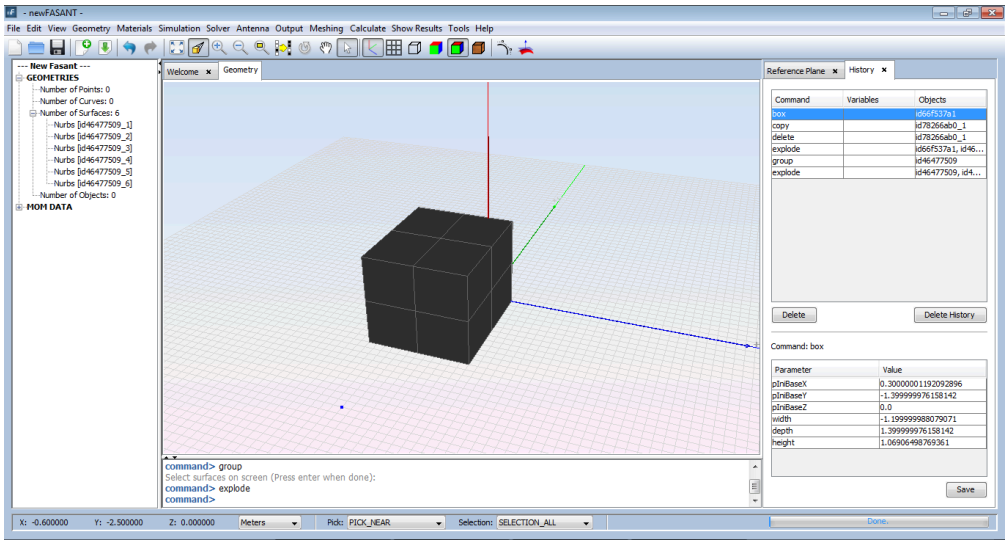


Figure 65: Explode

2.4.11 Geometric Operations

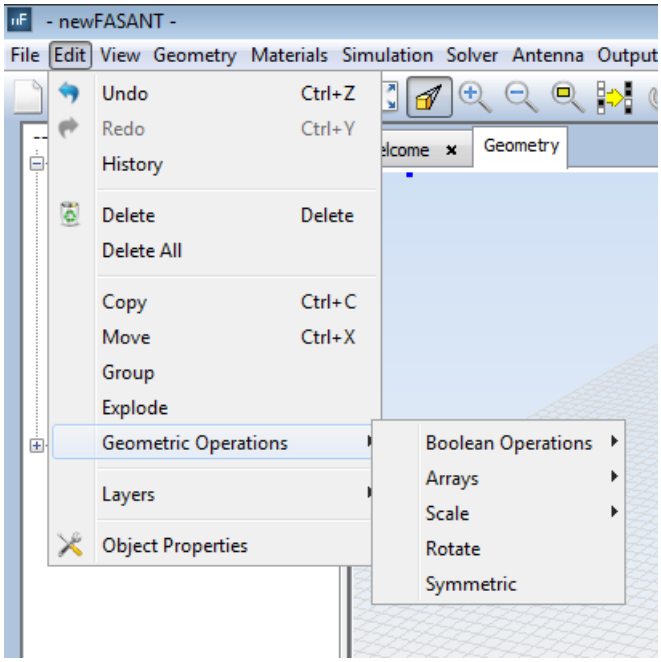


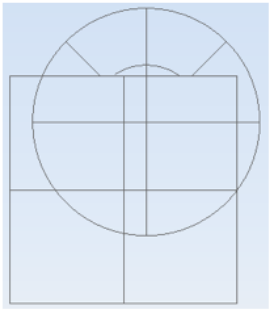
Figure 66: Geometric Operations Menu

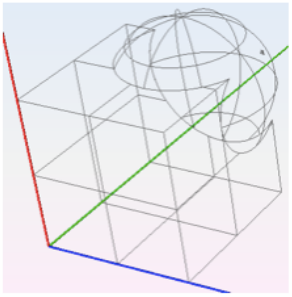
Boolean Operations

Union

The union operation computes the total regions where at least one of the two candidate objects exits, i.e., the OR logical operation. The order of the selected objects does not modifies the result.

Object A	Object B	OR
0	1	1
1	0	1
1	1	1

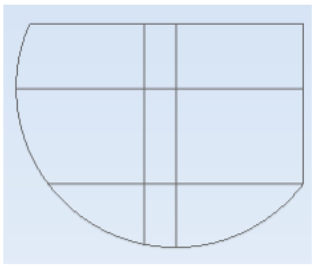


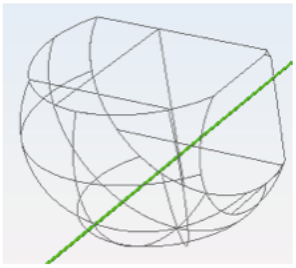


Intersection

The intersection operation computes the mutual regions of two different objects, i.e., the AND logical operation. The order of the selected objects does not modifies the result.

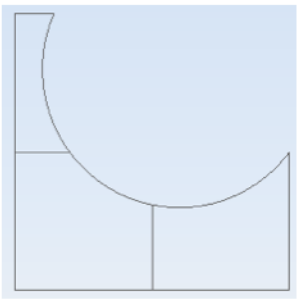
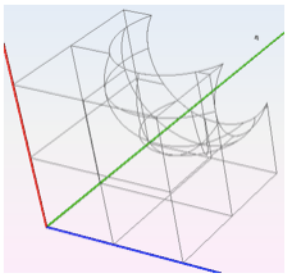
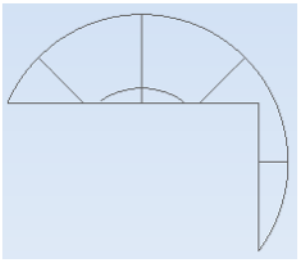
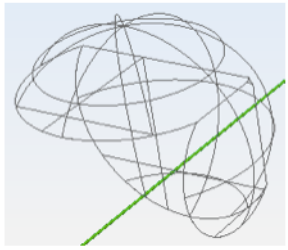
Object A	Object B	AND
0	1	0
1	0	0
1	1	1





Difference

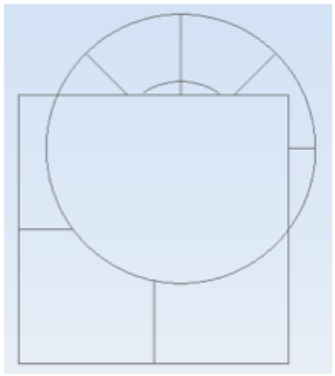
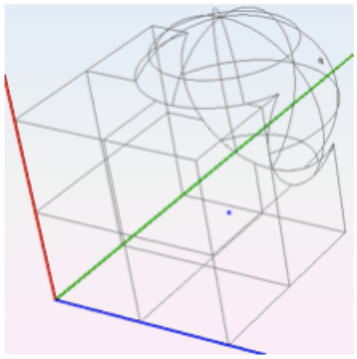
The difference operation computes the regions of the first object after subtracting the second one, i.e., the MINUS operation.

XOR

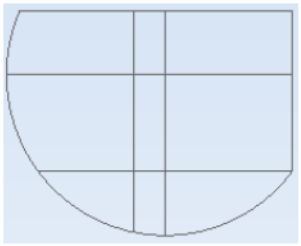
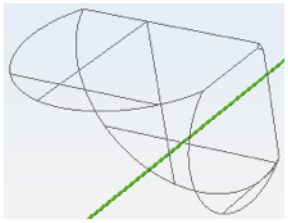
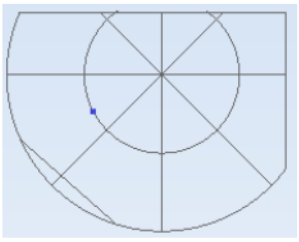
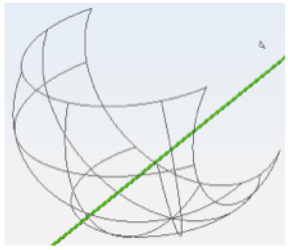
The union-exclusive operation computes the regions where only one of the two objects exits, i.e., the XOR logical operation. The order of the selected objects does not modifies the result.

Object A	Object B	XOR
0	1	1
1	0	1
1	1	0

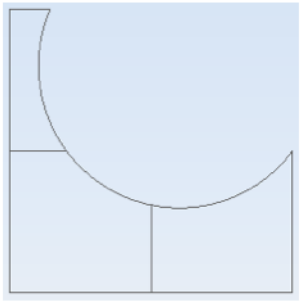
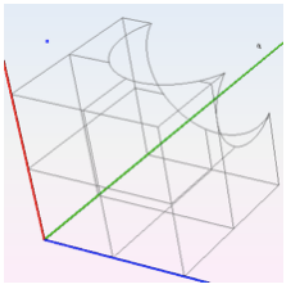
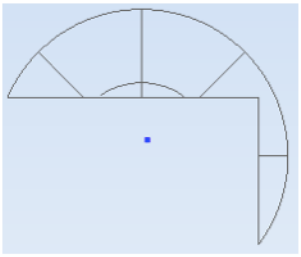
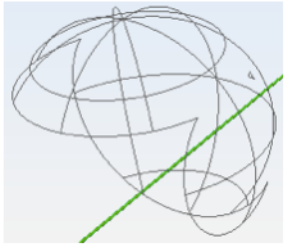
Inside

The inside operation computes the regions of the first object that is completely contained in the second one.

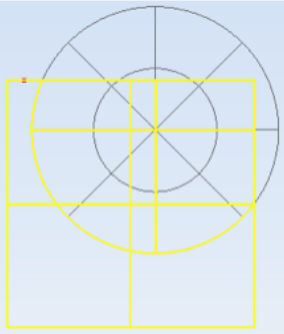
Outside

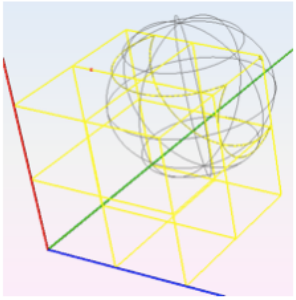
The outside operation computes the regions of the first object that is not contained in the second one.

Split

The split operation computes the cuts that an objects produce in the other one. The command may process both surfaces and objects, and the result is returned in the same format, i.e., after splitting a surface, the resulting sections is returned in multiple surfaces; however, an object is returned as the same object but with the resulting surfaces of the operation (explode it if necessary). Intersections only is computed between different objects.





Arrays

This menu contains the available options for arrays generation.

3D Array

This is an option to create an array from an existing geometry. This option lets the user easily create a matrix of geometries. It is possible to create a circular array using the array command with the parameter -cir <radius>.

First, create any geometry (for example a sphere). To create it, click on “Geometry”, “Solids”, and click on “Sphere”. Now click on “Edit”, “Geometric Operations”, and click on “Array”.

This option is used to double up the selected surface. To do this, enter the number of copies in X, Y and Z and the distance between them using a starting point and end point.

For example, create a rectangle and press the option **array**. Then select the desired surface as shown in the next Figure.

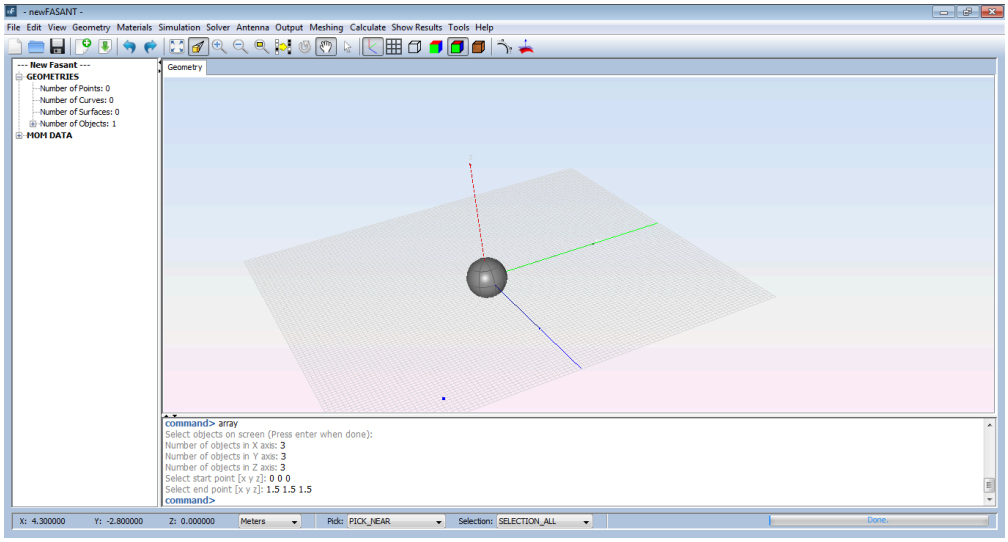


Figure 67: Selecting the sphere to replicate

Then introduce the parameters shown in the command line of the figure below and press enter. The results are seen in the next Figure.

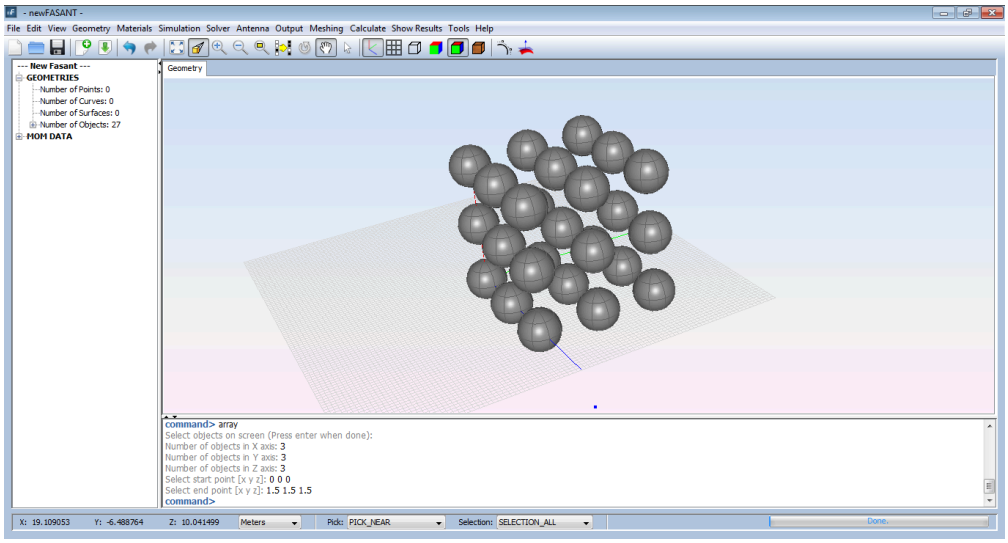


Figure 68: After applying the array function

ArrayOnSurface

This is an option to create an array from planar elements conformed to the selected surface.

Defining the cell structure on XY Plane and centered at the origin is the first step. In the example below, a disk with center in 0,0,0 and radius 0.04 is considered.

The next step is the creation of the object or surface where the cell structure will be placed on. For example, a sphere is created with center in 0,0,0 and radius 0.5, then it is exploded for extracting its surfaces.

Now click on Edit - Geometric Operations – Arrays – ArrayOnSurface option to start with the array generation.

The command ask the selection of the surfaces where the cell will be placed. This selection is only valid over surfaces entities, so objects should be exploded before using the command.

The first required option is the mode of array generation. Four different ways of cell distributions are available:

- **Parametric** This option generates a quasi-regular distribution of cells along the parametric dimensions of the surfaces, i. e., the cells sizes may vary in the surfaces but the structure shape is guaranteed. It is recommended for non-degenerated surfaces or when the size of the cell in X and Y dimension must be different.
- **Global** This option generates a more regular distribution of cells on the surfaces, i. e., the cells sizes and the structure shape may be slightly varied but the global quality of the array is improved. It is recommended for most of structures but it is only available when the size of the cell in X and Y dimension are identical.
- **Projected** This option generates a full regular distribution of cells along the reference plane and then they are projected onto the target surfaces, i. e., both the cells sizes and the structure shape are guaranteed whenever the curvature of the surfaces is soft enough. It is only recommended for soft-curvature surfaces. When this mode is selected, the command ask whether the cells have to be projected onto the surfaces bounds or not:
- ◦ Yes (by default) option generates and projects the cells on full surfaces, and the bounding cells are trimmed.
  ◦ No option only generates cells on the rectangles (Tx Ty) that are completely inside the surfaces.
- **Manual** This option place cells on the closest point on the surfaces to the introduced by the user, i. e., both the cells sizes and the structure shape are guaranteed whenever the curvature of the surfaces is soft enough in the selected points. It is recommended for customizing the cells distribution in most of structures.

The next parameter required is the type of array. Two different types of array types are available:

- **Capacitive** Metallic cell structures are generated as the original one.
- **Inductive:** Metallic cell structured are inverted to the original one, i.e., the generated cells are rectangular cells with holes with the shapes of the original cell.

Then, the size of the cell structure (in the units selected) is required. As a planar cell is considered, the size in X and Y dimensions are required. In the considered example, 0.1, 0.1 are introduced.

When the sizes of cell structure in X and Y are different, the global option returns an error because the two sizes must be identical for this mode.

The last parameter required is the selection of the planar object used as reference cell. Note that these objects must be centered at the origin of coordinates, contained in the XY plane and surrounded by the red rectangle (of Tx Ty sizes) highlighted. If multiple surfaces are required for the cell definition, all of them must be grouped in one unique object.

When the array operation is finished, the original surfaces and cells are in the geometry. It may be removed by using the **Delete** command.

The next figure shows the full example of the disk array on the sphere with all the parameters required by default (the original objects have not been deleted).

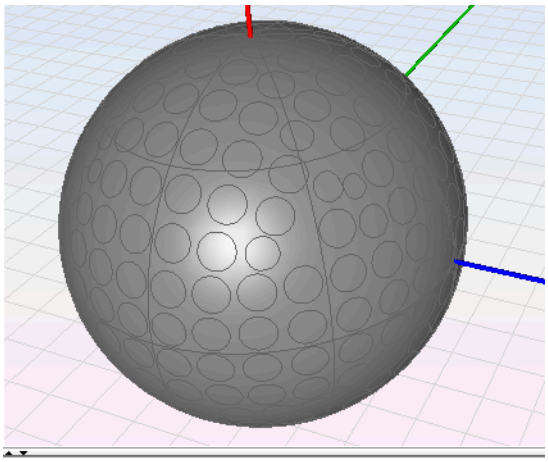


Figure 69: Disk array on sphere with global mode and capacitive type

The next figure shows the full example of the disk array on the sphere generated with parametric mode (the original objects have not been deleted).

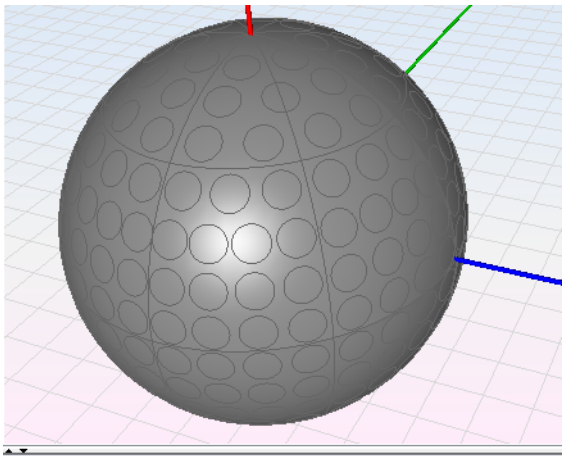
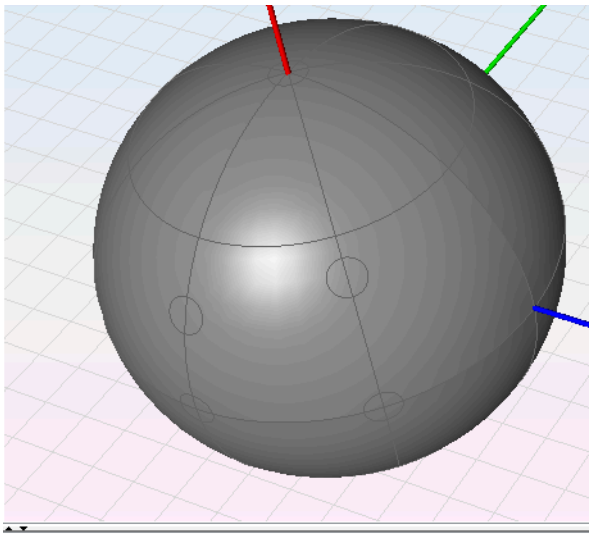


Figure 70: Disk array on sphere with parametric mode and capacitive type

The next figure shows the full example of the disk array on the sphere generated with manual mode (the original objects have not been deleted).



```
command> arrayOnSurface
Select surfaces on screen (Press enter when done):
Select array mode [1-Parametric,2-Global,3-Projected,4-Manual](default=2): 4
Select cell center points [Press Enter when done]
Point 0 [x y z]: 0 -0.5 0
Point 1 [x y z]: 0.5 -0.5 0
Point 2 [x y z]: 0.5 -0.5 0.5
Point 3 [x y z]: 0 -0.5 0.25
Point 4 [x y z]: 0 0 0.5
Point 5 [x y z]:
Select array type [1-Capacitive,2-Inductive](default=1):
Size of cell structure [Tx Ty]: 0.1 0.1
Select one object with planar elements (Press enter when done):
Please Wait...
command>
```

Figure 71: Disk array on sphere with manual mode and capacitive type

The next figure shows again the full example of the disk array on the sphere generated with the global mode and the inductive type (the original objects have been deleted).

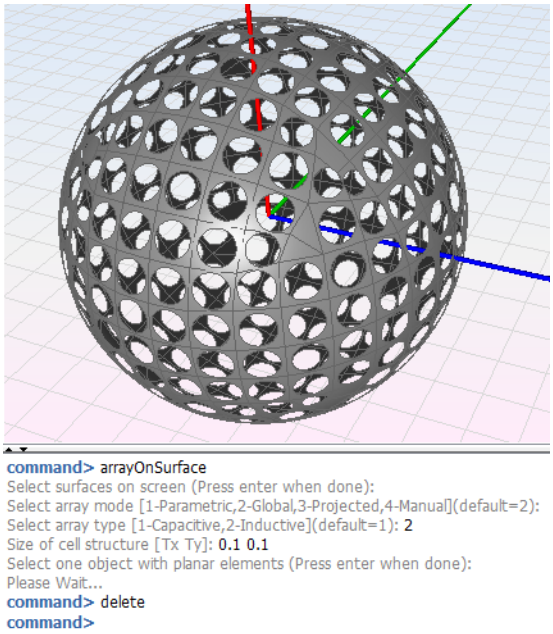


Figure 72: Disk array on sphere with global mode and inductive type

The next figure shows the full example of the disk array on the sphere generated with the parametric mode and the inductive type (the original objects have been deleted).

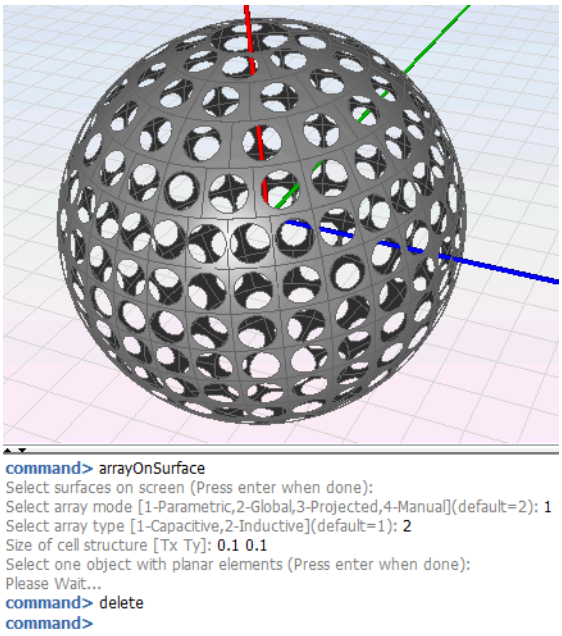


Figure 73: Disk array on sphere with parametric mode and inductive type

The next figure shows the full example of the disk array on the sphere generated with the manual mode and the inductive type (the original objects have been deleted).

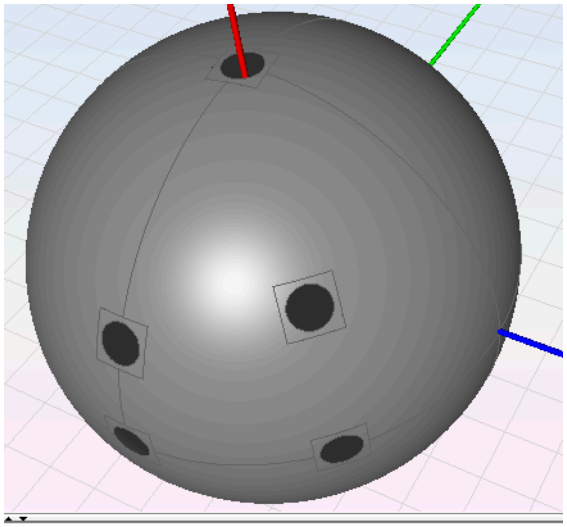
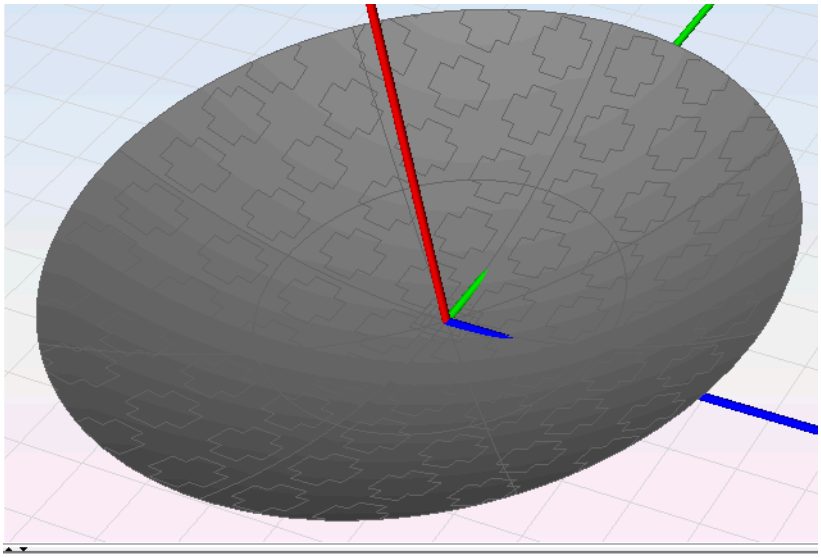


Figure 74: Disk array on sphere with parametric mode and inductive type

In the last part of this section, a paraboloid is considered to evaluate the generation of a crosses array with the projected mode.

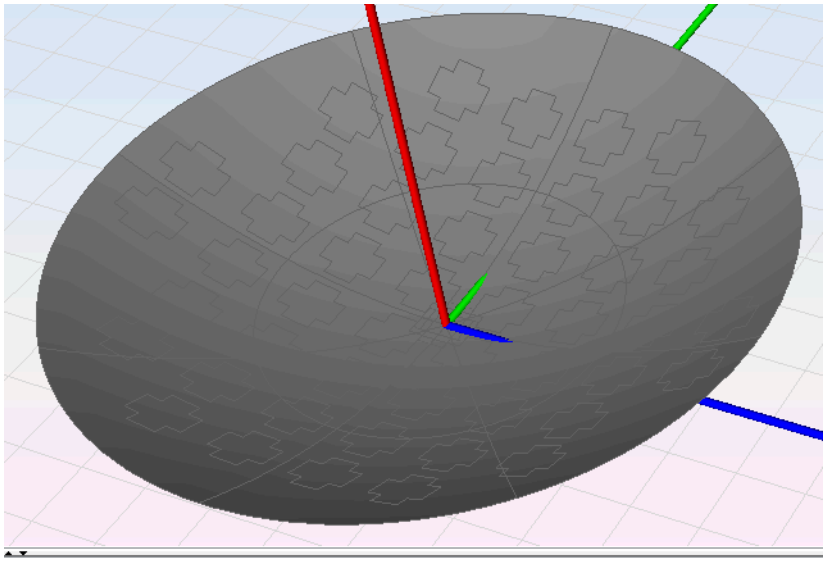
The next figure shows the full example of the crosses array on the paraboloid with the rest of required parameters by default (the original objects have not been deleted).



```
command> paraboloid
Select center [x y z]: 0 0 0
Hole radius [double]: 0
Focus distance [double]: 0.4
Diameter X [double]: 1
Diameter Y [double]: 1
command> cross
Select center [x y z]: 0 0 0
Size A [double]: 0.04
Size B [double]: 0.02
command> explode
command> arrayOnSurface
Select surfaces on screen (Press enter when done):
Select array mode [1-Parametric,2-Global,3-Projected,4-Manual](default=2): 3
Do you want to project the FSS onto surface bounds [y/n] (defautl=y):
Select array type [1-Capacitive,2-Inductive](default=1):
Size of cell structure [Tx Ty]: 0.12 0.1
Select one object with planar elements (Press enter when done):
Please Wait...
command>
```

Figure 75: Cross array on paraboloid with projected mode, projecting onto surface bounds and capacitive type

The next figure shows the full example of the cross array on the paraboloid without projecting onto surface bounds, and the rest of required parameters by default (the original objects have not been deleted).



```
command> paraboloid
Select center [x y z]: 0 0 0
Hole radius [double]: 0
Focus distance [double]: 0.4
Diameter X [double]: 1
Diameter Y [double]: 1
command> cross
Select center [x y z]: 0 0 0
Size A [double]: 0.04
Size B [double]: 0.02
command> explode
command> arrayOnSurface
Select array mode [1-Parametric,2-Global,3-Projected,4-Manual](default=2): 3
Do you want to project the FSS onto surface bounds [y/n] (defautl=y): n
Select array type [1-Capacitive,2-Inductive](default=1):
Size of cell structure [Tx Ty]: 0.12 0.1
Select one object with planar elements (Press enter when done):
Please Wait...
command>
```

Figure 76: Cross array on paraboloid with projected mode, without projecting onto surface bounds and capacitive type

The next figure shows the full example of the cross array on the paraboloid with inductive mode and the rest of required parameters by default (the original objects have been deleted).

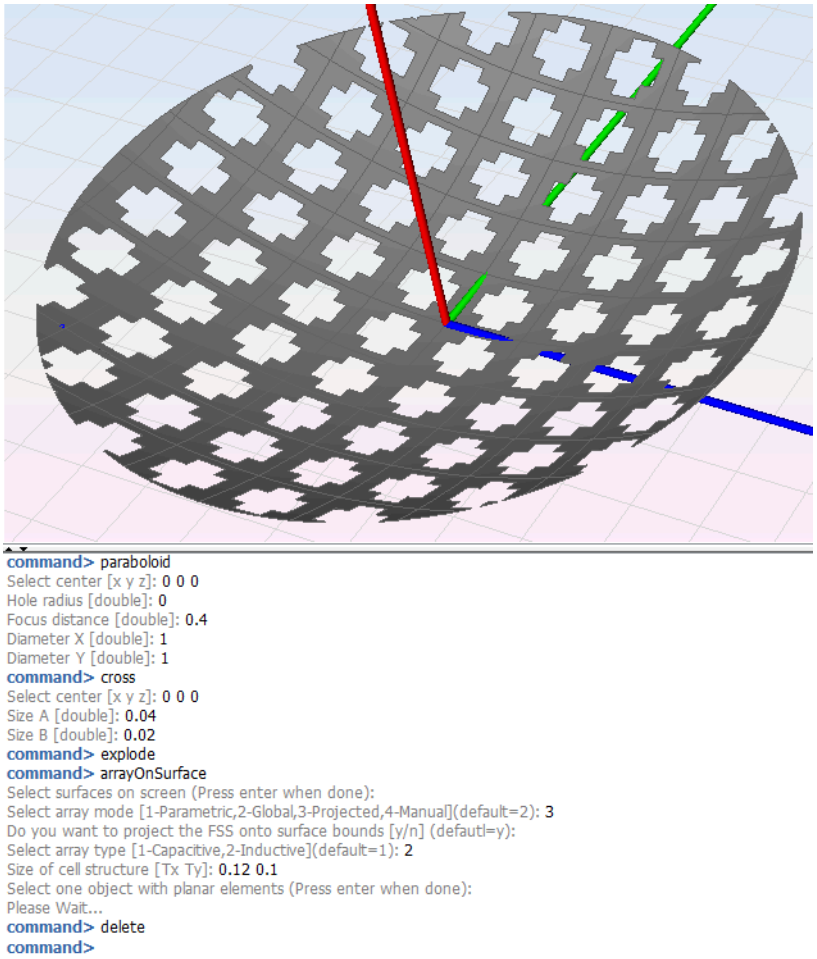


Figure 77: Cross array on paraboloid with projected mode, projecting onto surface bounds and inductive type

The next figure shows the full example of the cross array on the paraboloid without projecting onto surface bounds inductive mode, and the rest of required parameters by default (the original objects have been deleted).

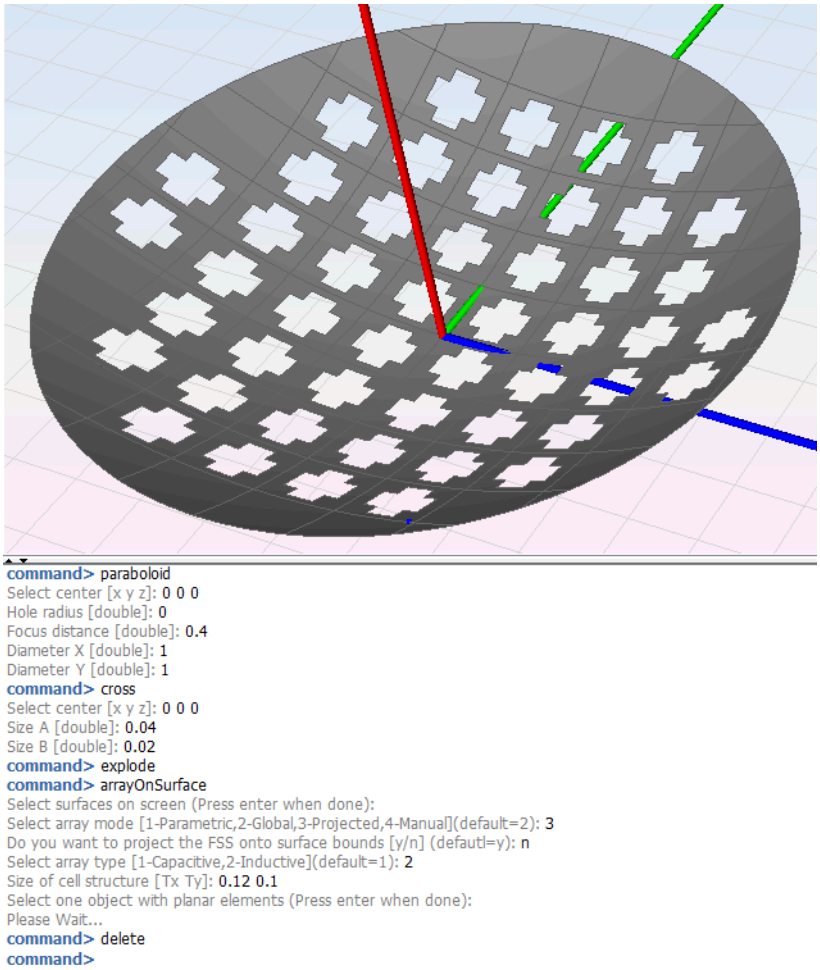


Figure 78: Cross array on paraboloid with projected mode, without projecting onto surface bounds and inductive type

Scale

Scaling any geometry is also possible by using the Scale menu. When the Edit – Geometric Operations - Scale button is clicked, different options to select the objects to be scaled are available:

- Scale Selection multiple objects can be selected by pressing simultaneously “Ctrl + Left Mouse button”.
- Scale All to select all existing objects in the geometry.

Several options of scaling are available in this menu. To select one of them, click on Edit - Geometric Operations – Scale menu. A cylinder centered at the origin with radius of 0.25 and height of 1 has been generated for the following sections.

Scale 1D

By using this option, only the X, Y or Z dimension is scaled by the scale factor.

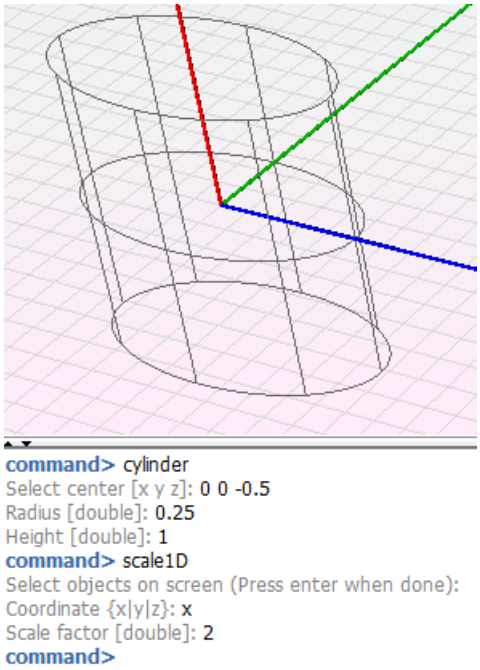


Figure 79: Example of a cylinder scaled in the X dimension

Scale 2D

By using this option, only two of the X, Y or Z dimensions are scaled by the same scale factor.

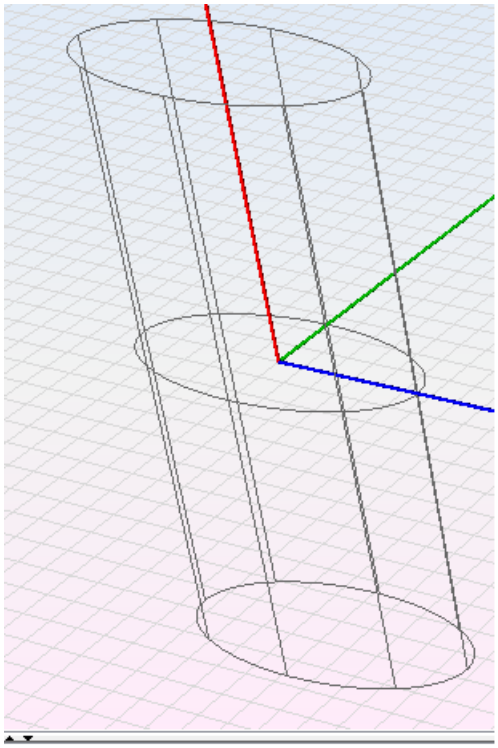


Figure 80: Example of a cylinder scaled in the X and Z dimensions

Scale 3D

By using this option, the three X, Y and Z dimensions are scaled by the same scale factor. This function is the corresponding to the “scale” or “scale3D” commands.

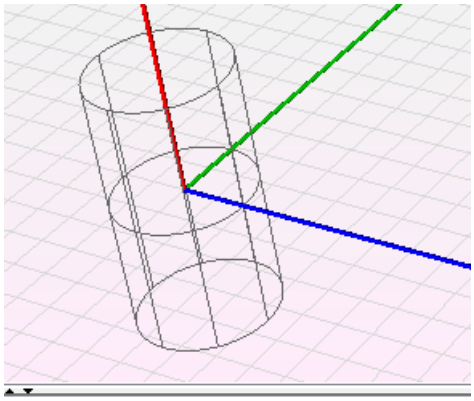


Figure 81: Example of a cylinder scaled in the three dimensions

Scale Non-Uniform

By using this option, the three X, Y and Z dimensions are scaled by the corresponding scale factor each one.

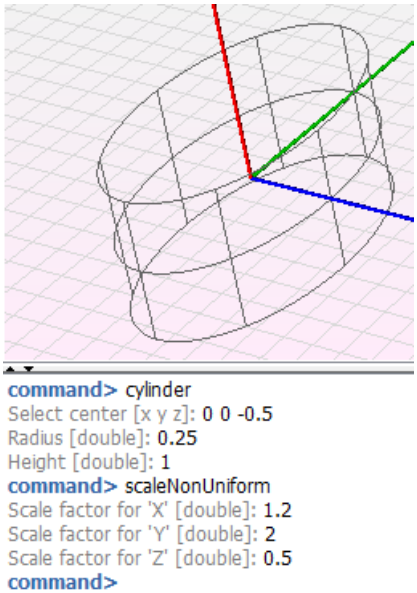


Figure 82: Example of a cylinder scaled in the three dimensions with different factors

Rotate

This option is used to rotate the selected objects. The user will select the objects and enter the coordinates of the points that form the rotation axis and the angle of rotation and press enter, as shown in the following figure.

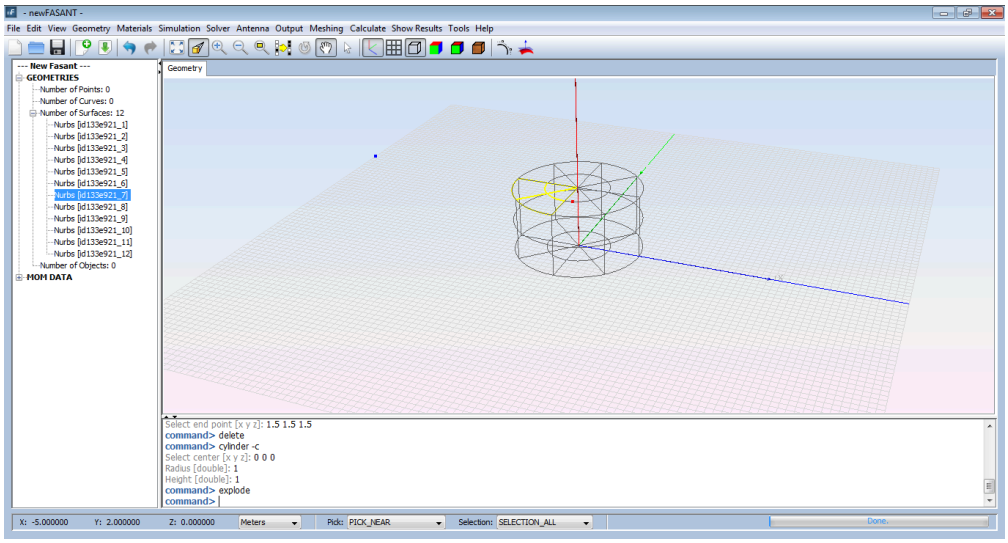


Figure 83: Selecting the surface to rotate

After selecting the surface and having selected the option to rotate, introduce the desired points. The result appears in the next Figure.

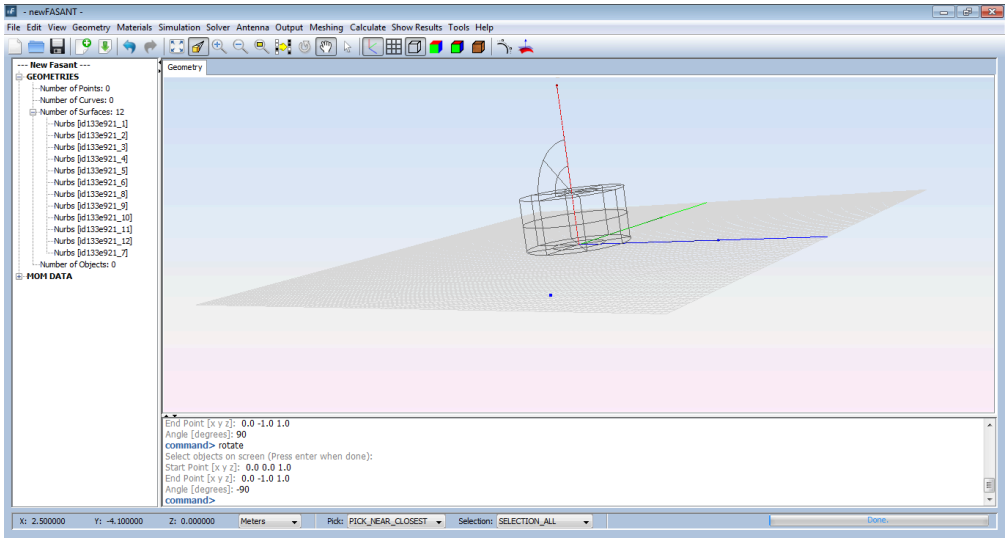


Figure 84: Rotate

Symmetry

The symmetry operation can be used to generate new objects. After clicking on the Edit – Geometric Operations – Symmetric button, different options to select the objects are available:

- **Symmetry Selection** multiple objects can be selected by pressing simultaneously “Ctrl + Left Mouse button”.
- **Symmetry All** to select all existing objects in the geometry.

By selecting the proper surface or figure, the system will require to introduce two points. These two points form a vector which is the vector normal to the plane of symmetry. Pressing enter creates the symmetric of the selected surface from this plane.

Note that this option modifies the original input object. In case of wanting a copy and then symmetry operation on the copied objects, the two operations may be grouped by using the command 'symmetric -c'.

For example, introduce a cylinder and click on Edit - Geometric operations and symmetric, and select the surface as shown in the figure, pressing enter, introduce points as those seen in the following Figure and the result is shown in the subsequent Figure.

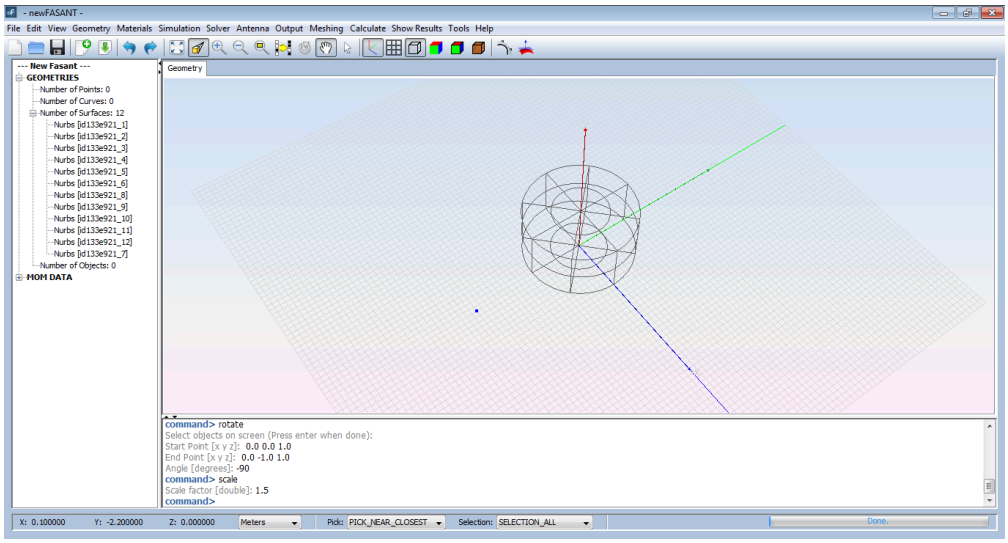


Figure 85: Symmetry

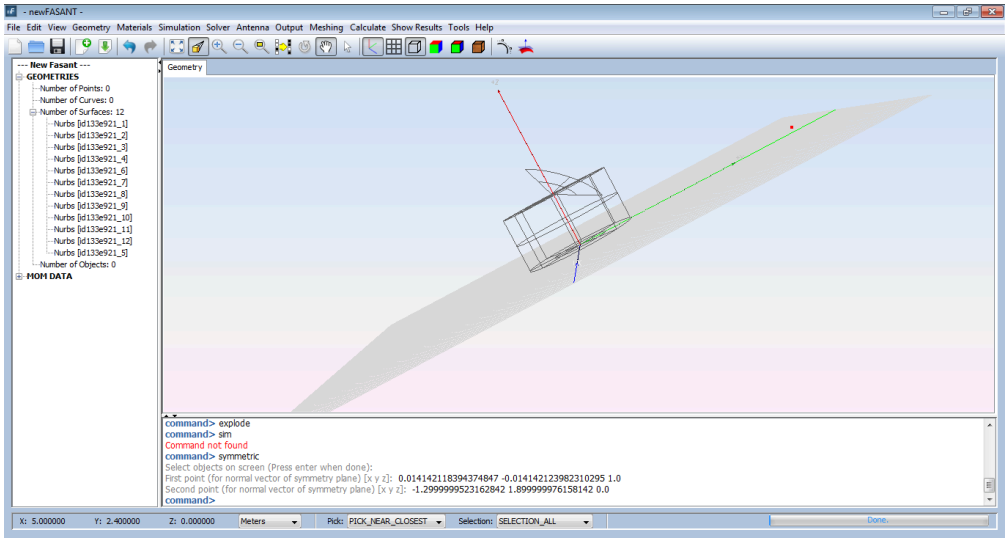


Figure 86: Symmetry

Transform to Reference Plane

This option applies the transformation of the **Reference Plane** to the selected objects, and also can be used by using the **localTransform** command. It is useful to create geometrical objects in the default origin and then translate and orientate them with the desired reference plane.

The figure below shows two planes of different sizes that have been created in the default system.

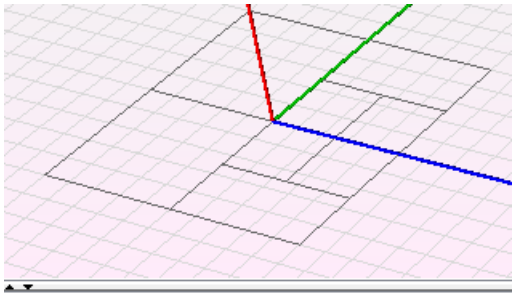


Figure 87: 2 planes created with the default reference plane

Then, the reference plane has been centred in the right edge of the first plane. It has been also oriented to be contained in the YZ plane, so its Z-Axis is pointing to the X axis. By using the new reference plane, select the second created plane and then click on **Edit - Geometric Operations - Transform to reference plane** option. Note that the selected plane has been moved an rotated according to the reference plane parameters, as shown in next figure.

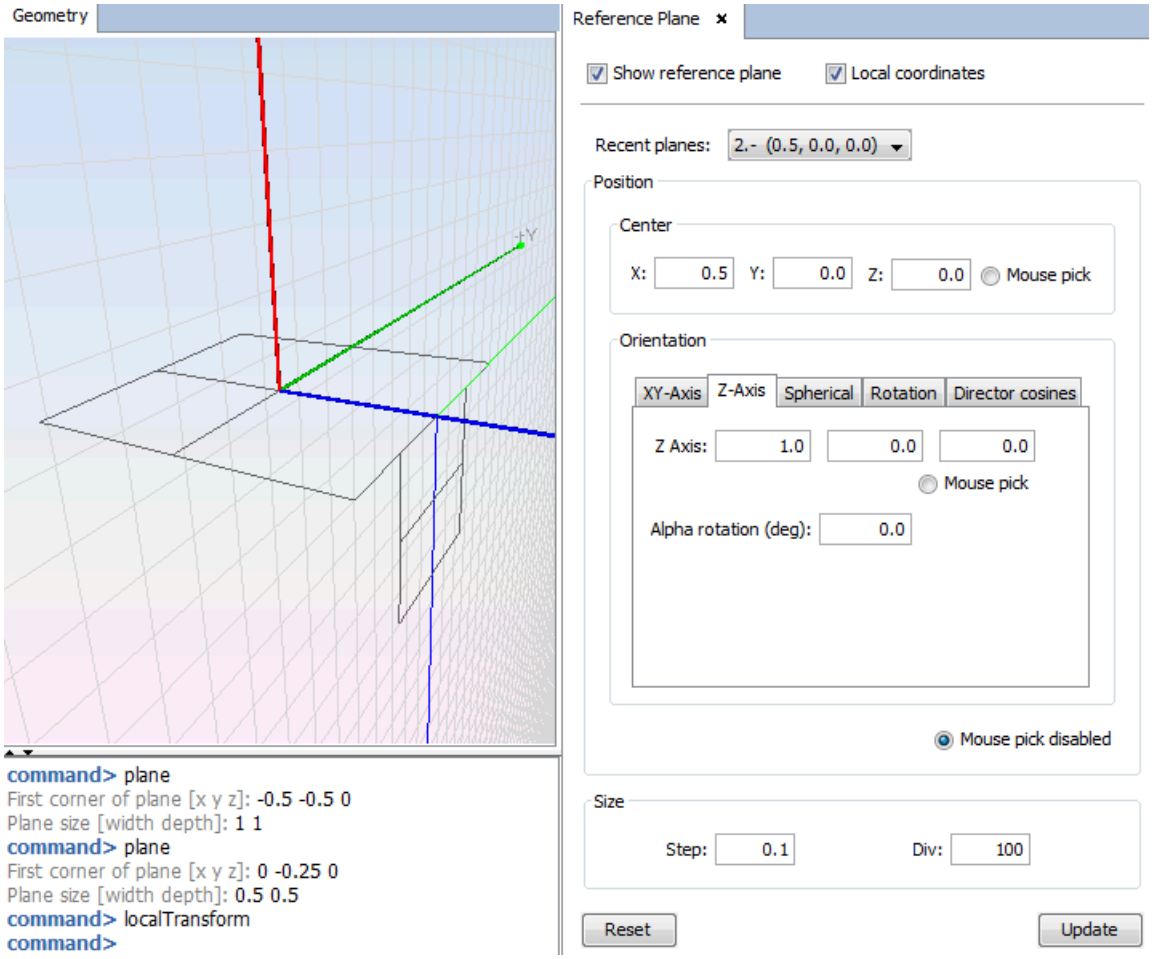


Figure 88: Dihedral after applying the 'localTransform' command to the second plane.

2.4.12 View Control Points

This menu allows you to change the control points of surfaces. It is possible to view and edit the component values for each control point part of a surface. To use this menu, a surface has to be selected. Objects cannot be selected unless they are exploded and a single surface part of the object is selected.

To use this feature, select a surface, and then select View Control Points from the Edit menu or click the toolbar icon.

A new window will appear on the right side showing a table with all the x, y, z components for each control point the surface is made of.

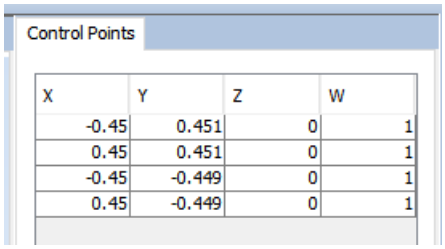


Figure 89: control point list

It is possible to select a row from this table in order to view the control point on the 3D viewer. For instance, click on a row to tint the control point to make it visible.

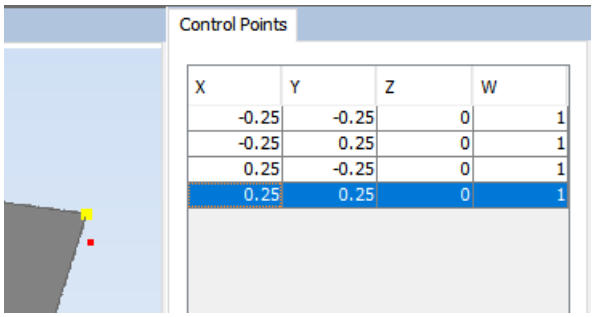


Figure 90: the control point is tinted yellow when the row is clicked.

Additionally, it is possible to click on a control point to select the row in the table.

If a control point gets selected, and then dragged using the mouse, it is possible to translate it, transforming the original geometry. This allows you to draw using the mouse the desired location of the control points. It is not possible to change the weight of such nodes using the mouse.

Alternatively, for a more precise behaviour, it is possible to double click a cell from the table and manually set the value for that cell. It is possible to change any cell, including position and weight for the control point.

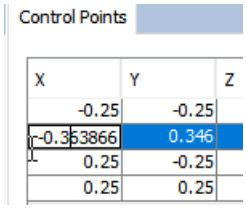


Figure 91: modifying the value for a component.

To exit this mode, simply select View Control Points again from the Edit menu or click the toolbar icon.

2.4.13 Invert Normals

The Invert Normals command menu allows you to invert the normals of objects and surfaces. The direction the normal vector of a surface is pointing to affects the way some core components of newFASANT such as the meshing process or the simulation process work.

In order to use this functionality, select an object or a surface and then choose Invert Normals from the Edit menu or from the toolbar. This will change the view to the Normal Editing view, displaying the normal vectors of the surface, or from all the surfaces the selected object is made of.

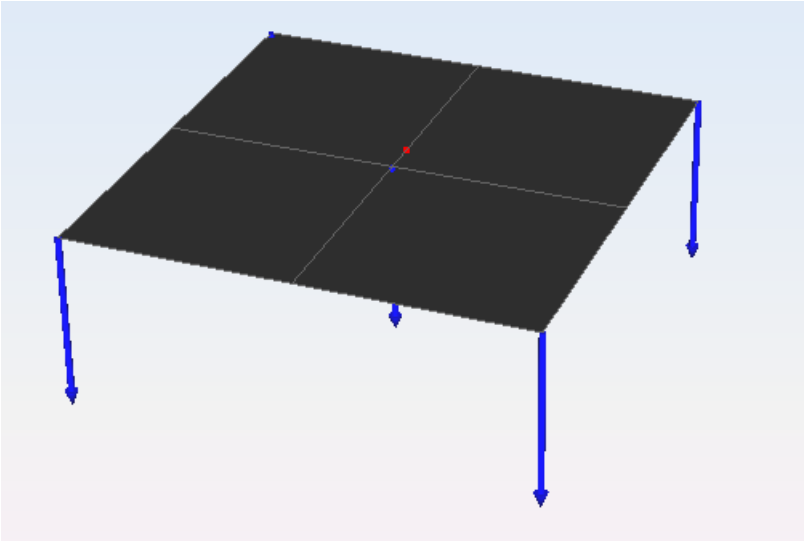


Figure 92: normal vectors for a plane.

To invert the normals now, click on the geometry panel. This will swap the orientation of the normal vectors so that now they are pointing to the other side.

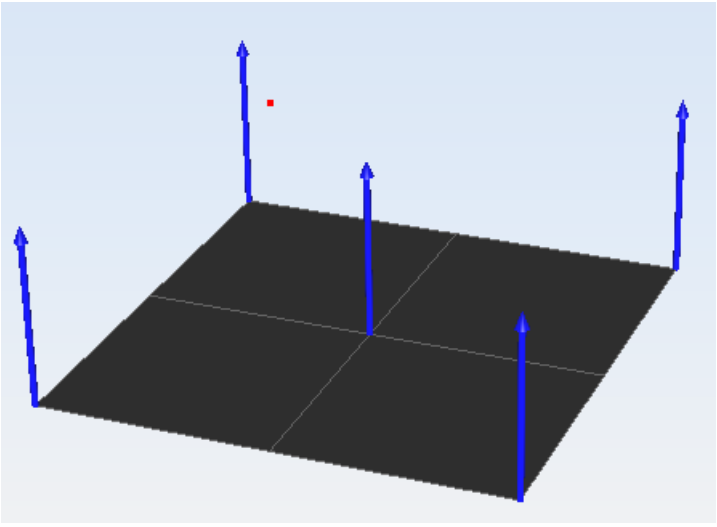


Figure 93: normal vectors of such planar surface once inverted.

Though, to save the new normal vectors, select again Invert Normals from the Edit menu or click the toolbar icon again. This will update the normals of the object

One common perception after inverting normals is that the brightness of the planar surface changes. The following screenshot was taken after inverting the normals. The camera was not panned nor rotated since. Note how the surface is now brighter than on previous screenshots.

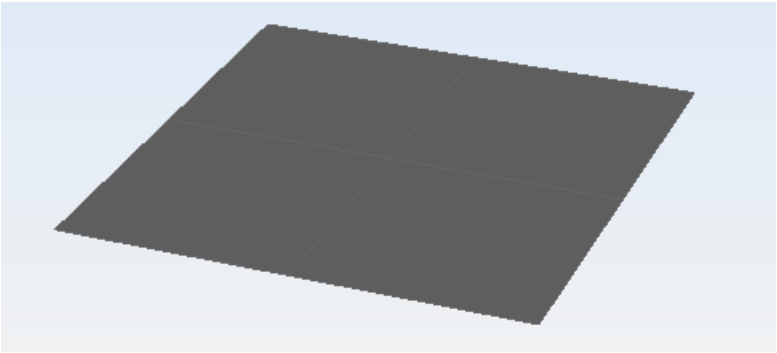


Figure 94: an inverted surface.

Surfaces are always brighter when being looked from the direction the normal vectors are pointing to.

2.4.14 Layers

The **Layer Menu** is opened when the user clicks on the submenu **Layers – Manage Layers.**

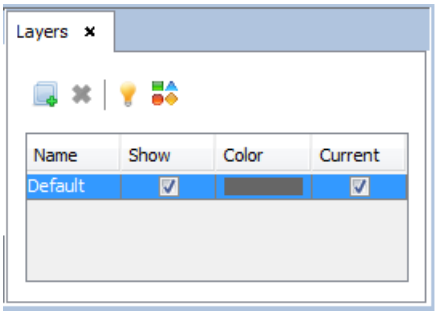


Figure 95: Layers Menu

The layers-based approach is useful to distribute the geometries into groups. Another advantage is the possibility of making a layer invisible for an easier way to visualize and/or work with the geometry.

Layers also allow the possibility to select the objects that are on the first visual level, that is to say, the objects that can be visualized without having others between them and the camera view.

In the following example, an imported geometry is considered. The first step is to explode the geometry, so that the user can select independently each component (click on Edit- Explode).

After exploding the geometry, it is possible to select independently every component and then to assign it to a new layer.

Clicking on layers around the object that is assigned to the default layer "Default". We can create new layers by clicking on **New Layers**. New layers appear with fields name, show, color and selected.

- **Name** clicking twice we can rename the layer.

- **Show** to make a layer visible or hide it.
- **Color** selects the color of the layer.
- **Current** to select the working layer where new objects will be inserted.

It is possible to delete layers by clicking on the layer and then clicking **Delete** button. The default layer can not be deleted. See next Figure.

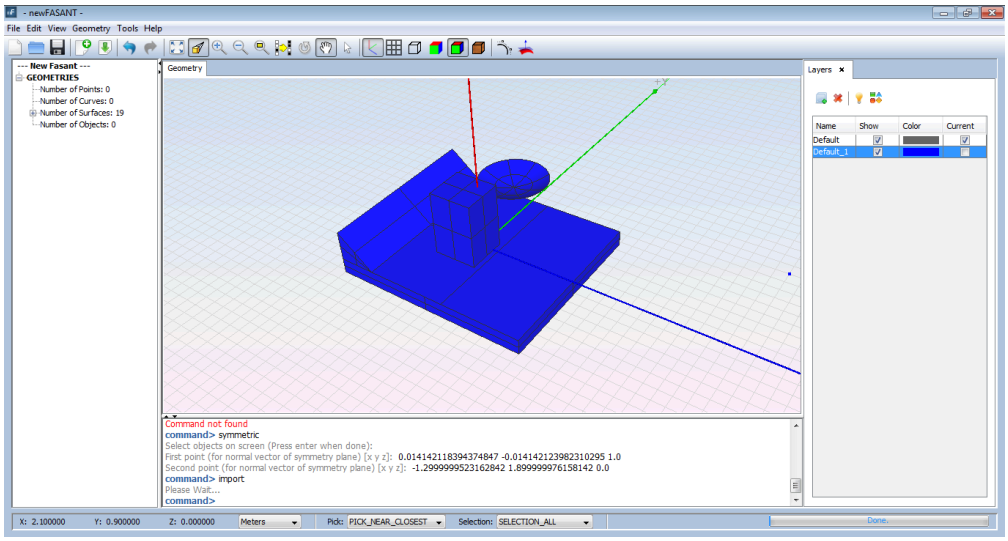


Figure 96: Layers

2.4.15 Object Properties

This option is used to visualize and edit some of the parameters of every geometrical element.

When selecting this option, and having selected an object, the following menu appears:

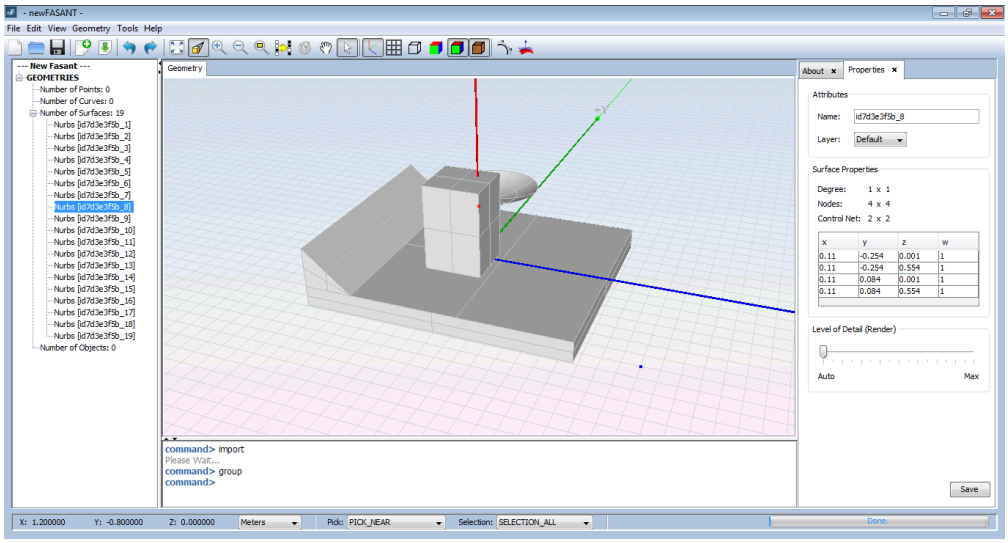


Figure 97: Object Properties Menu

The following parameters may be modified within this panel:

- Name objects may be renamed.
- Layer objects may be moved from their current layer to different ones.
- Level of Detail (Render) change this bar to increase or decrease the resolution of rendering.

By depending on the type of selected object, different properties that can not be modified within this panel are shown.

A table can also be seen in the bottom right indicating the properties of the surfaces.

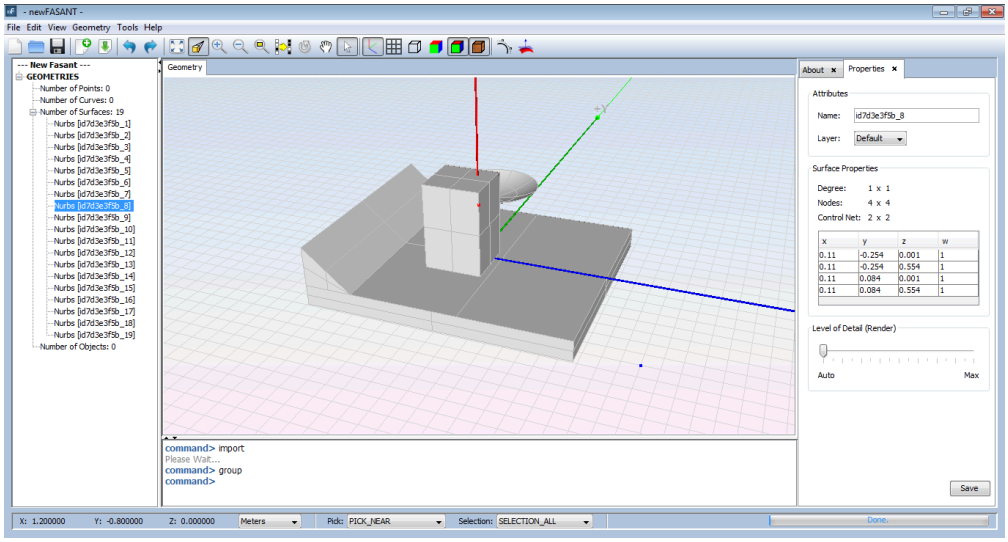


Figure 98: Object Properties

2.5 Geometry Menu

2.5.1 Importing and Exporting

When selecting a geometry, the following menu appears:

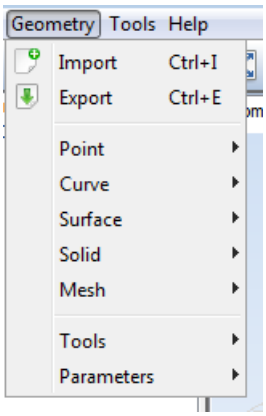


Figure 99: File Menu

The supported formats to be imported and exported by newFASANT are:

- NURBS file format (`.nur` extension), used only in the newFASANT software.
- IGES (Initial Graphics Exchange Specification, `.igs` extension) file format, widely used format for exchange of information among CAD software.
- STEP file format.
- DXF (Drawing Exchange Format) file format.
- MSH (Mesh) file format.
- STL (STereo Lithography) file format.
- NAS (NASA STRuctureANalysis extension) file format.
- Points file, in `.txt` format. List or points (every point in a different line) given by their X, Y, and Z in the same line.

Import (Ctrl+I)

The Import option allows the user to import an external geometry. After selecting this option, the window represented in the following Figure shows up asking for the file to import. Select the file and click on open.

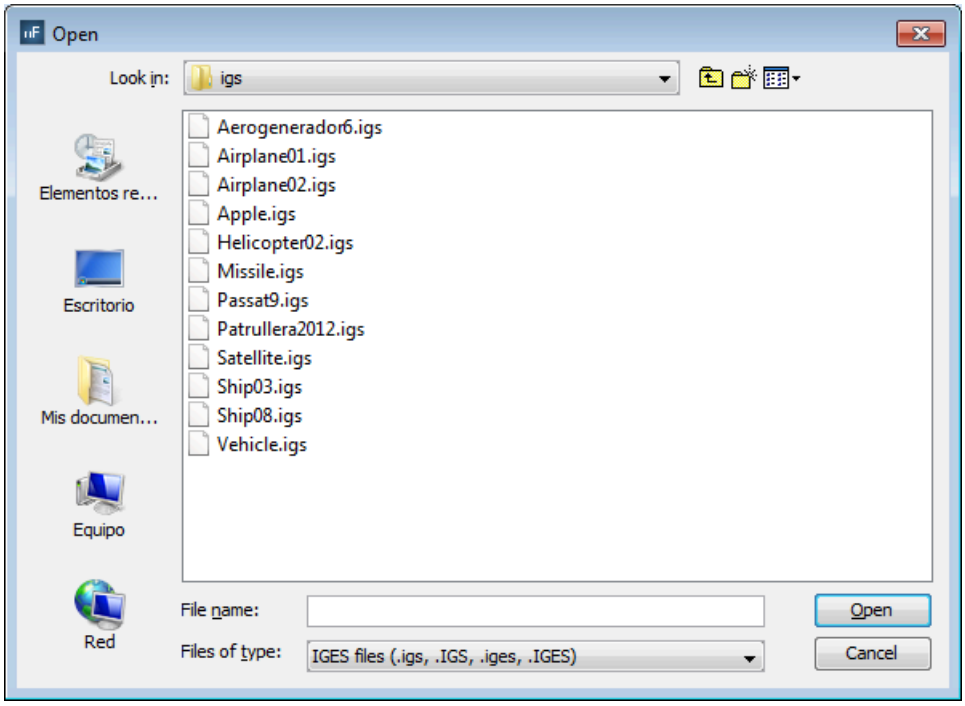


Figure 100: Open geometry dialog

Export (Ctrl+E)

The **Export** button lets the user export the geometry contained in the current project. Select the file and click on **Save**.

To export just the selected elements instead of the whole geometry, use the "**export -s**" command.

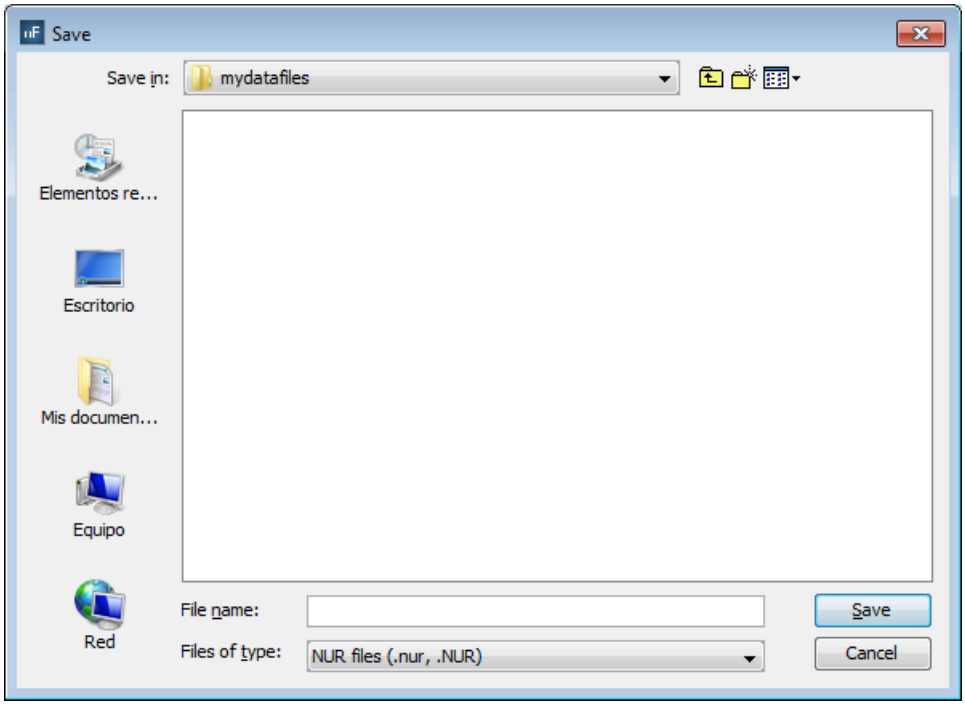


Figure 101: Export geometry dialog

2.5.2 Primitives

This section will explain the available primitives to generate the geometry. These primitives include points, curves, surfaces and solid volumes (objects). Some remarks to take into account while reading this section are:

- Every primitive is associated with a particular command. This means that the user is able to pass the parameters needed to draw the primitives using the command line. In most cases, the user can also pass these same parameters by clicking a specific point of the view area (e.g. it is possible to pass a point parameter to a command clicking on the corresponding point in the view area and pressing Enter). Note that for this functionality to work properly, it is required to select a Pick option different than “NO_PICK” in the bottom menu bar, as shown in the figure below.

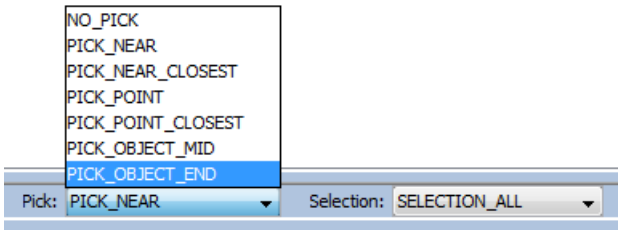


Figure 102: Pick options

- In some cases, figures can only be generated in a way that they are contained in a plane parallel to the XY plane (in other words, the points in the figure share the same Z coordinate). It may be necessary to rotate and move the figure to adjust its position to specific needs.

- The user may limit the geometric selection to the desired type of entities. The commands that require to select specific entities assign the **Selection** to the desired type automatically.

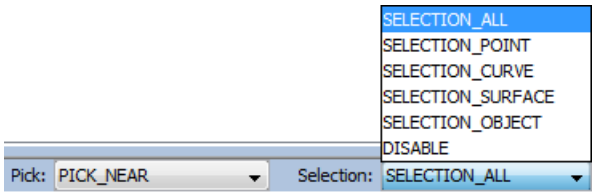


Figure 103: Selection options

Point

When selecting the **point** menu item, the following submenu is shown:

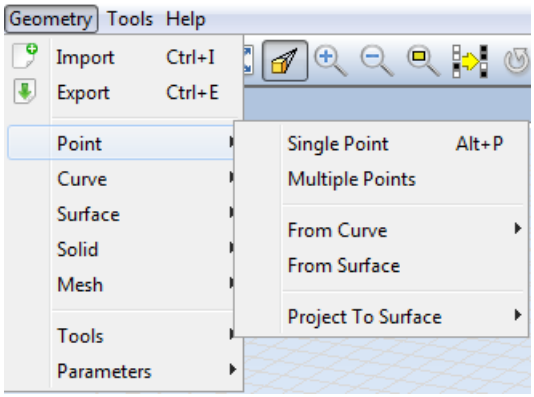


Figure 104: Geometry- > Point menu

Single Point (Alt + P)

With this option the user can build a point in the space by selecting directly on the geometry view and pressing Enter or by entering the point coordinates directly in the command line.

If the user decides to write the point directly in the command line, the components of the coordinate must be separated with a space.

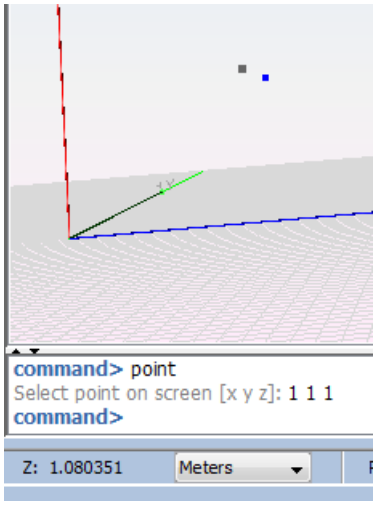


Figure 105: Drawing a point

Multiple Points

With this option the user can place several points. Each point is given by its three Cartesian coordinates. When as many points as desired have been inserted, the user can press the Enter button to generate them.

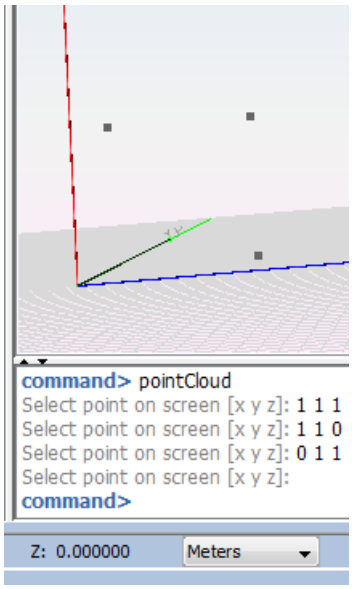


Figure 106: Drawing multiple points

From Curve

With this submenu the user can generate several points along a selected curve. There are two different options within this submenu that are explained in the following by considering a circle centered in the origin and with a radius of 0.5.

By Divisions

The points are generated along a curve as a result of its division in several subcurves (the curve is not split) of identical size. The user can select the number of points to be inserted.

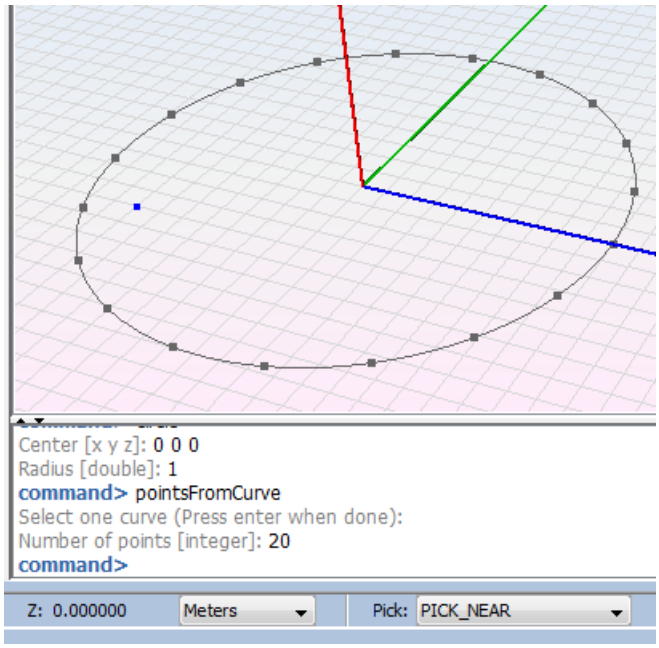


Figure 107: Points from Curve By Divisions.

By Distance

The points are generated along a curve when the desired length of a subcurve (the input curve is not split) is reached. The user will introduce the desired length of every step (no points are generated when the length of the curve is lower than the specified length).

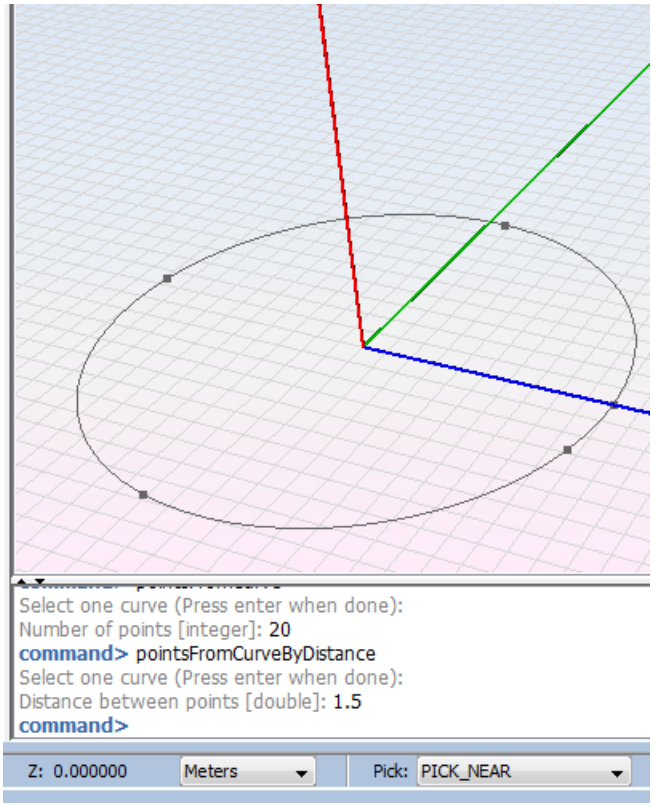


Figure 108: Points from Curve ByDistance.

From Surface

With this submenu the user can generate several points along the two parametric dimensions of a selected surface. The user needs to introduce the maximum number of points along the U-dimension and the V-dimension of the surface. When the selected surface is a trimmed surface, only the points in the valid region (by default) or all the grid points on the surface (-noTrim option) can be obtained.

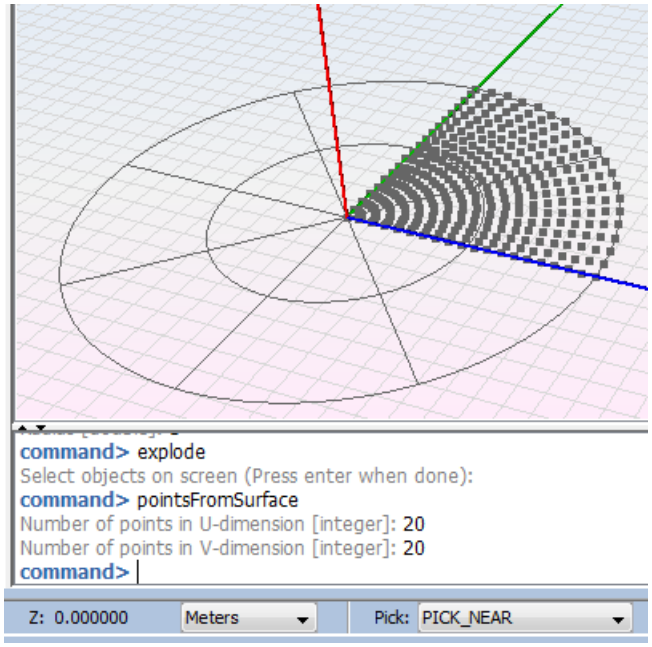


Figure 109: Points from Surface.

Project To Surface

This menu contains two options for projecting points onto surfaces.

To Closest Point

Every point is projected onto the selected surfaces. When multiple target surfaces are selected the points may be projected onto all surfaces, whenever the projections are perpendicular to the normal vectors.

This projection may be slow because the points are evaluated onto all surfaces.

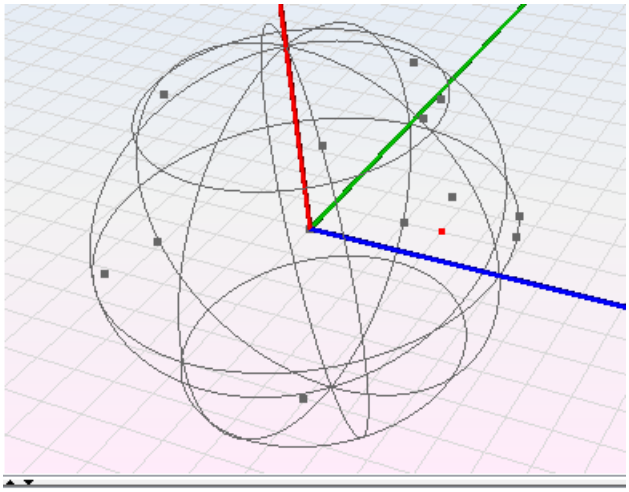


Figure 110: Project points to closest point.

By Direction

Every point is projected only onto the selected surfaces that are in the specified direction from the point.

This projection is faster than the closest one due to the direction check.

After selecting the target surfaces, the direction vector is required.

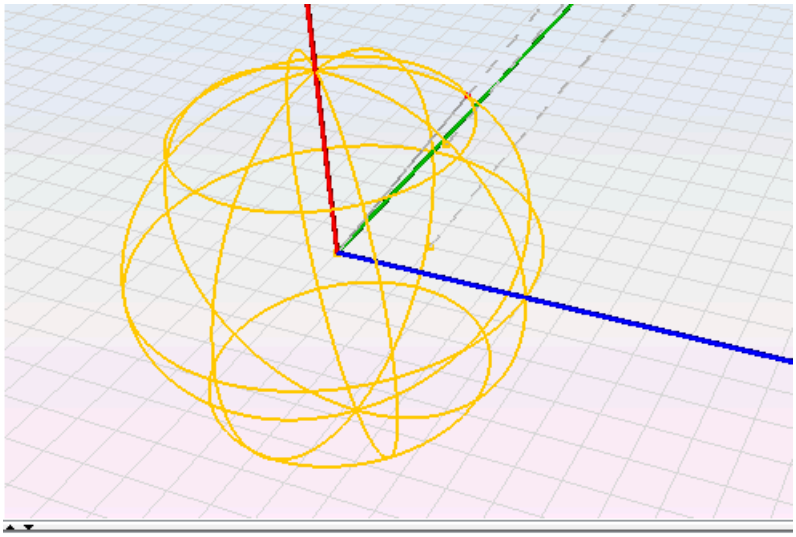


Figure 111: Project points by direction

Curve

This menu contains the following options:

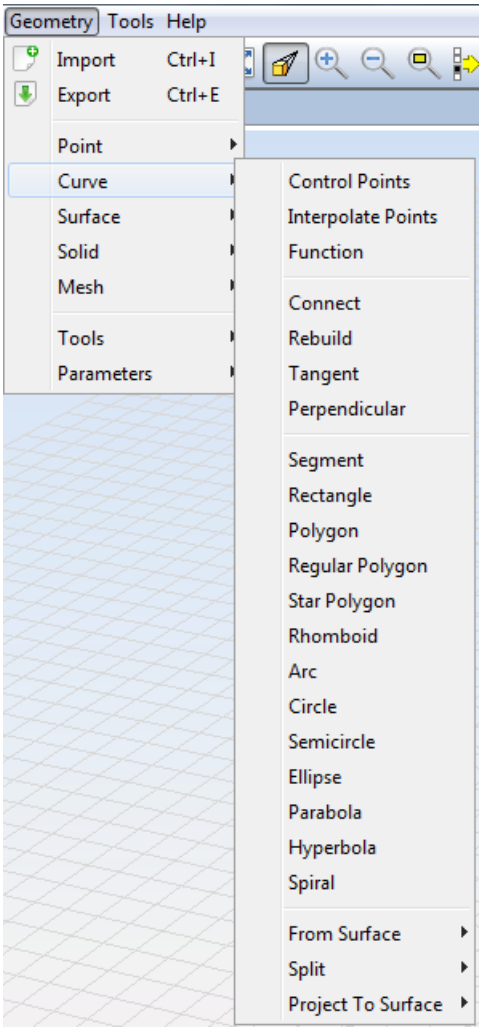


Figure 112: Curve submenu

Control Points

This option allows the user to generate a NURBS curve specifying its control points. After selecting this option, the console will require the degree of the curve (e.g. 1 for a linear NURBS curve, 2 for a quadratic NURBS curve and so on). After that, the user needs to specify as many control points (specifying their Cartesian coordinates as well as the weight associated with each point) as wished. When enough points are inserted, the user can press Enter to finish.

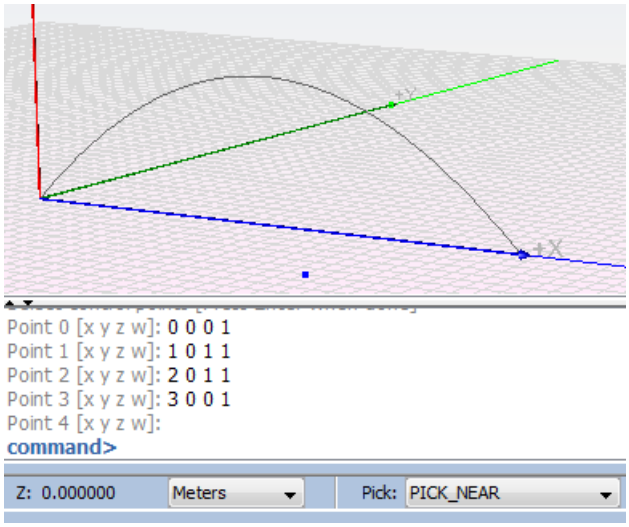


Figure 113: Drawing a cubic NURBS curve

Interpolate Points

This option allows the user to generate a curve that goes through a set of points by applying polynomial interpolation. After selecting this option, the program will ask the degree of the polynomial to use and the actual points to be interpolated (note that a minimum of points are needed for the interpolation to be possible). When enough points are specified, the user can press the Enter key to finish.

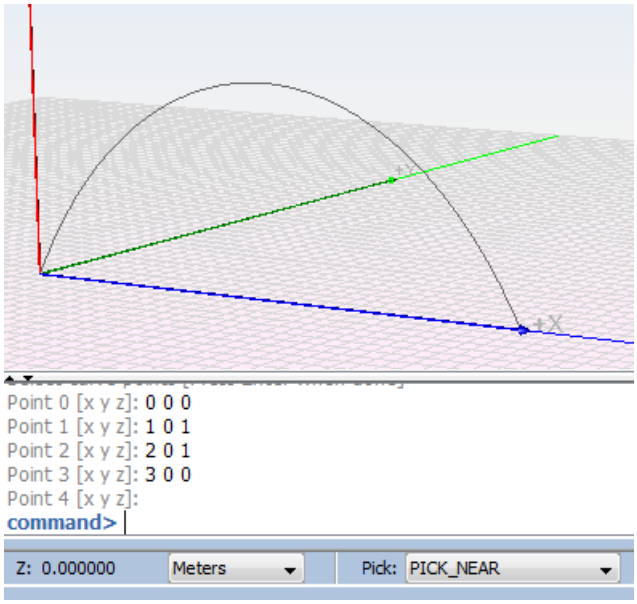


Figure 114: Drawing a curve using cubic interpolation

Function

This option creates a curve by defining a parameter and three functions. To generate the curve it is necessary to define the '$u' parameter. With this parameter, the user defines three functions f($u), for the x, y, z coordinates. The resulting curve will pass through all the points.

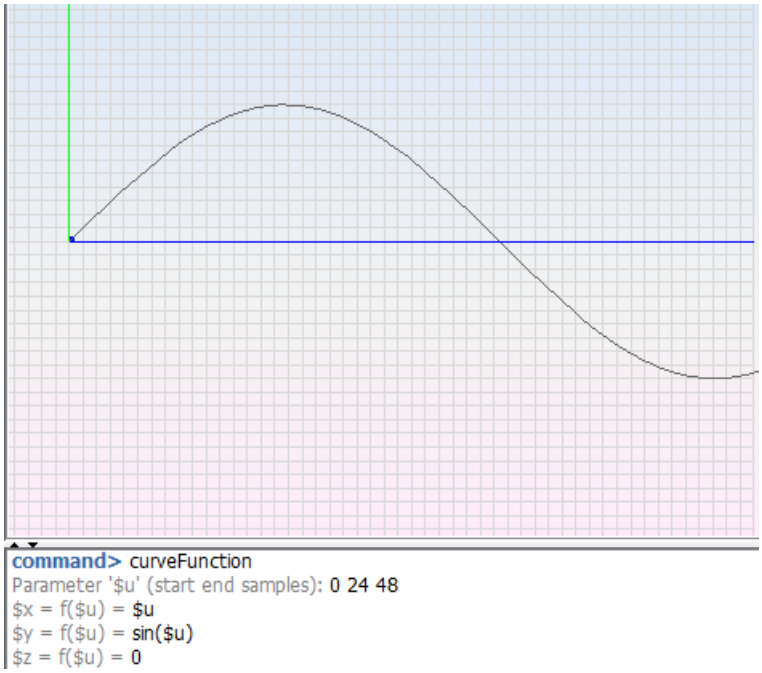


Figure 115: Drawing a curve using function

Connect

This options allows the user to connect two distinct curves (that are in contact at least in one pont) into one single curve. By using this operation, the two curves are combined in just one curve. The following example shows the connection of two arcs to get a circle.

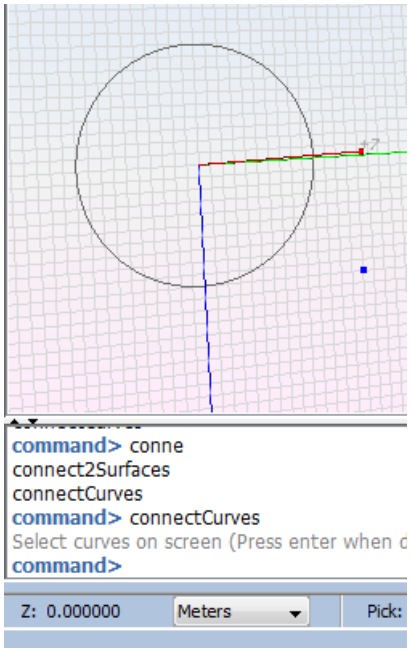


Figure 116: ConnectCurves function

Rebuild

This option allows rebuilding any given curve by sampling and interpolating the resulting points. The user needs to specify the degree of the new curve and the number of samples to take into account on the original curve.

The figure above shows the example of a rebuilt arc.

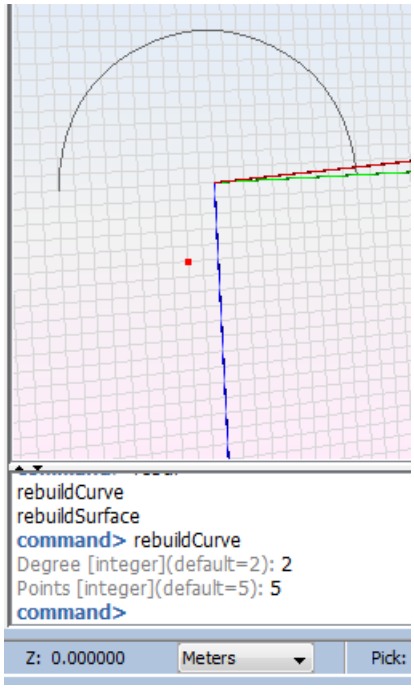


Figure 117: Rebuild a Curve from an original arc

Tangent

This option generates tangent segments to the selected curve that cross the introduced point. Only one curve must be selected.

If the introduced point is on the curve, only one tangent segment is generated; otherwise, more than one segment may be inserted.

After confirming the point to across with the tangent line, the length of the segment is required. Positive or negative lengths determine the orientation of the resulting segments.

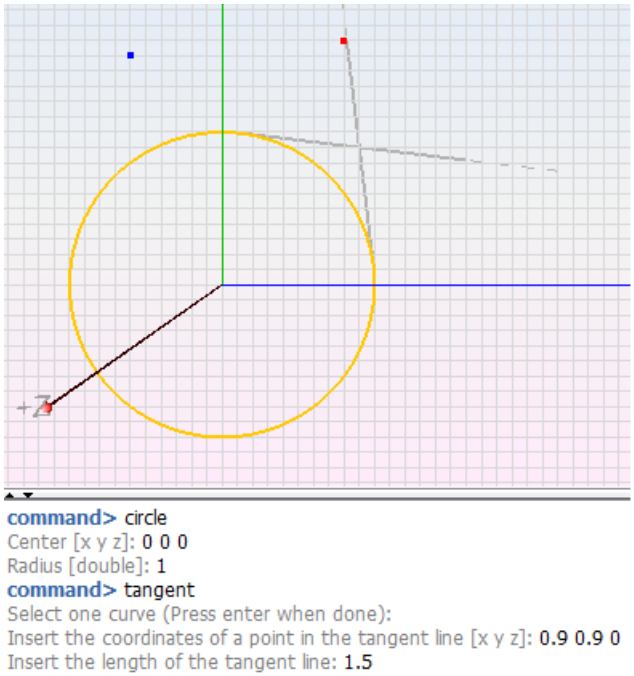
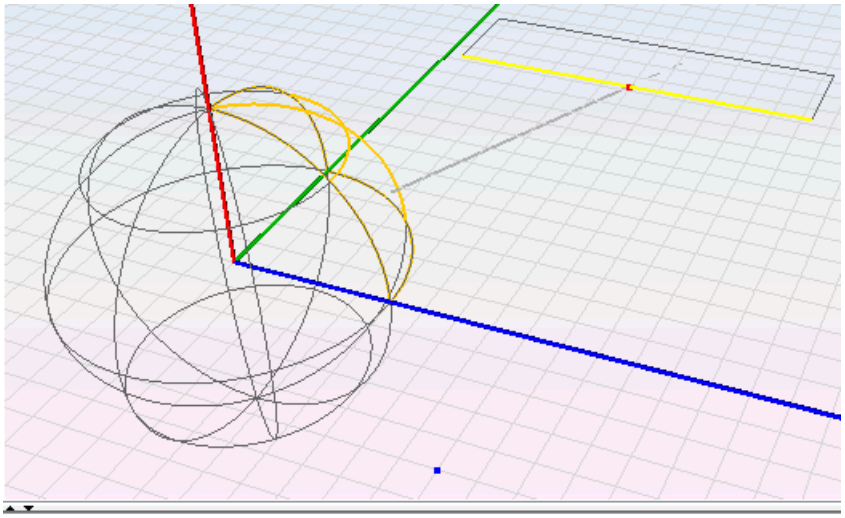


Figure 118: Example of tangent lines from a circle.

Perpendicular

This option generates a normal segment to the selected curve or surface that cross the introduced point. Only one curve or surface must be selected.

After confirming the point to across with the perpendicular line, the length of the segment is required. Positive or negative lengths determine the orientation of the resulting segments.

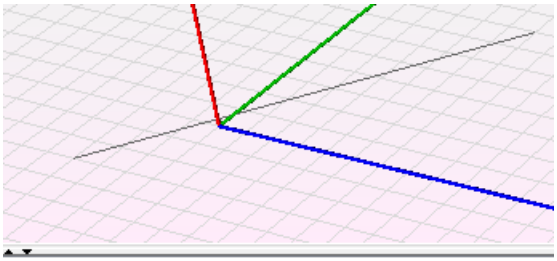


```
command> sphere
Select center [x y z]: 0 0 0
Radius [double]: 0.5
command> rectangle
First corner of rectangle [x y z]: 0.5 0.5 0.5
Size [width depth]: 1 0.25
command> explode
command> perpendicular
Insert the coordinates of a point in the perpendicular line [x y z]: 1.0 0.5 0.5
Insert the length of the perpendicular line: 0.7247448713915889
```

Figure 119: Example of perpendicular line from a sphere.

Segment

This option allows to create a segment between two points specified by the user.



```
command> segment
First point [x y z]: -0.3 -0.3 0.0
Second point [x y z]: 0.6 0.8 0.0
command>
```

Figure 120: Drawing a segment

Rectangle

This option allows the user to create a rectangle by specifying the coordinates of two opposite corners.

Remark the second corner point implicitly assumes the same z coordinate as the first point.

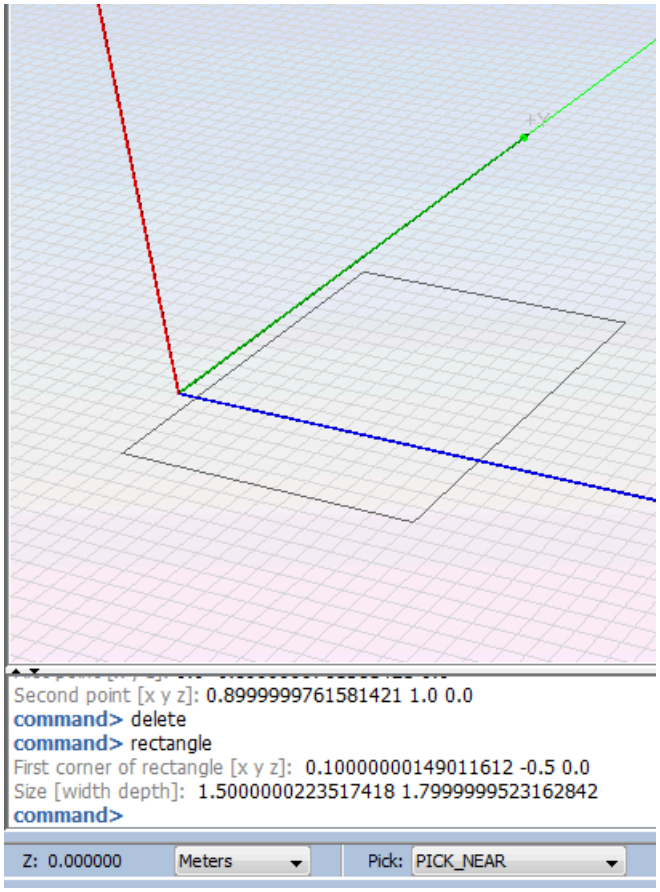


Figure 121: Drawing a rectangle

Polygon

This command allows the user to create a polygon specifying its vertices.

Remark The second and following points assume implicitly the same z coordinate as the first point.

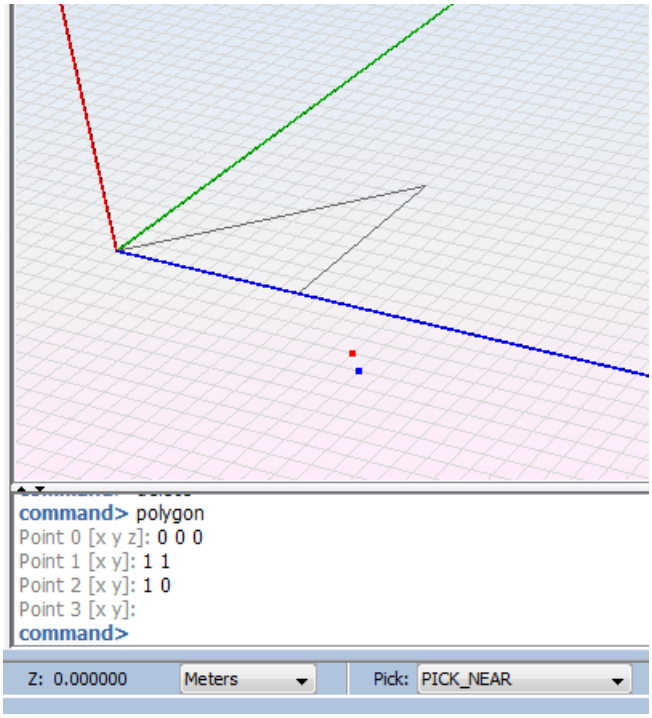


Figure 122: Drawing a polygon

Regular Polygon

This option allows the user to create a regular polygon specifying the number of sides, the center and the radius of the circumference it is inscribed in.

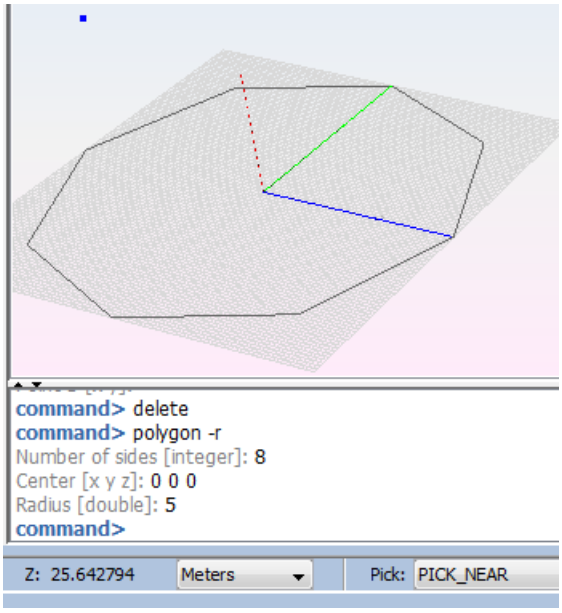


Figure 123: Drawing a regular octagon

Star Polygon

This option allows the user to create a star polygon specifying the number of vertexes, the center and the two radius of the circumference they are inscribed in.

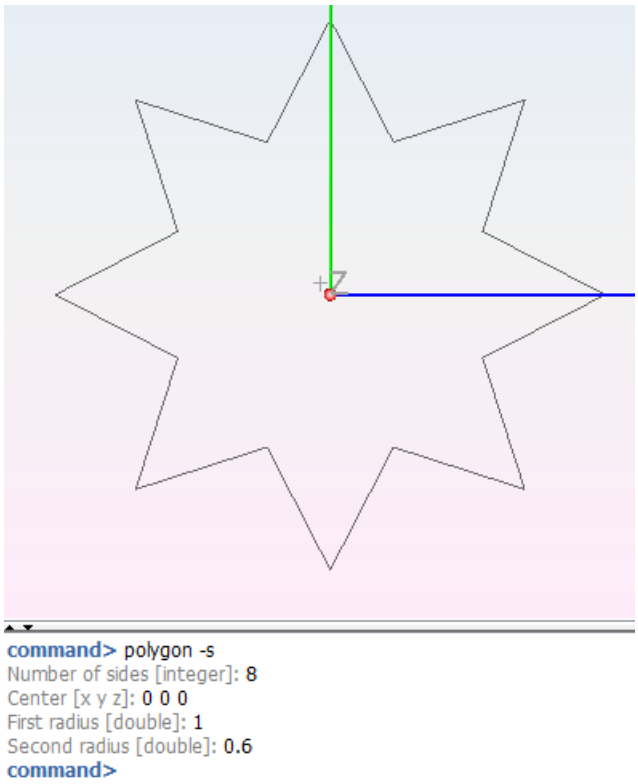


Figure 124: Drawing a star of eight vertexes.

Rhomboid

This command allows the user to create a rhomboid specifying three of its vertices.

Remark The second and third points assume implicitly the same z coordinate as the first point.

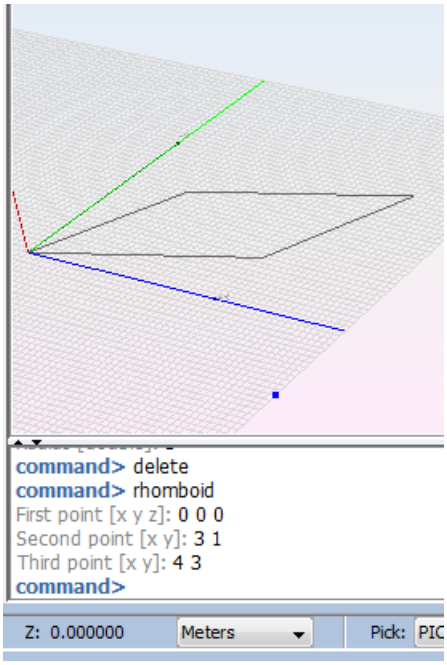


Figure 125: Drawing a rhomboid

Arc

This command allows the user to create a circumference arc specifying its central point, radius, initial and final angles (in degrees), in this order.

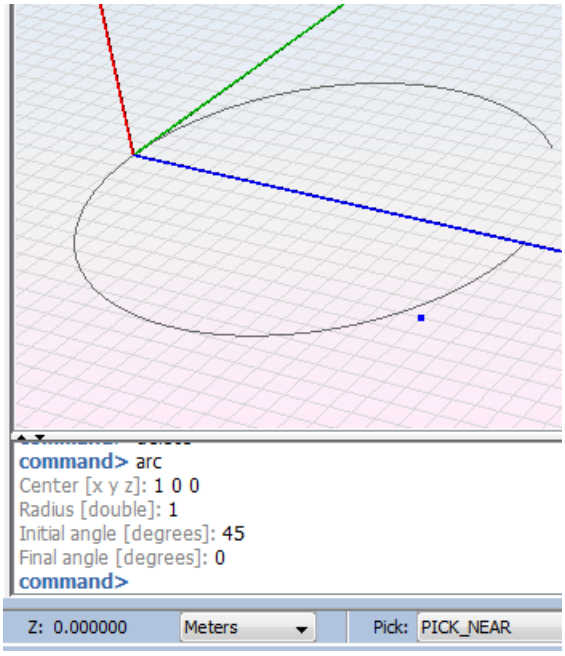


Figure 126: Drawing an arc

Circle

This option creates a circumference around a center point and with a given radius, both specified by the user.

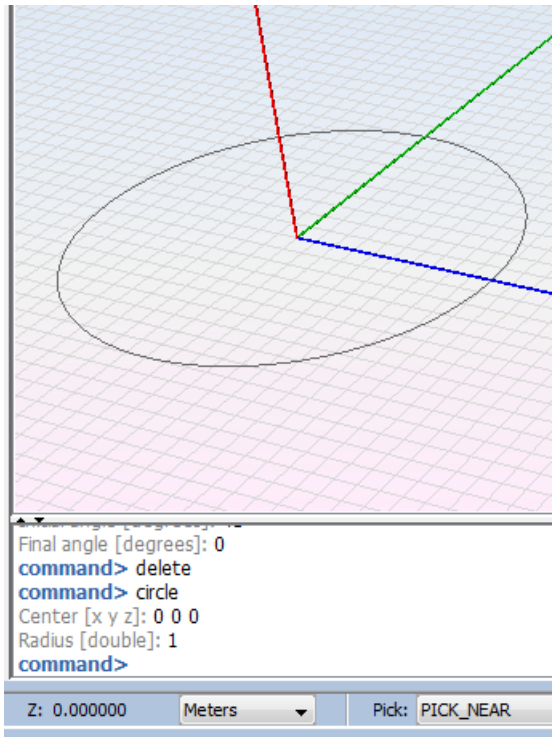


Figure 127: D rawing a circle

Semicircle

This option creates a semicircle around a center point and with a radius, both specified by the user. This function also receives as a parameter the direction in which the arc of the semicircle is generated if the angle is 0, the semicircle is created in the positive Y axis direction, and greater angles describe a counter-clockwise rotation around the Z axis).

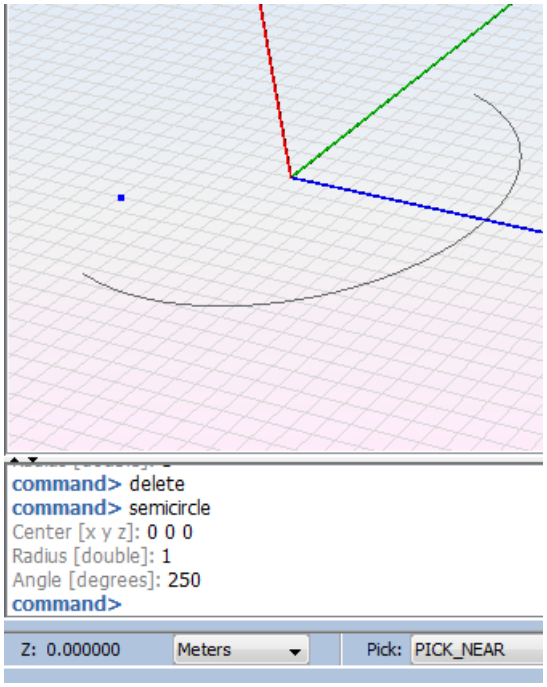


Figure 128: Drawing a semicircle

Ellipse

This option allows the user to create an ellipse by specifying its center and both radiuses.

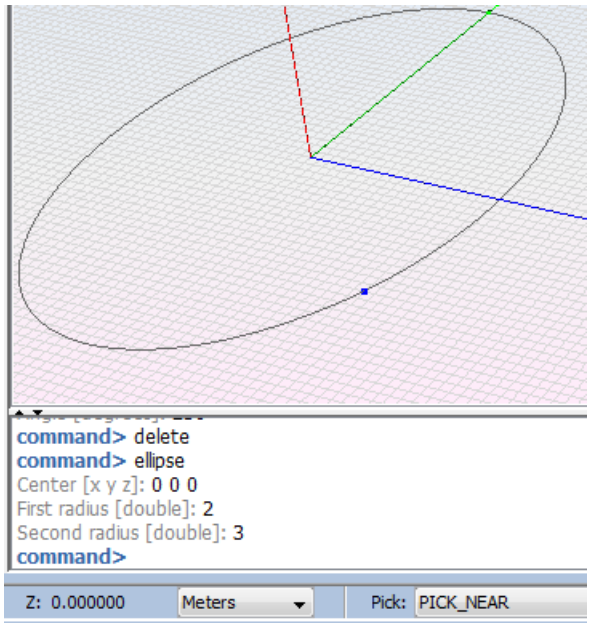


Figure 129: Drawing an ellipse

Parabola

This option allows the user to create a parabola. The parameters to this function are directrix center point of the parabola, focus point and corner point. The directrix point determines the axis of symmetry of the parabola. The focus of a parabola is a fixed point on the interior of a parabola used in the formal definition of the curve . The end point determines the length and curvature of the parabola.

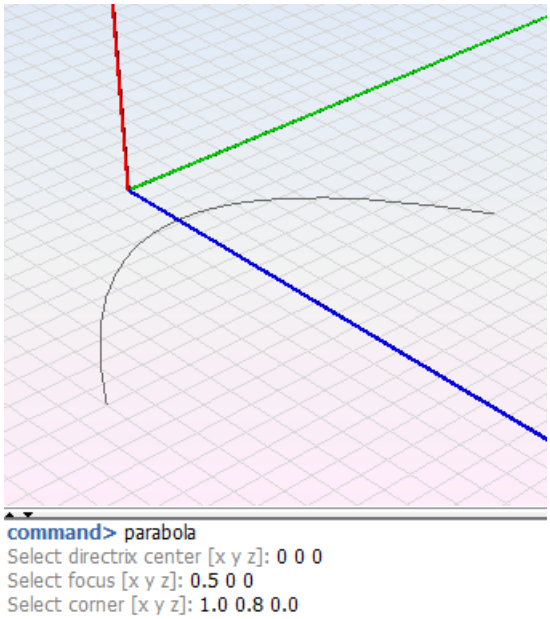


Figure 130: Drawing a parabola.

Hyperbola

This option allows the user to create a hyperbola. The function receives the following parameters three points that represent the directrix center, focus and corner points.

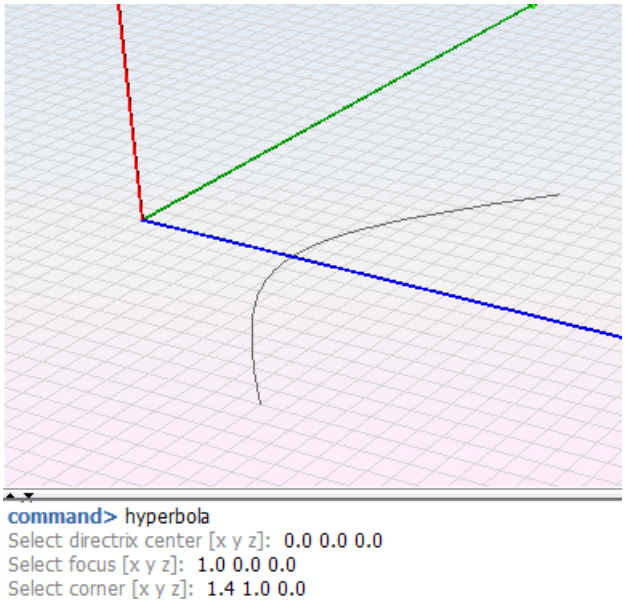


Figure 131: D rawing a hyperbola.

Spiral

The primitive spiral generates a coplanar curve that turns from the center to the end point, specified by the radius and the number of turns. The first point of the curve is at the center and the last point is at the radius distance.

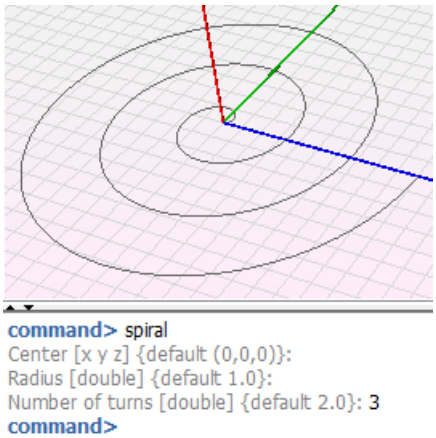


Figure 132: Spiral example of 3 turns and counter-clockwise direction

By default, the generated spiral is generated in counter-clockwise direction. To generate spirals in clockwise direction, the parameter "-rightTurned" must be included.

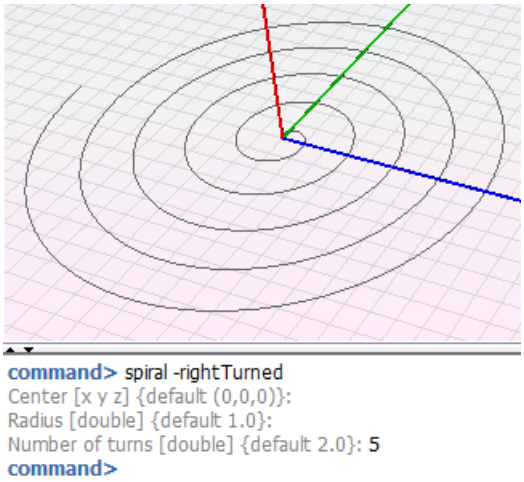


Figure 133: Spiral example of turns and clockwise direction

From Surface

This submenu contains several options to obtain curves from existing surfaces.

Edges

This option allows obtaining the curves that bound the selected surfaces (multiple surfaces and objects may be selected). After selecting the surfaces to be considered, the corresponding curves appear as selectable curves to be added.

- If no curves are selected, all the curves are added to the geometry.

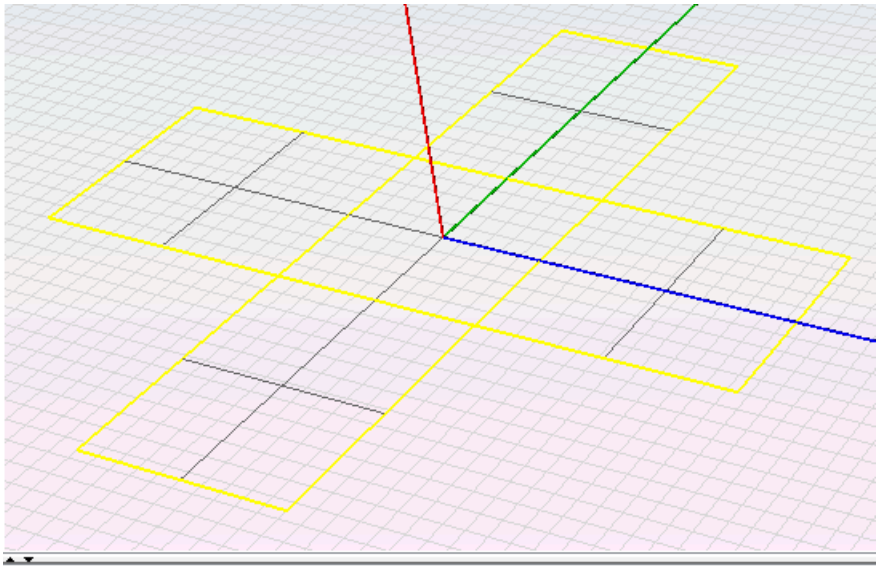


Figure 134: Extraction of all the curves from a cross primitive

- If one or more curves are selected, only the selection is added to the geometry. Multiple curves may be selected by using the Ctrl-Key and the Left-Mouse button.

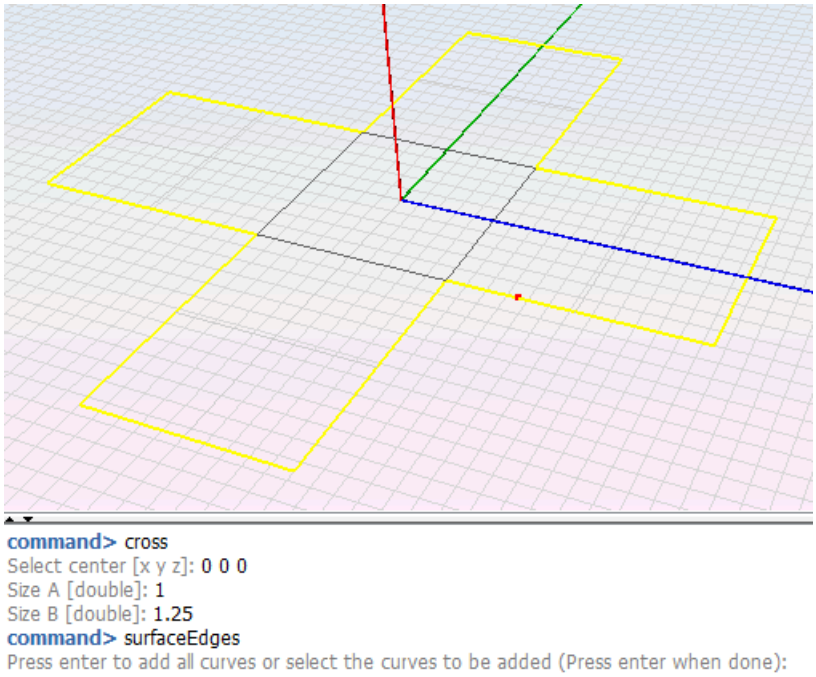


Figure 135: Selection of the outer curves from a cross primitive to be extracted

Isoparam

Any isoparametric curve of a surface (in the U- or V-dimension) may be obtained by using this command. To do this, only one surface will be selected. Then, tip “u” or “v” to change the parametric dimension where the curve will be extracted and insert the parametric value of the desired coordinate (pick mouse functions make easier this operation).

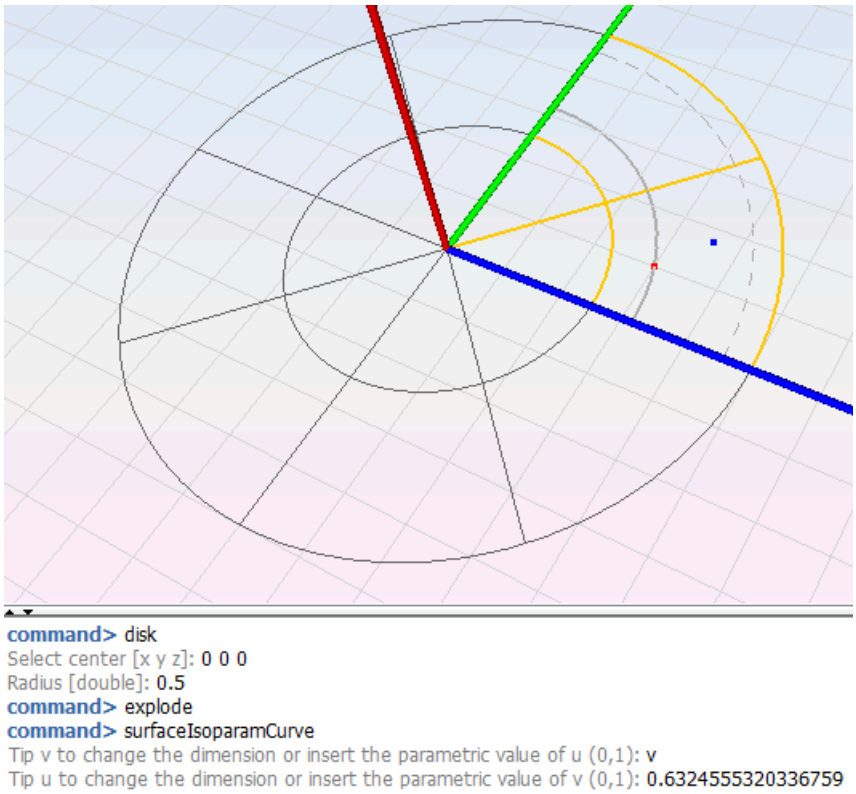


Figure 136: Isoparametric curve from a disk-sector

Split

This menu contains the available options for splitting curves.

By Curves

Select this option when multiple curves must be mutually split. Every curve will be divided in as many sections as required by the intersecting curves.

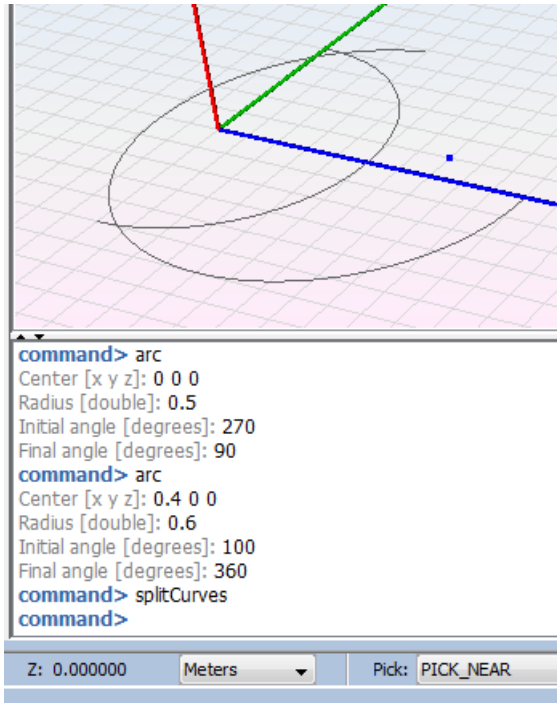


Figure 137: Mutual split of two arc curves

Parametric

This option is useful when a curve need to by split at a point on the own curve. The point is specified as a parametric coordinate (between 0 and 1), and the pick mouse functions may be useful to split the curve close to any desired point.

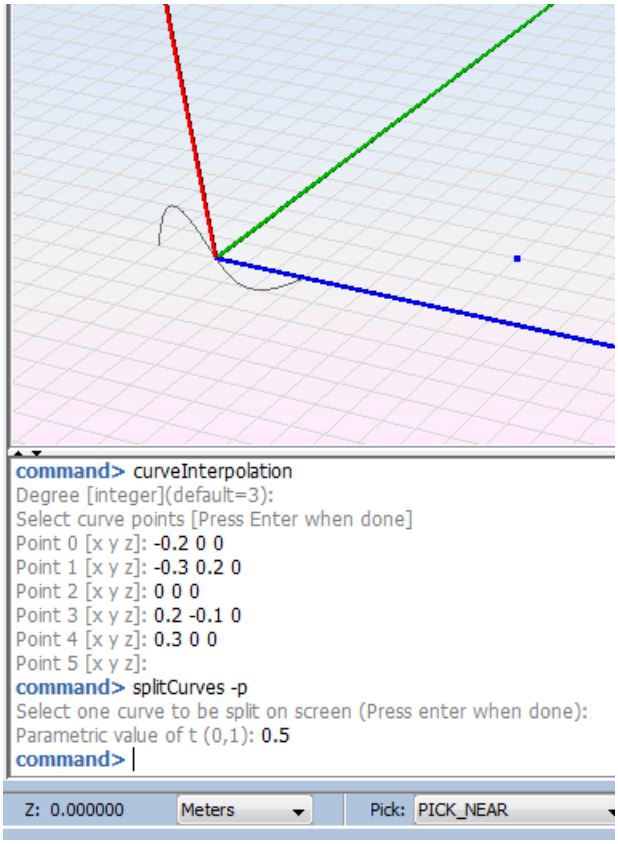


Figure 138: Split a curve by a parametric coordinate

By Cutting Objects

Select this option when only some curves must be split by different objects (the parameter–cis used). All the curves to be split must be selected in the first step, and after confirming the selection, the cutting objects are selected.

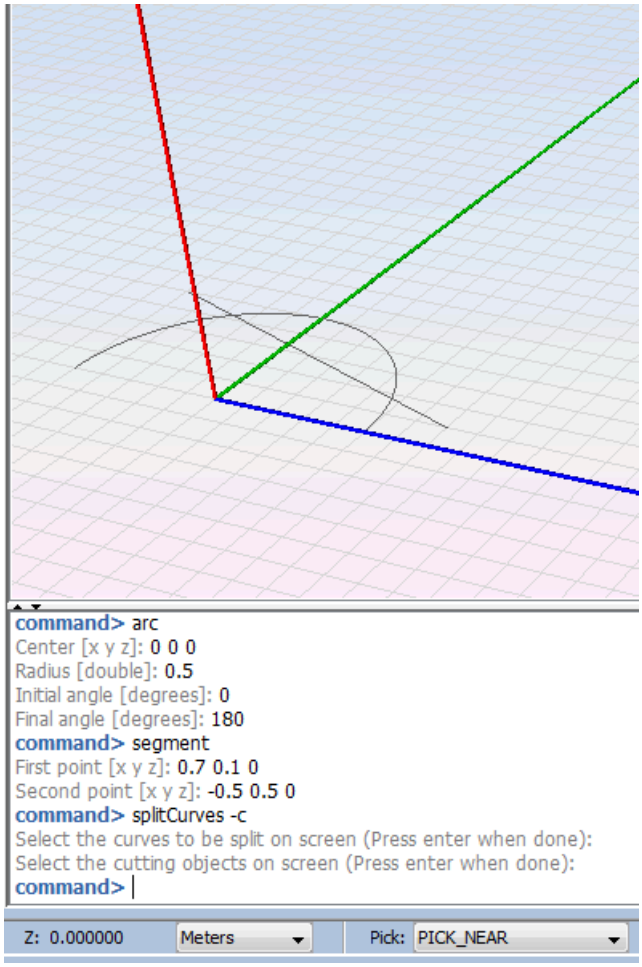


Figure 139: The arc or the segment is split by the other curve

Project To Surface

This menu contains two options for projecting curves onto surfaces.

To Closest Point

Every curve is projected onto the selected surfaces. When multiple target surfaces are selected the curves may be projected onto all surfaces, whenever the projections are perpendicular to the normal vectors.

This projection may be slow because the curves are evaluated onto all surfaces.

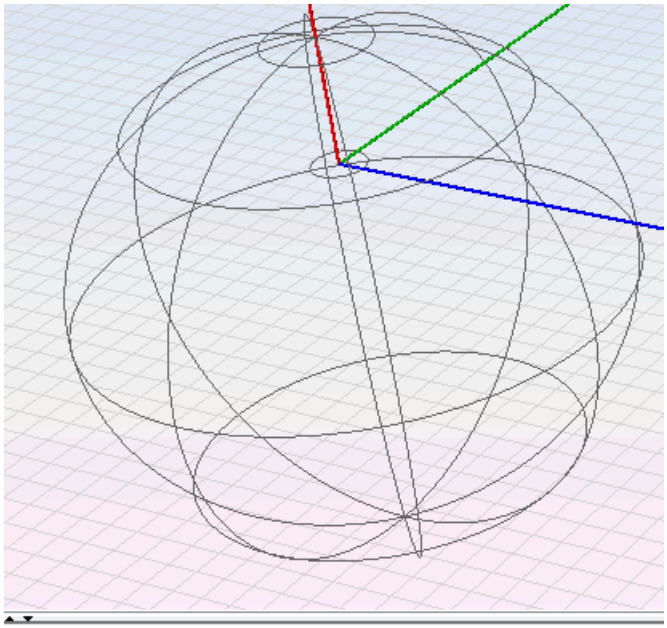


Figure 140: Project curves to closest point

By Direction

Every curve is projected only onto the selected surfaces that are in the specified direction from the curve.

This projection is faster than the closest one due to the direction check.

After selecting the target surfaces, the direction vector is required.

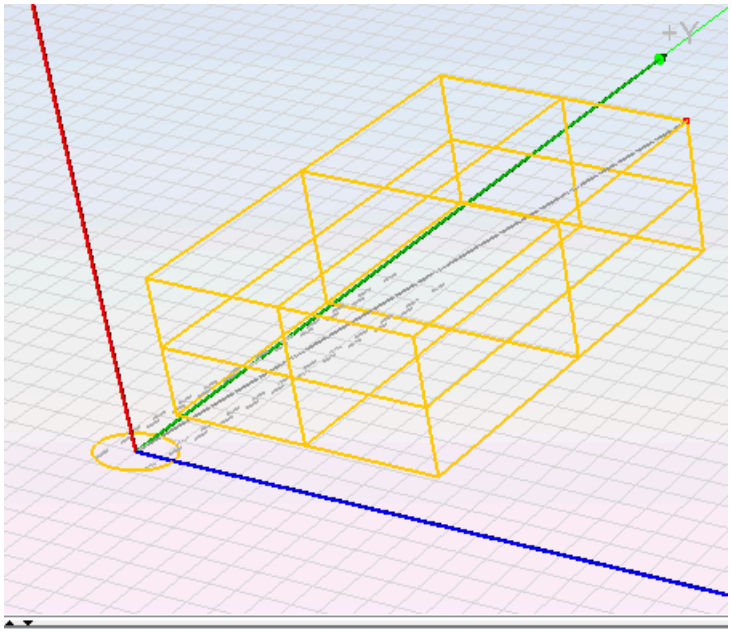


Figure 141: Project curves by direction

Surface

This menu contains the following options:

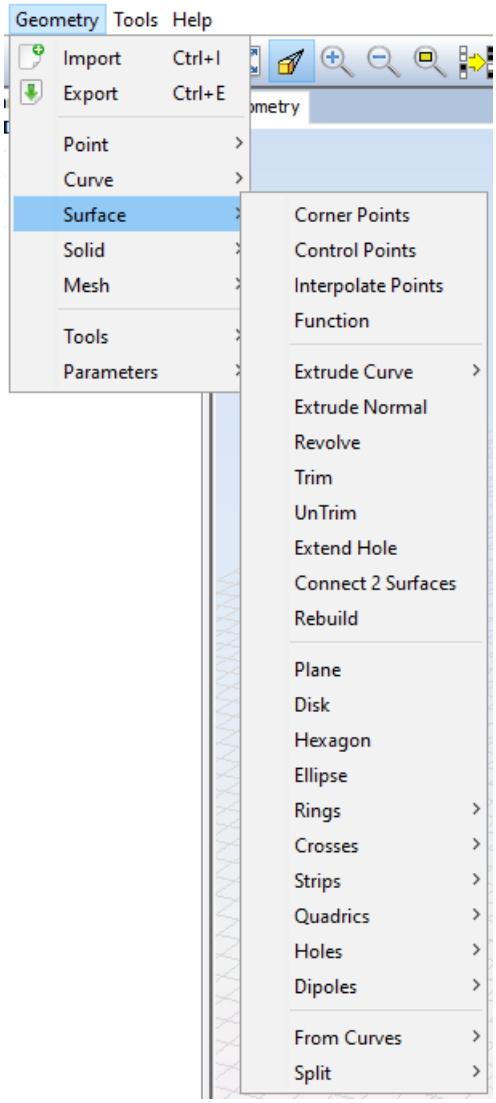


Figure 142: S urface submenu

Corner Points

This option creates a new surface of three or four vertex from the given corners. The command requires the four vertexes in order. Insert no coordinates in the fourth vertex to get a triangular surface.

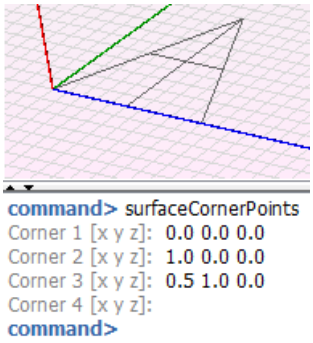


Figure 143: Surface from 3 corner points

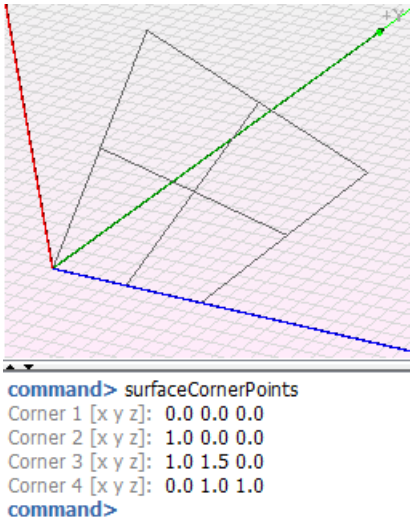


Figure 144: Surface from 4 corner points

Control Points

This option allows the user to create a NURBS surface. The parameters that this function receives are the number of points in u and v direction (this determines the degree of the NURBS surface) and the points themselves. The number of points the program will ask for will be the product of the number of points in each direction.

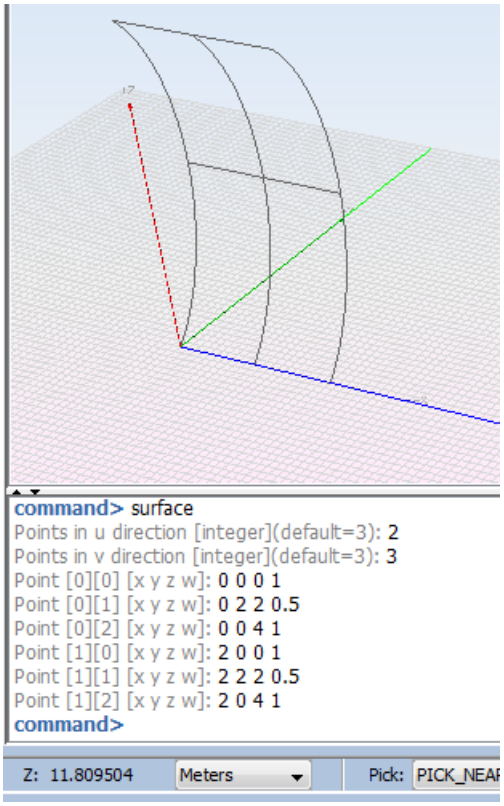


Figure 145: Drawing a NURBS surface

Interpolate Points

This option allows to create a surface that passes through the points specified by the user, using interpolation. The parameters of this function are the number of points in each direction of the surface and the points themselves.

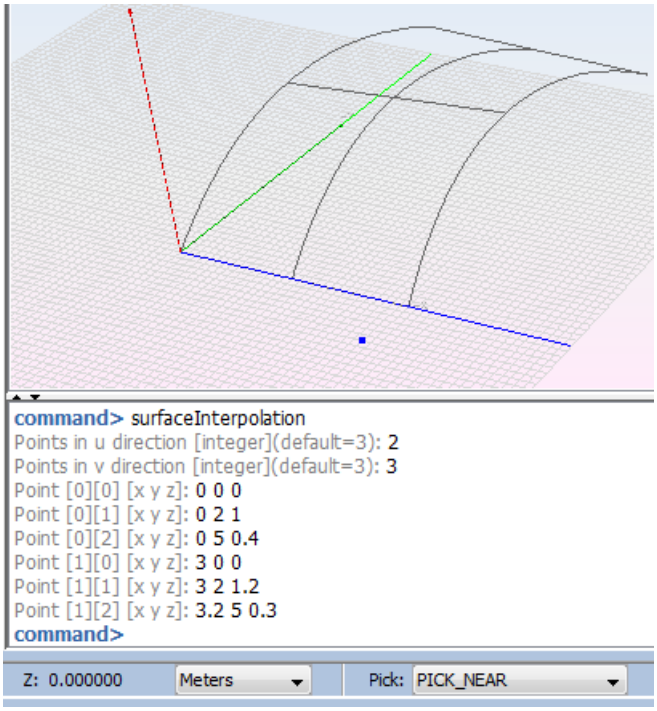


Figure 146: Drawing a surface using interpolation

Function

This option creates a surface by defining two parameters and three functions. To create the curve is necessary to define the '$u', '$v' parameters. With these parameters, the user will define three functions f($u, $v), for the x, y, z coordinates. The resulting surface will pass through all of the points.

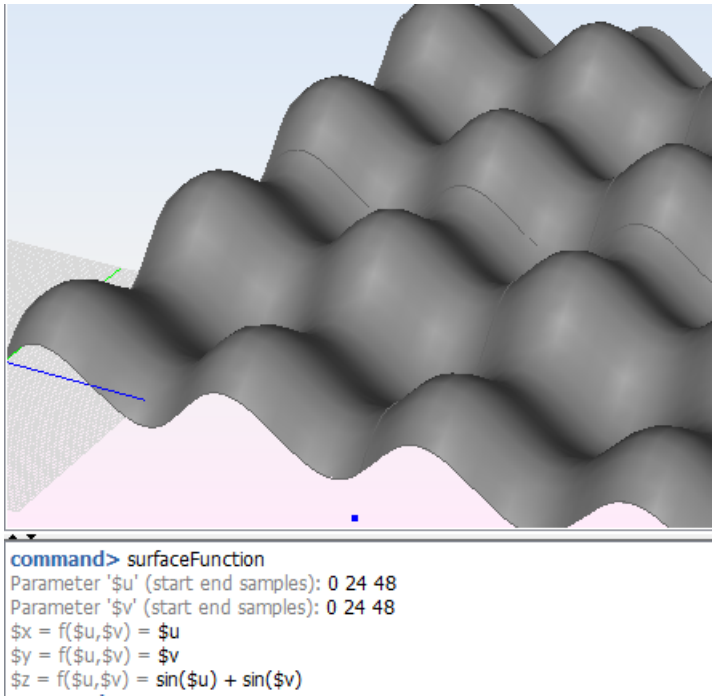


Figure 147: Drawing a surface using functions

Extrude Curve

These options allow creating a surface based on a curve. There are two different ways to do this:

- **Straight** This function creates a surface by applying a translation vector to an existing curve. The parameters this function receives are the curve to be extruded (the user can select it by clicking it on the view area or by selecting it in the tree structure), a start point and an end point. The vector that goes from the start point to the end point will be the translation vector applied to the curve.

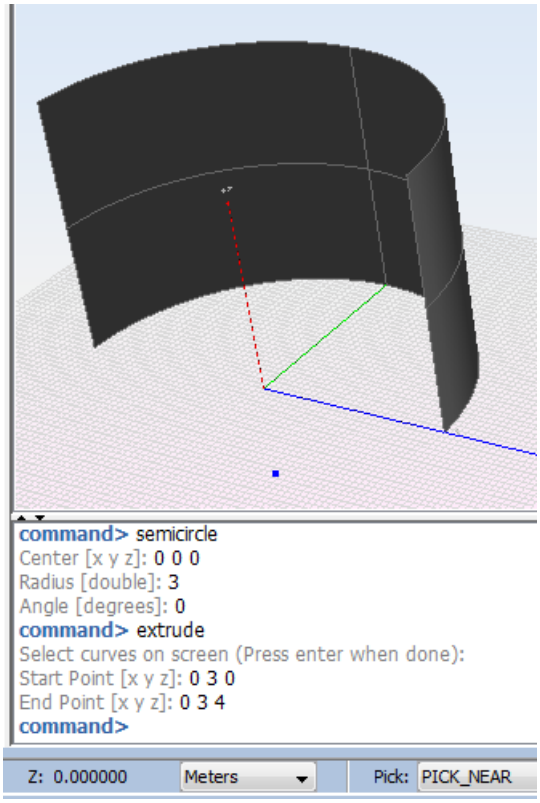


Figure 148: Extrude straight

- **To point** This function creates a surface extruding a curve to a point. This is, given a curve to extrude and an extrusion point, the function creates the surface resulting from connecting all the points in the curve to the extrusion point.

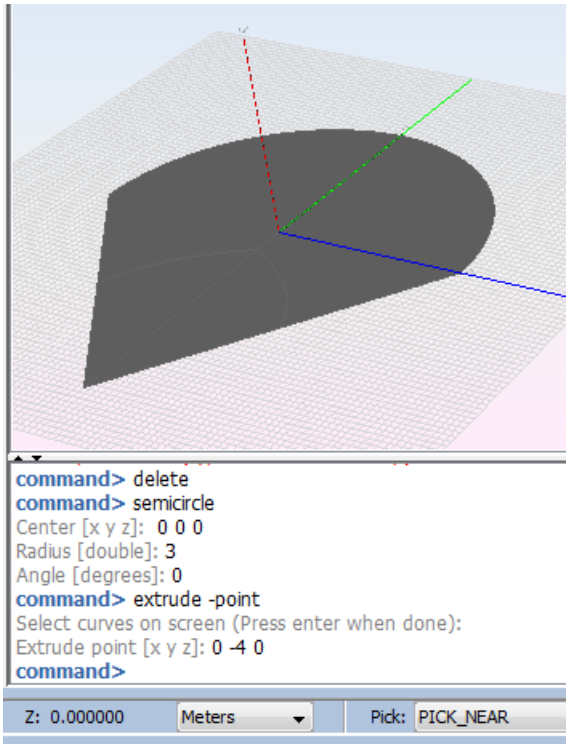


Figure 149: Extrude to point

Extrude Normal

This function returns an object with the surfaces resulting of extruding the input surfaces according to their local normal vectors.

The resulting surfaces are a parallel approximation of the original ones, so in case of you need multiple parallel layers, it is recommended to generate each one from the original surfaces. Take special care in concave regions which may result wrong shapes.

The only required parameter is the extrusion distance that may be positive or negative.

The next figure shows and example of the resulting surfaces when this function is applied to a parabolic reflector.

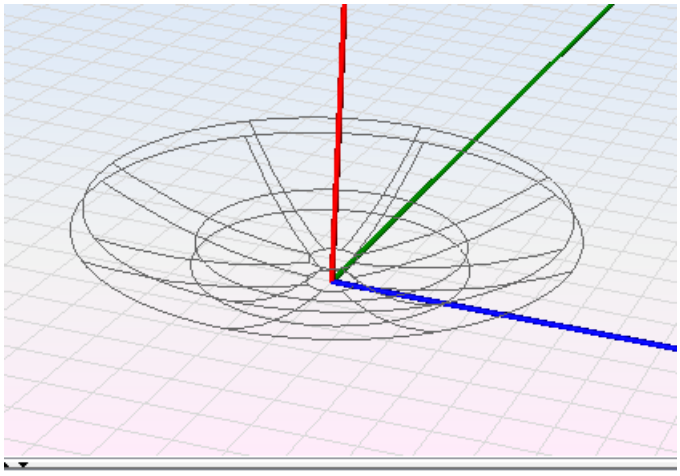


Figure 150: Generation of one parallel layer from a reflector.

Revolve

This function creates a surface by rotating a curve around a rotational axis. This function receives the following parameters the curve to rotate, two axis points and the initial and final angles of rotation. The line that contains the two axis points will be considered the rotation axis.

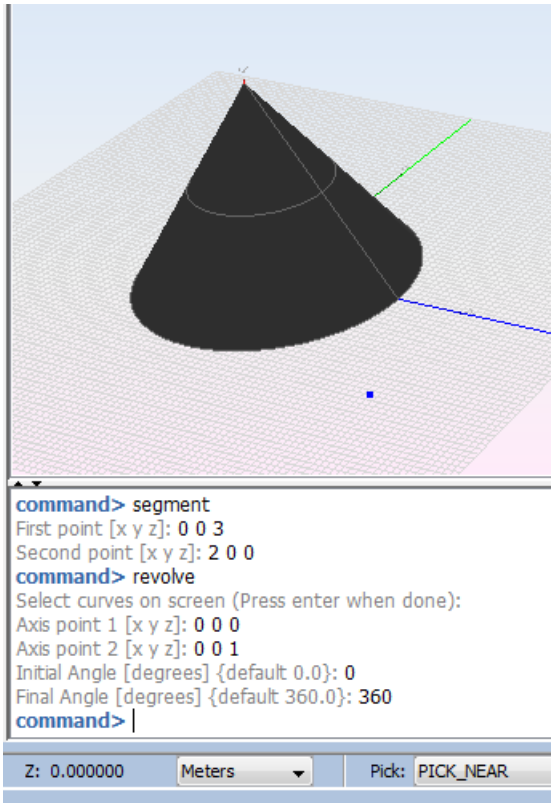


Figure 151: Revolve a segment to create a cone

Trim

This function allows the user to trim a surface with a set of selected curves. The function receives the surface and one or more (closed) curves.

Note that the final surface depends on the direction of the curve used for the trim operation, as shown below.

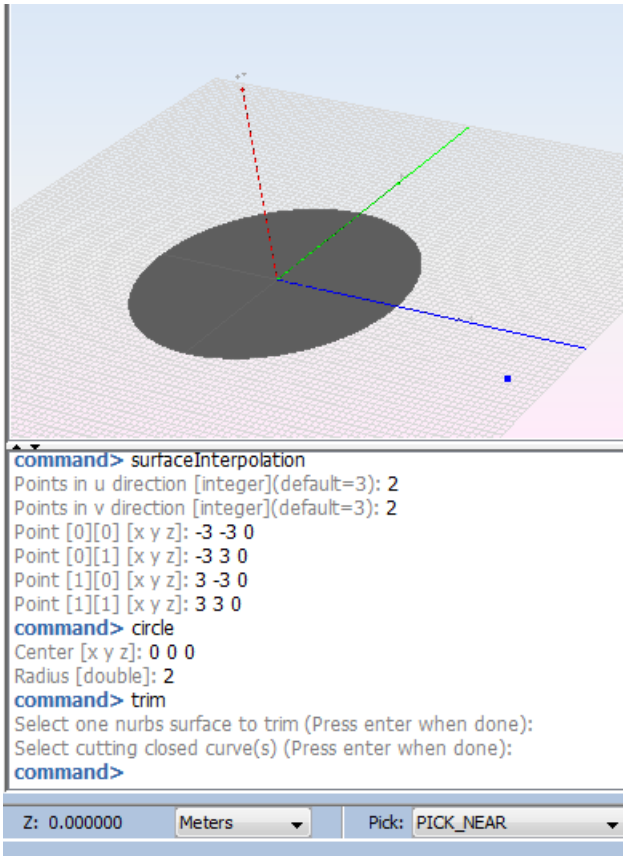


Figure 152: Squared surface trimmed by a circle

Untrim

This function does the inverse operation of the trim function. It receives a trimmed surface as a parameter and removes the cutting loops from it.

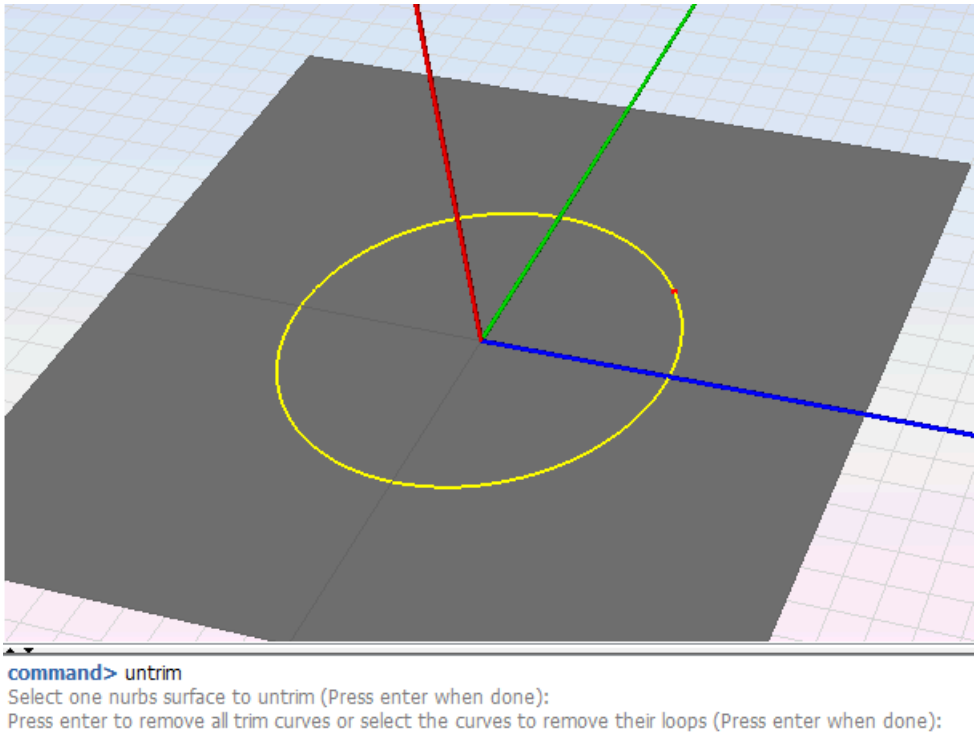


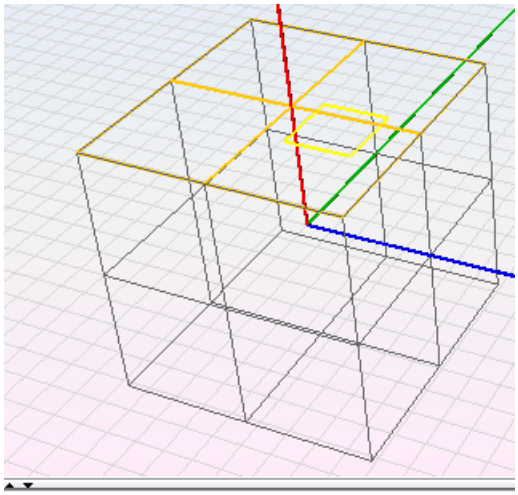
Figure 153: Untrim a rectangular surface that had been bounded by a circle

Extend Hole

This option allows the user to trim a surface with a loop and extrude the resulting cut off part of the surface.

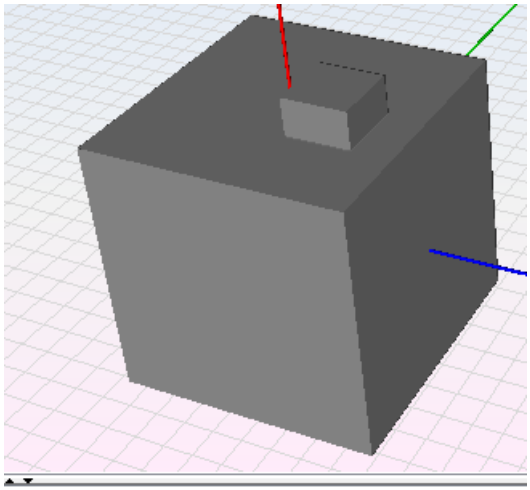
To use this operation,click on “Edit → Geometric Operations → Extend Hole” option. Then, the selection of the surface to be extruded is required. After that, select a curve or set of curves that form a closed loop. Finally, the length of the extruded pipe is required.

The figure below shows a box that has been created and exploded. Its upper surface is going to be extended with a hole defined by a rectangle.



```
command> box
First corner of base [x y z]: -0.5 -0.5 -0.5
Base size [width depth]: 1 1
Height [double]: 1
command> rectangle
First corner of rectangle [x y z]: 0.1 -0.2 0.5
Size [width depth]: 0.25 0.25
command> explode
command> extendHole
Select one surface to prolong (Press enter when done):
Select a closed set of curves (Press enter when done):
```

Figure 154: Definition of a rectangular hole in a surface



```
command> box
First corner of base [x y z]: -0.5 -0.5 -0.5
Base size [width depth]: 1 1
Height [double]: 1
command> rectangle
First corner of rectangle [x y z]: 0.1 -0.2 0.5
Size [width depth]: 0.25 0.25
command> explode
command> extendHole
Select one surface to prolong (Press enter when done):
Select a closed set of curves (Press enter when done):
Height [double]: 0.15
command>
```

Figure 155: Extended rectangular hole in a surface

Connect 2 Surfaces

This option allows the user to connect two separated surfaces by creating a new surface between them. To use this operation select one surface, confirm it and then select the second surface.

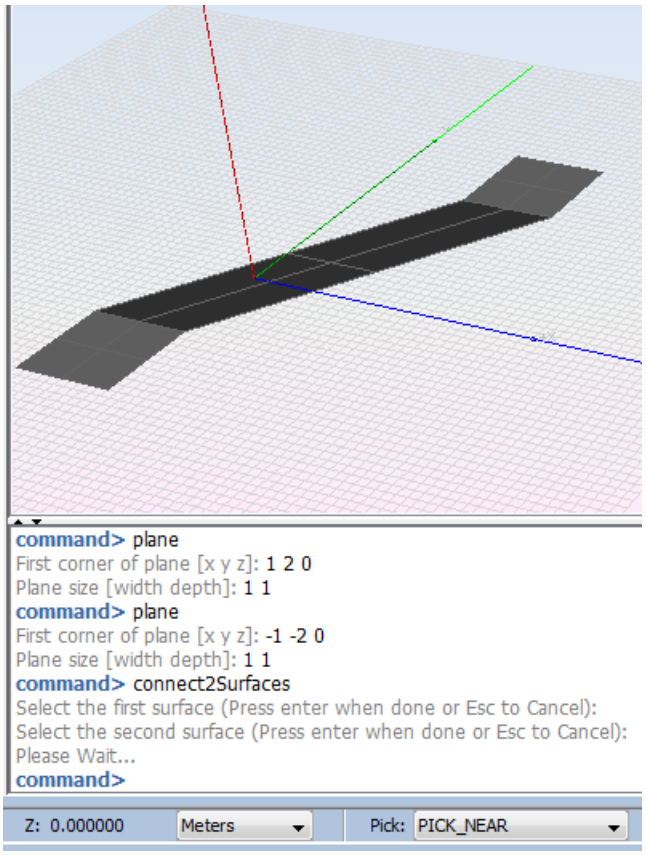


Figure 156: Connec2Surfaces function

Rebuild

This option allows rebuilding any given surface by sampling and interpolating the resulting grid of points. The user needs to specify the degrees of the new surface and number of samples (both in U- and V-dimensions) taken into account on the original surface.

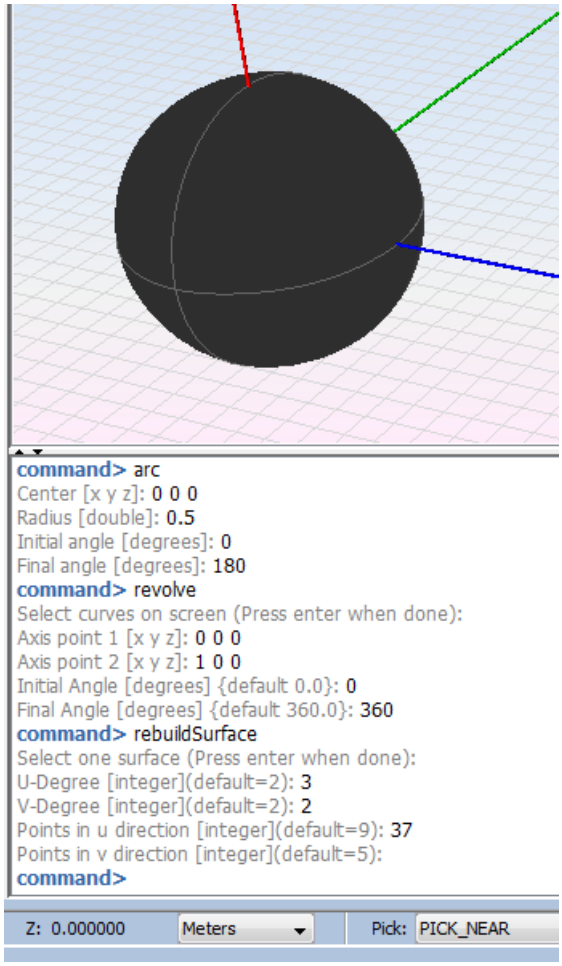


Figure 157: Rebuild a surface

Plane

This option allows the user to create a finite plane specifying its corner points.

Remark The second point assumes implicitly the same z coordinate as the first one.

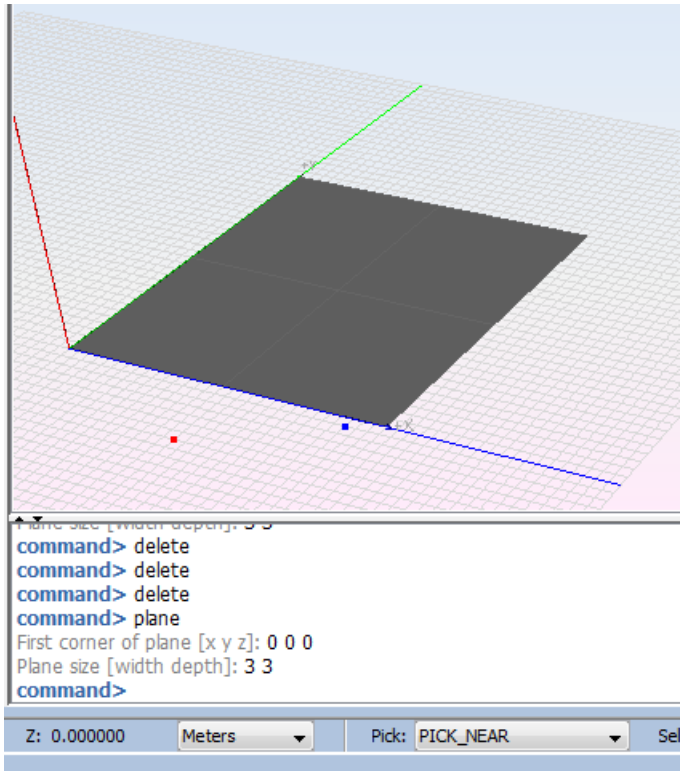


Figure 158: Creating a plane

Disk

This option allows the user to create a disk by specifying its center point and the radius.

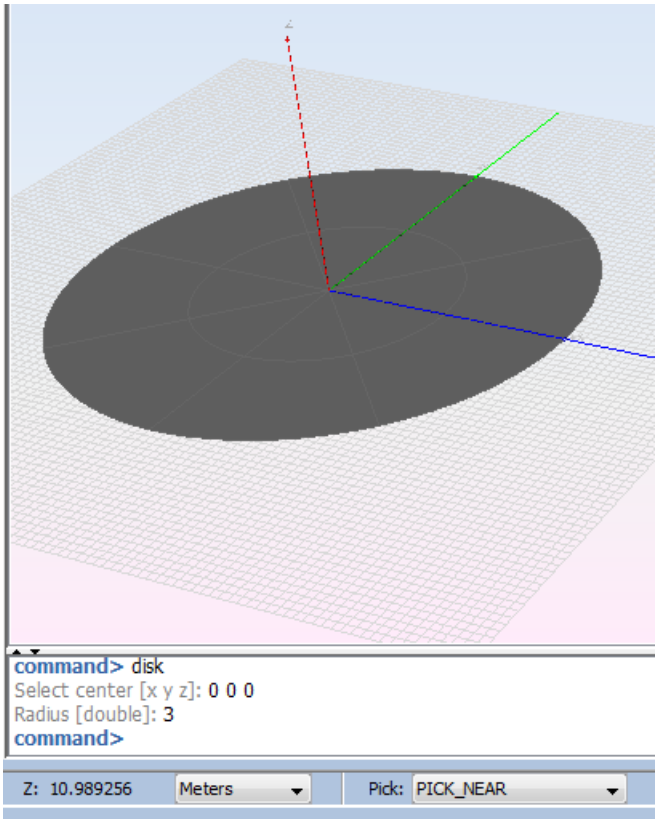


Figure 159: Creating a disk

Hexagon

The hexagon primitive can be used to create a solid hexagon surface.

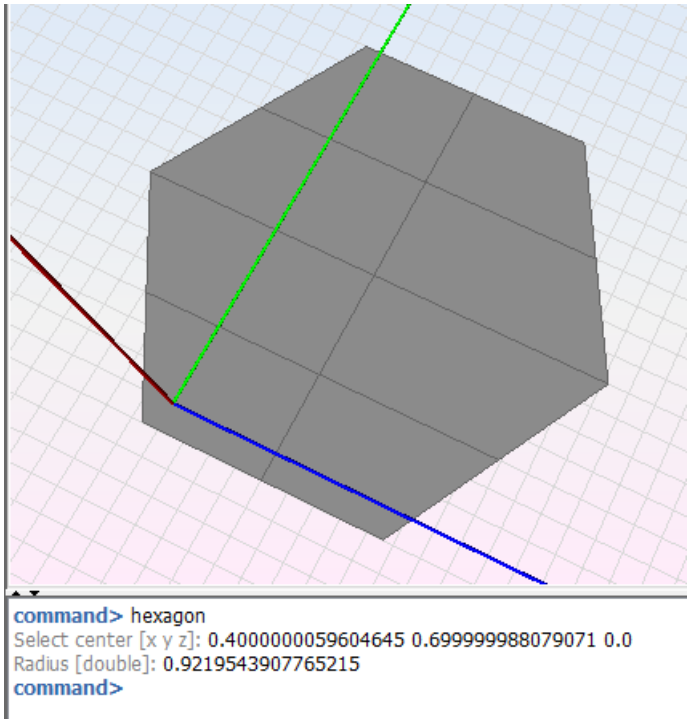


Figure 160: Hexagon created

Ellipse

The ellipse option allows the user to create a solid ellipse surface, given the radius along the X axis, the radius along the Y axis and the center point.

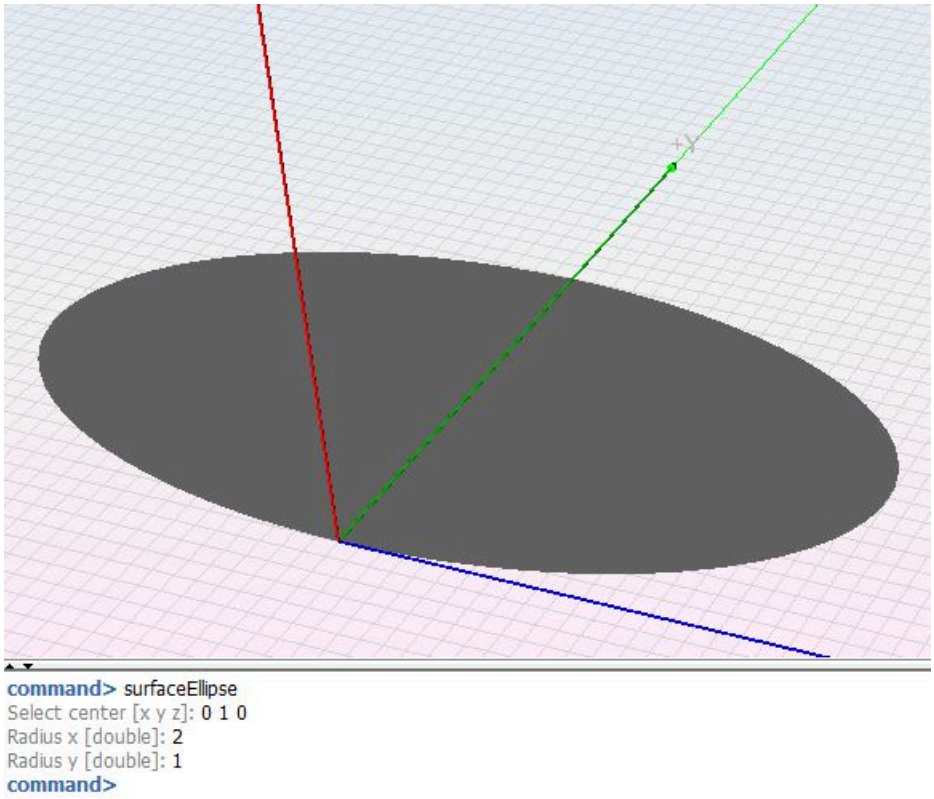


Figure 161: Creating an ellipse

Rings

This menu gives the user the option to create several types of ring-shaped geometries.

Ring

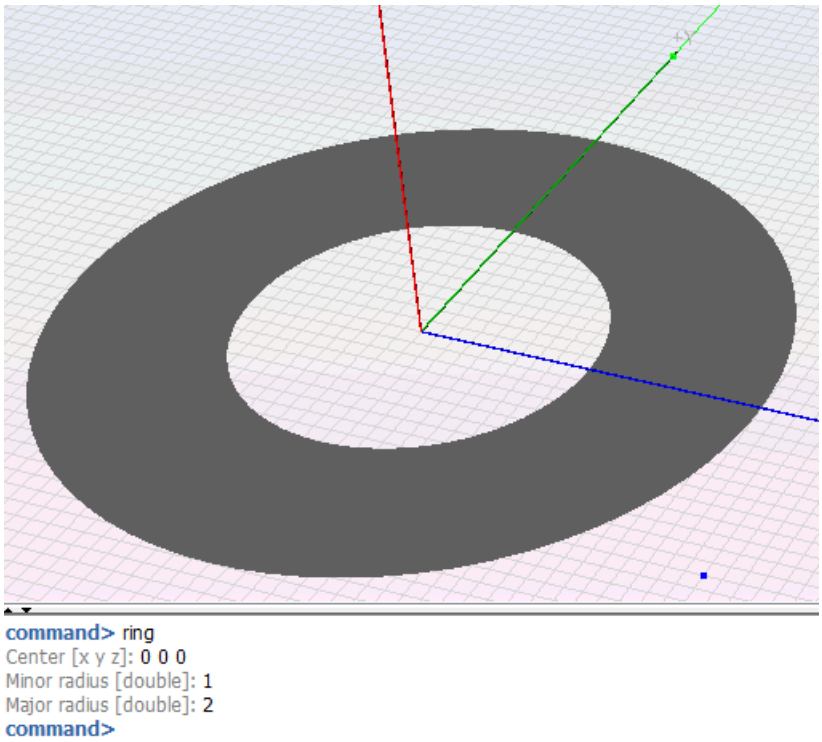


Figure 162: Creating a ring

Open Ring

This option allows the user to create an open ring surface. The parameters asked for the creation of this type of surface are the following:

- The center point of the ring.
- The minor (inner) radius and major (outer) radius.
- The angle of the opening, in degrees.
- The direction of the opening, in degrees, that determines in which direction the opening is facing. A angle of zero degrees means that the opening will be facing in the direction of the positive Y axis, while larger angles of this parameters are interpreted as counter-clockwise rotations.

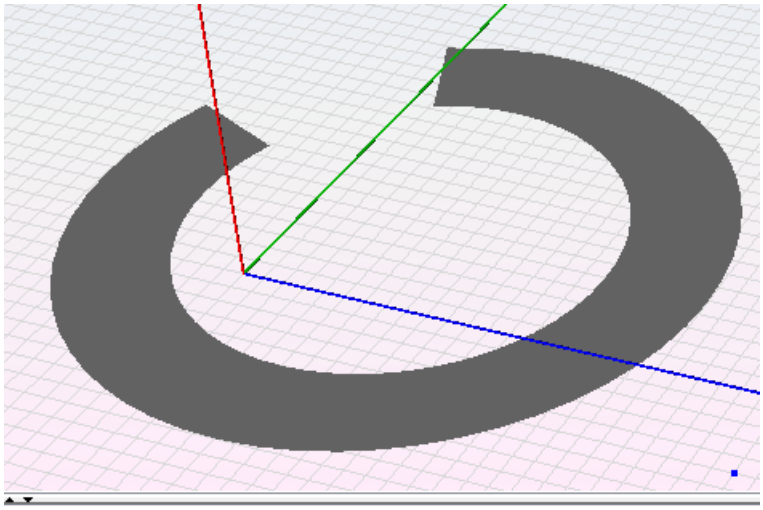


Figure 163: Creating an open ring

Split Ring

This option allows the user to create a split ring, that is, a ring surface that is open at two opposite points. The parameters that need to be supplied by the user are:

- The center point of the ring.
- The minor (inner) and major (outer) radius, as in the previous geometries.
- The opening angle, in degrees, that determines how large the openings are. Both openings are created the same size.
- The opening direction, in degrees, that determines how the openings are oriented. An opening direction of 0 degrees means that the openings will be oriented in the direction of the Y axis. Larger angles are interpreted as a counter-clockwise rotation.

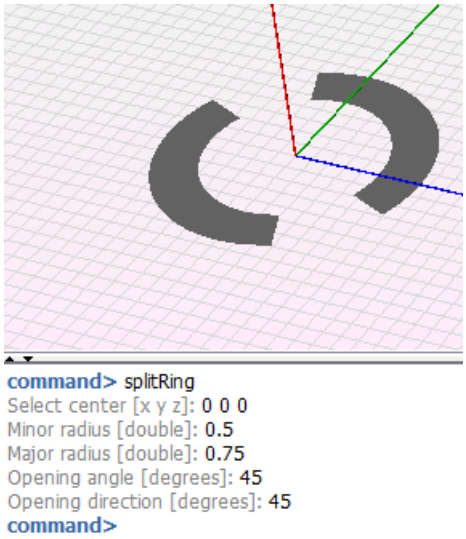


Figure 164: Creating a split ring

Crosses

This menu lets the user create different kind of crosses.

Cross

This function allows the user to create a cross-like surface. The parameters that this function receives are the center point and two distances (Size A and Size B). Size A defines the side of the central square forming the cross, while Size B defines one of the sides of each of the four rectangles around the square.

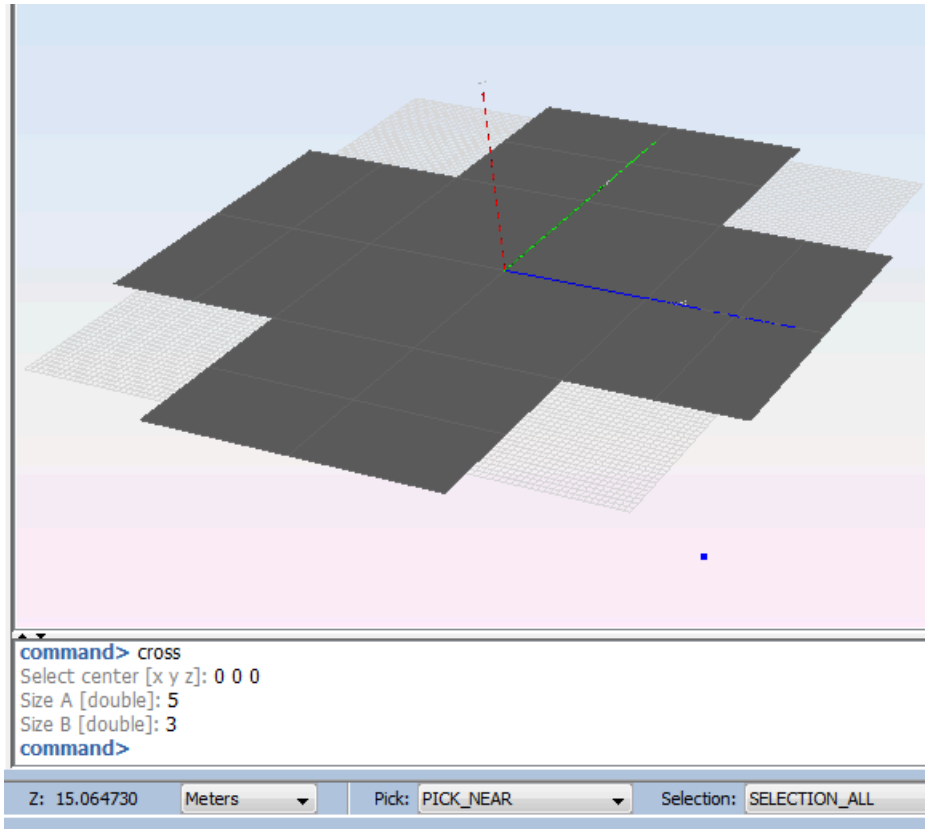


Figure 165: Creating a cross

Spiral Cross

This option allows the user to create a spiral cross surface. The parameters needed for the creation will be asked to the user, namely, the center point of the cross, the thickness of the segments and the lengths of each segment.

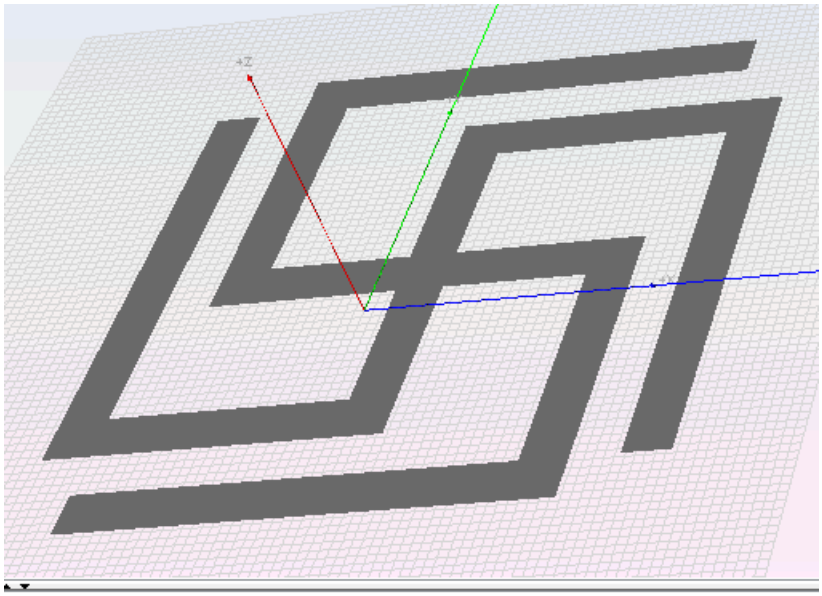


Figure 166: Creating a spiral cross

T Cross

This option allows the user to create a cross with T-shaped segments. The parameters needed for the creation of this type of surface are the following:

- The center point of the cross.
- The thickness of the segments.
- The length of the first segment, that is, the height of each T-shaped segment.
- The length of the second segment, that is, the length of the top of the T-shaped segment.

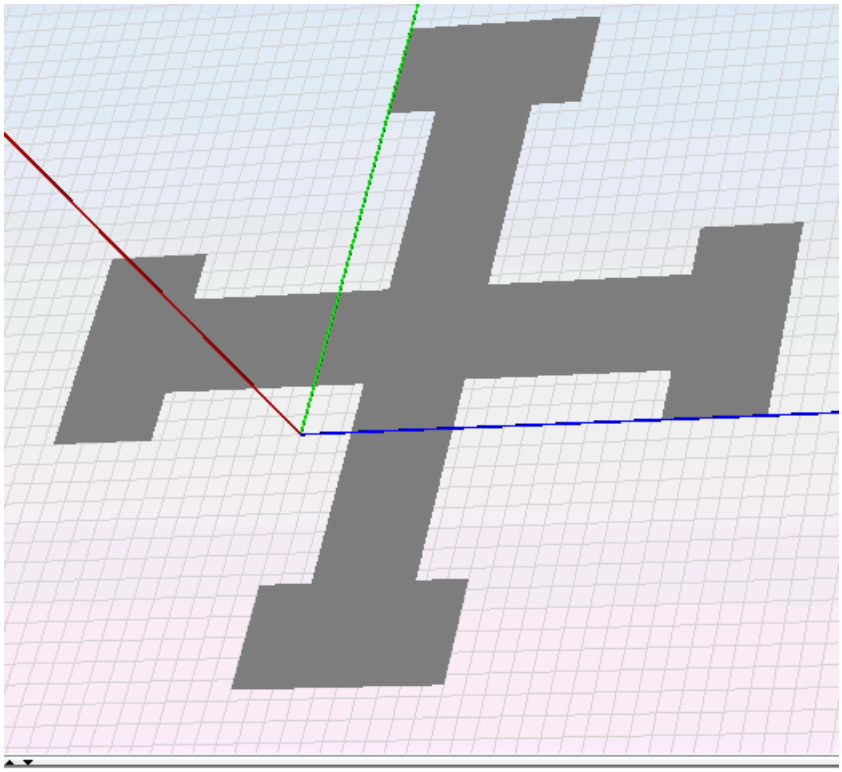


Figure 167: Creating a T Cross

Trifilar

This option allows the user to create a new trifilar surface, that is, a cross with three evenly spaced segments. The user needs to supply several parameters, namely, the central point of the trifilar, the thickness of the segments and the length of the segments.

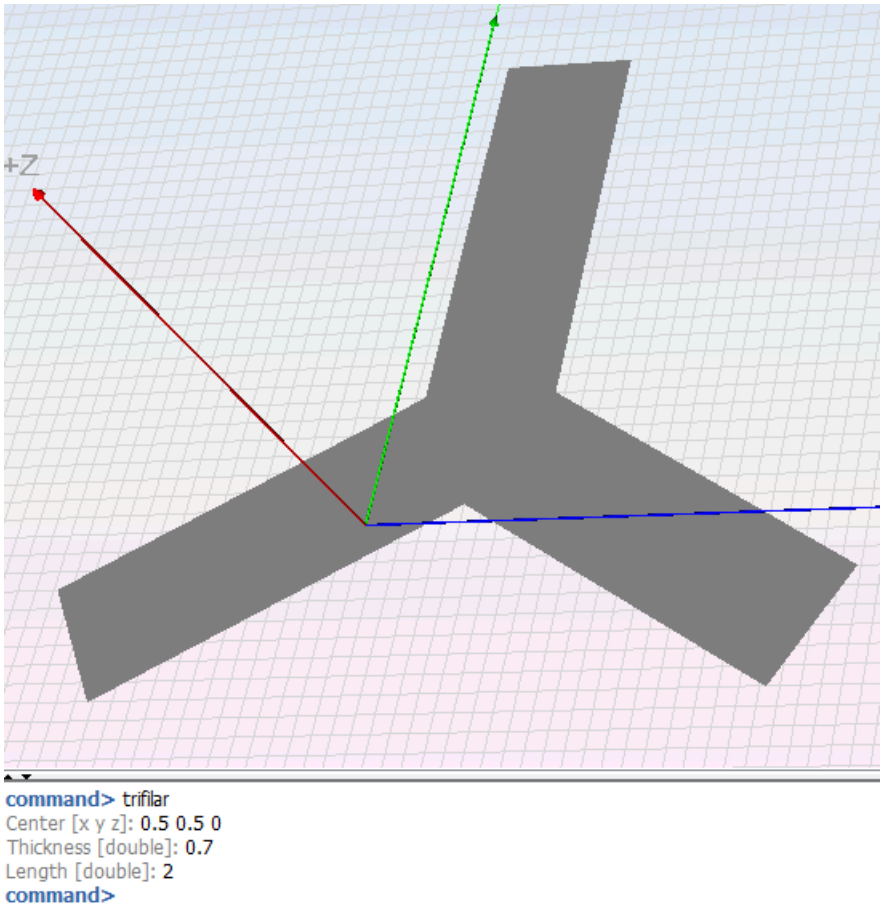


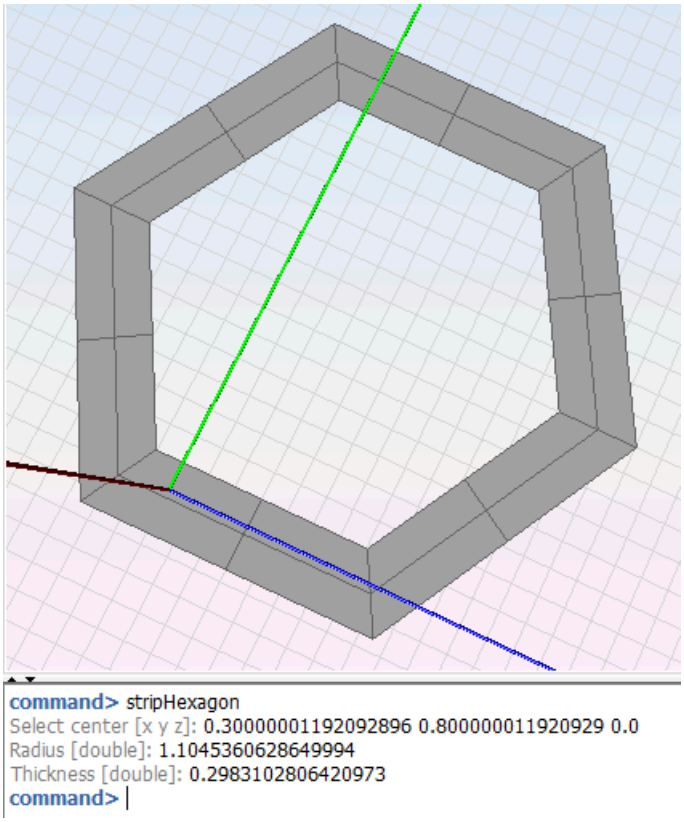
Figure 168: Creating a trifilar

Strips

This menu lets the user create different kinds of strip primitives such as spirals or interdigital lines.

Strip Hexagon

The stripHexagon primitive creates an hexagon that has a given outer thickness. The inner area of the surface has a hole.



Strip Trifilar

This option allows the user to create a hollow trifilar surface. The parameters needed for the creation of a strip trifilar are the central point of the trifilar, the thickness of the (equivalent, not hollow) trifilar, the length of the segments and the thickness of the hole of the trifilar.

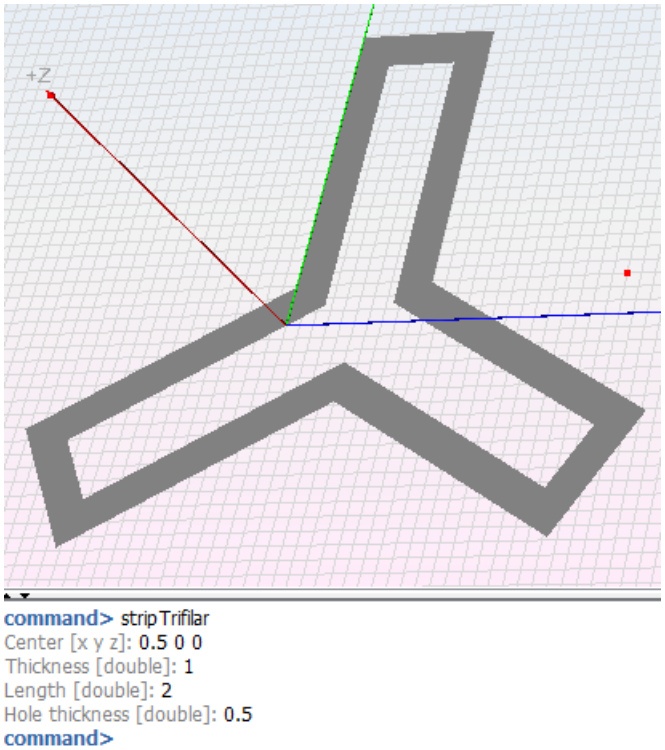
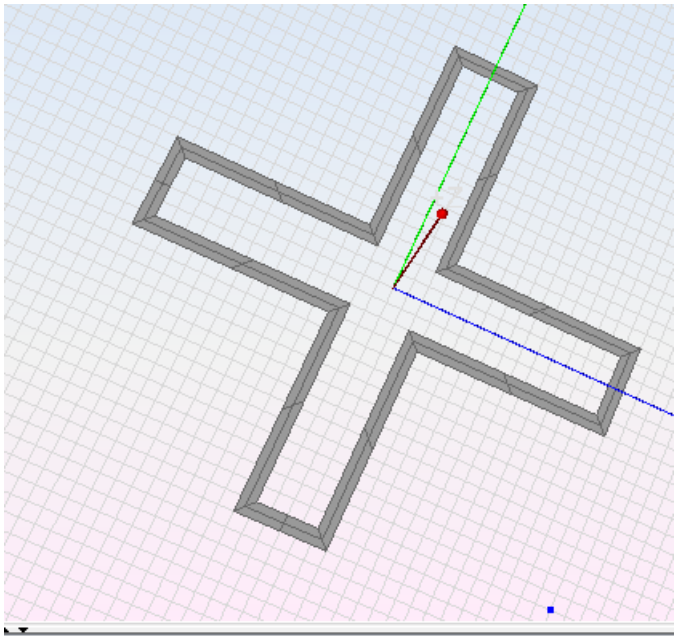


Figure 169: Creating a strip trifilar

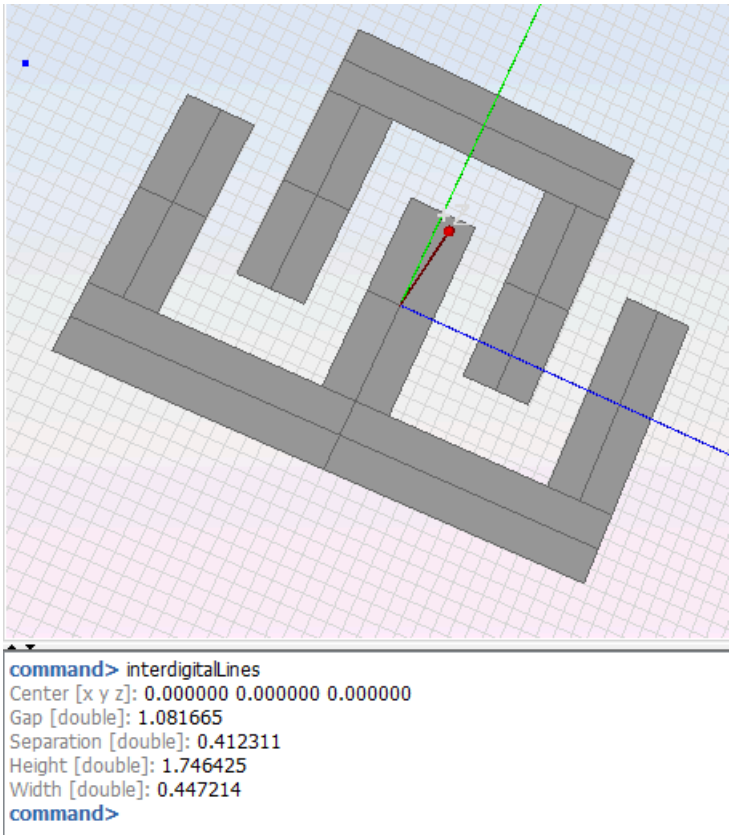
Strip Cross

The stripCross primitive can be used to generate a cross primitive that has a defined outer thickness and an inner hole.



Interdigital Lines

The interdigitalLines primitive lets the user create a stripline interdigital filter. The user can define different parameters for the geometry. Most of the settings can be selected either by writing the correct value in the command prompt or by pick-pointing the screen. The user can preview the geometry as it is being defined.



Capacitance

This option allows the user to create a capacitance surface. The needed parameters are the center point of the capacitance (equidistant to both plates), the separation between plates, the size (width and height) of the plates and the length of the capacitance.

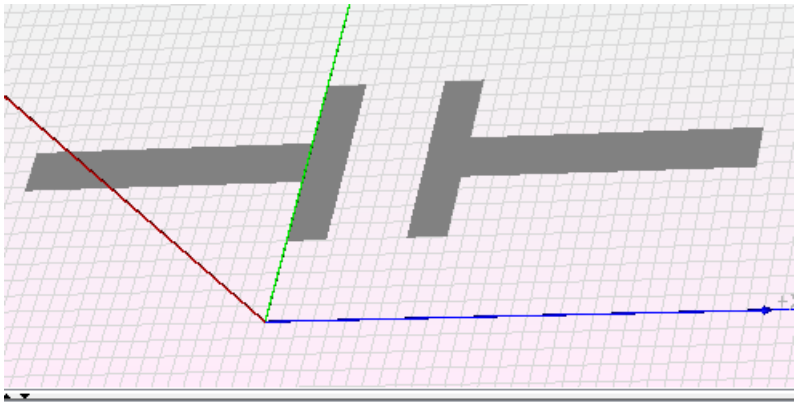


Figure 170: Creating a capacitance

Spiral

The primitive surfaceSpiral generates a coplanar set of surfaces that turns from the center to the end point, specified by the width, the radius and the number of turns. Some configurations of these parameters may generate invalid spirals, so the command will return an error with some suggestions for the parameters review.

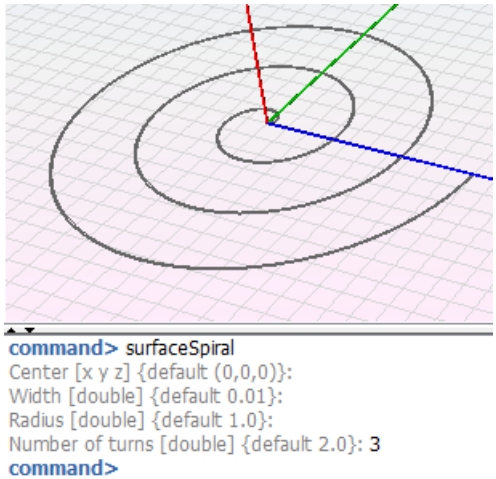


Figure 171: Surface Spiral of 3 turns and counter-clockwise direction

By default, the generated spiral is generated in counter-clockwise direction. To generate spirals in clockwise direction, the parameter "-rightTurned" must be included.

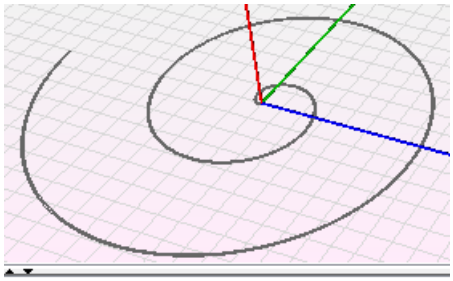


Figure 172: Surface Spiral of 2 turns and clockwise direction

Quadrics

This menu lets the user create different kinds of hypersurfaces.

Paraboloid

This option allows the user to draw a paraboloid. This function receives the following parameters center point (vertex), focus distance, radius and hole radius.

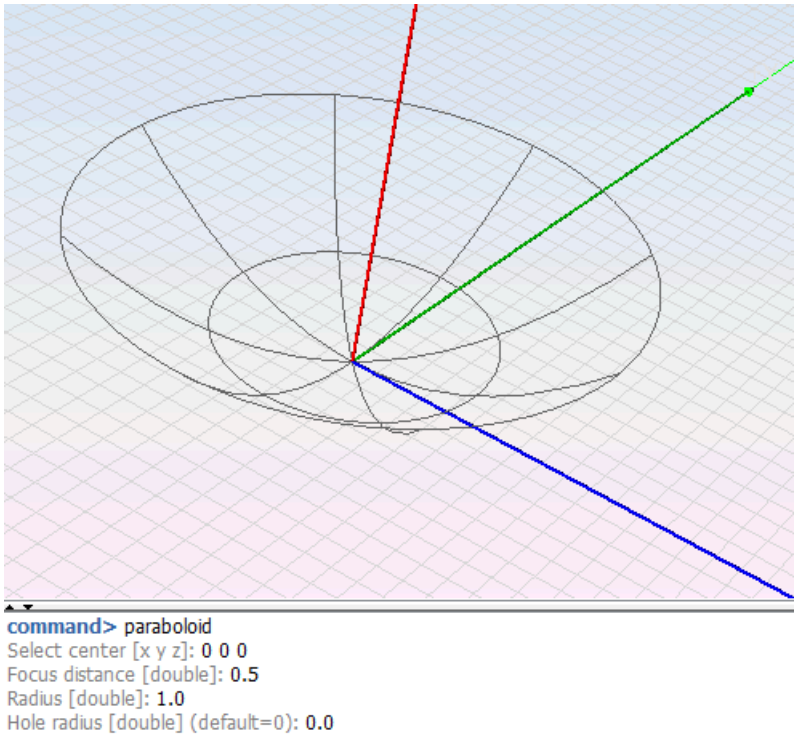


Figure 173: Drawing paraboloid.

Hyperboloid

This option allows the user to create a hyperboloid. The parameters needed for this function are:

- Center point: Describes the vertex of the hyperboloid.
- Focus distance: It is defined as the distance between the vertex and focus point.
- Radius: It is the distance from the vertex to the corner point in XY plane.
- Height: It is the distance from the vertex to the corner point in Z.
- Hole radius: It is the size of the hole in the center of hyperboloid.

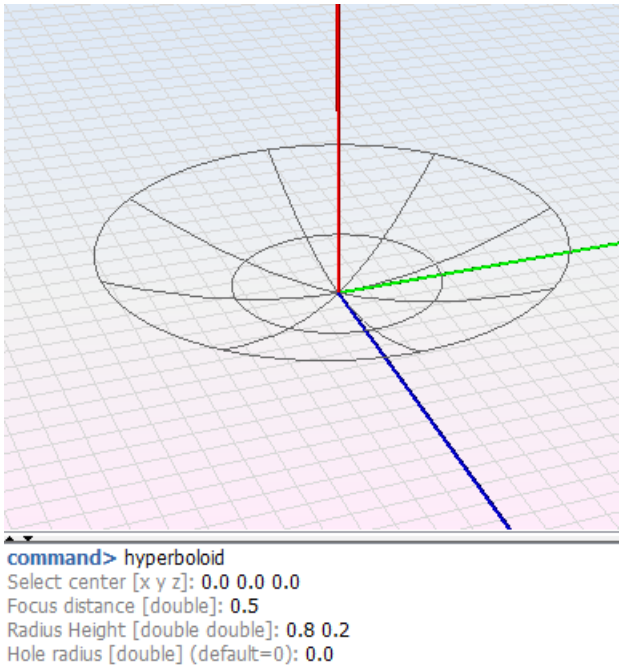


Figure 174: Drawing a hyperboloid.

Holes

This menu lets the user create different kinds of holes.

Circular Hole

This option allows the user to quickly create a plane with a circular hole in it. The parameters for this primitive are the following the corner point of the plane, the size of the plane (width and depth) and the radius of the circular hole in the plane.

Figure 175: Creating a circular hole

Rectangular Hole

The rectangularHole primitive allows the user to quickly create a rectangular surface with a hole defined on its center. Usage:

- First corner of plane: click the 3D View to pick-point a corner of the rectangle or provide a valid 3D value.
- Size of the plane: the size of the plane can be previewed by moving the cursor through the 3D View and clicking for setting the size. Alternatively, the width and height can be provided.
- Size of the hole: the hole will be centered in the outer rectangle. Move the cursor through the 3D View to preview the size and click to set, or provide the numeric width and height in the console.

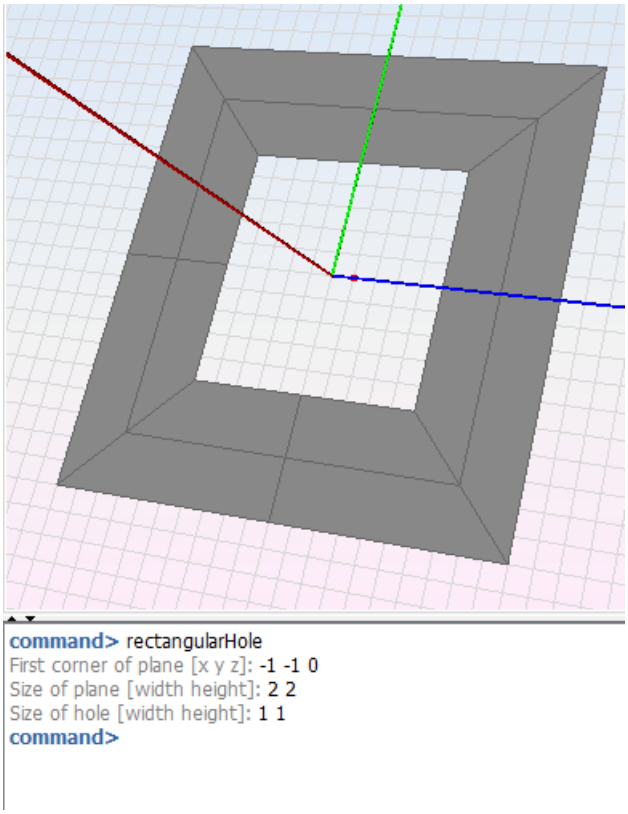


Figure 176: Creating a rectangular hole

Cross Hole

This option allows the user to quickly create a plane with a cross-shaped hole in it. The parameters needed to create this primitive are the corner point of the plane, the size (width and depth) of the plane, the thickness of the cross-shaped hole and the length of the segments of the cross.

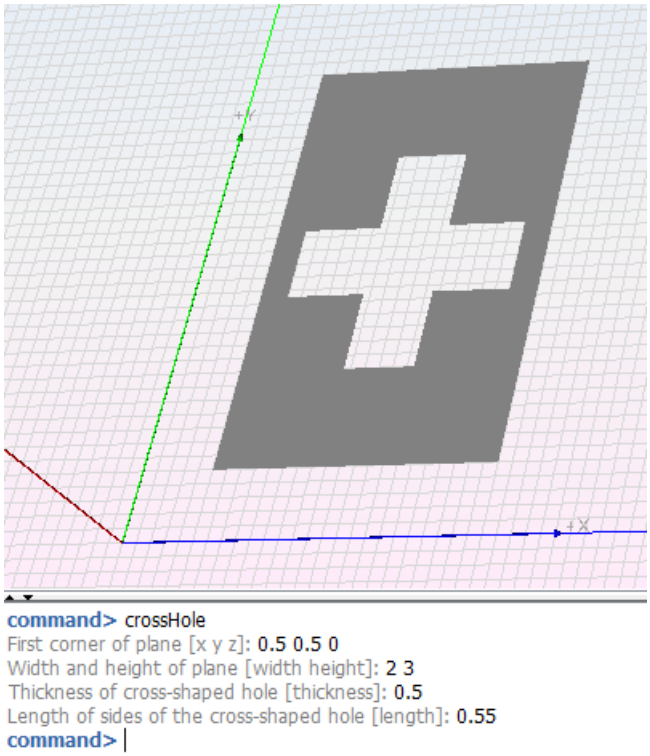


Figure 177: Creating a cross hole

Dipoles

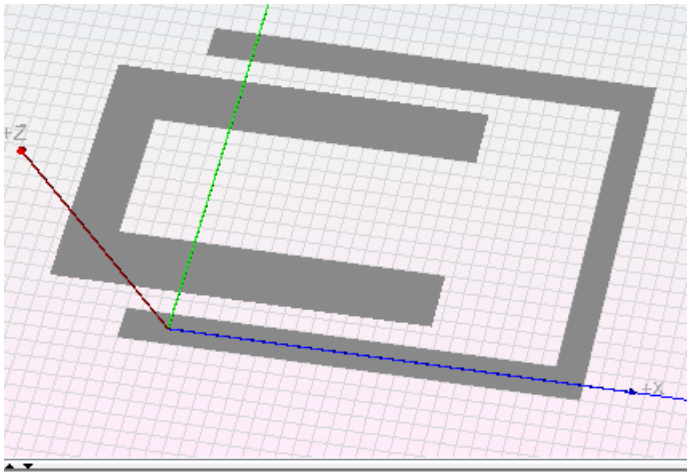
This menu allows the user to create several types of dipoles.

Bent Dipoles

This option allows the user to create two bent dipoles in the geometry. The following parameters will be asked to the user when creating a pair of bent dipoles:

- The center point. This point will be equidistant to the central segment of both dipoles.
- The separation between both dipoles.
- For each of the dipoles the height, thickness and length of the dipole.

These parameters must be chosen so the resulting geometry is valid (i.e dipoles do not overlap). If this is not the case, an error will be issued and the geometry will not be created.

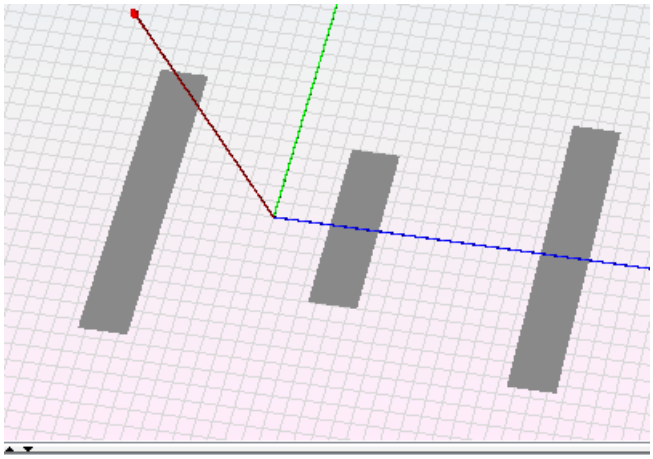


command> bentDipoles
Select center [x y z]: 1 1 0
Separation [double]: 3
Height of first dipole [double]: 2.2
Width of first dipole [double]: 0.2
Length of first dipole [double]: 3
Height of second dipole [double]: 1.5
Width of second dipole [double]: 0.35
Length of second dipole [double]: 2.5
command>

Figure 178: Creating bent dipoles

Three Dipoles

This option allows the user to create three dipoles in the geometry. The following parameters will be asked center point of the set of three dipoles, dipole size of the central dipole (width and height), separation between dipoles and height of the lateral dipoles.



command> threeDipoles
Central point [x y z]: 0.5 0 0
Dipole size [width height]: 0.3 1
Dipole separation [gap]: 1
Lateral dipoles height [height]: 1.7
command>

Figure 179: Creating three dipoles

Three Dipoles Union

This option allows the user to create three joined dipoles in the geometry. When creating a primitive of this type, the following parameters will be asked the central point of the set of dipoles, the size of the central dipole (width and height), the separation between dipoles, the height of the lateral dipoles and the width of the union of the three dipoles.

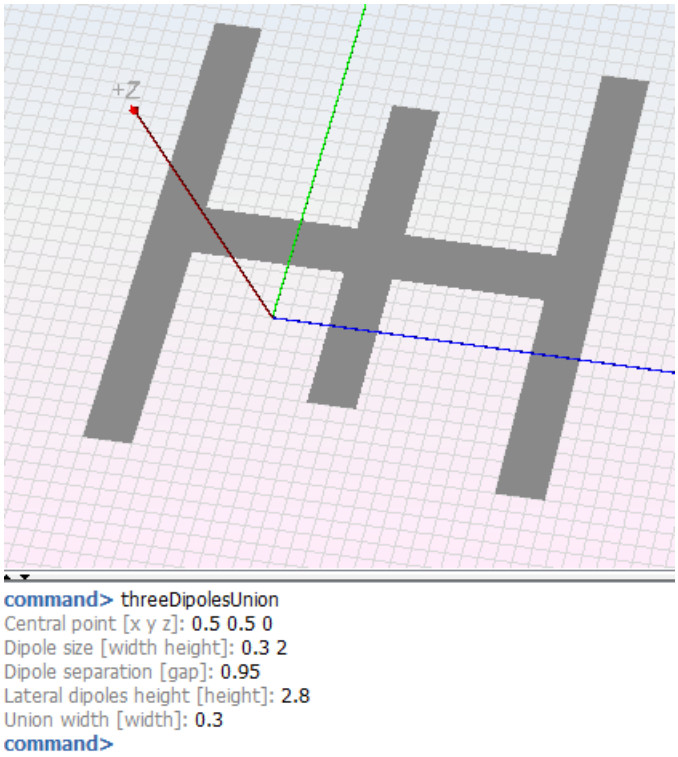


Figure 180: Create three joined dipoles

From Curves

This submenu contains several ways for creating surfaces by using more geometrical objects.

From Edges (Coons)

This function constructs a new surface from the closed set of edge curves that bounds the surface. This command only needs to select the two, three or four edges that forms the boundary loop.

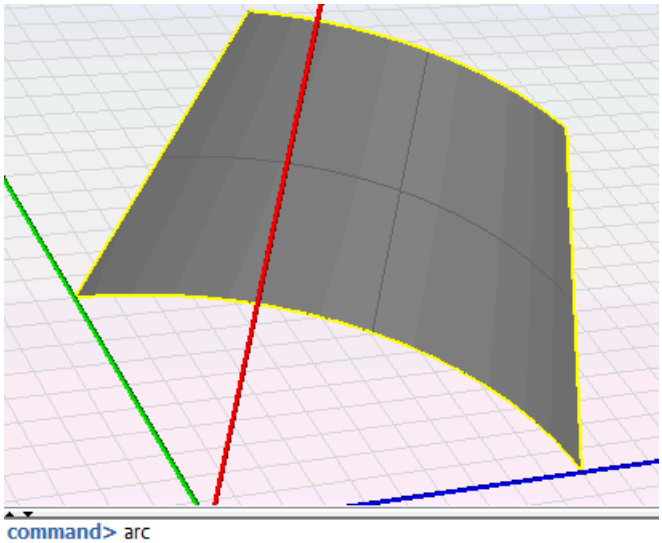


Figure 181: Coons surface

Planar Surface

This option allows creating planar surfaces from multiple coplanar lines. The operation searches for all the closed loops created by the input curves, and surfaces are generated when at least one loop is detected.

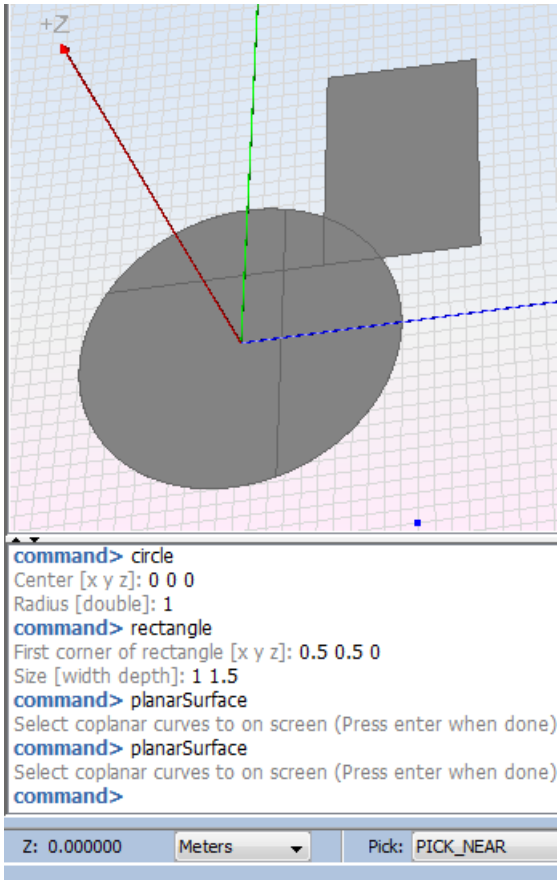


Figure 182: Creating a planar surface from a circle and a rectangle

Skinned

This function creates a surface with a set of curves that is required in order. Skinning is a process of blending the section curves together to form a surface. The blend direction is the v direction, sometimes called the longitudinal direction.

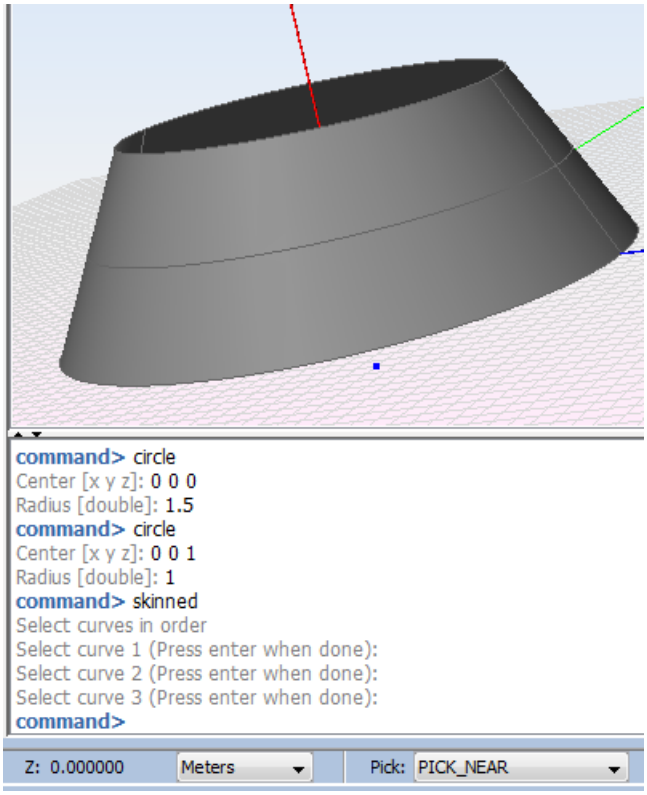


Figure 183: Joining the curves as a surface

Sweep

The command sweep generates a surface by sweeping a given cross-section (profile) curve along the rail curve. The resulting surface is always placed on the rail curve.

Note that the directions of the rail and cross-sections curves may vary the resulting surfaces.

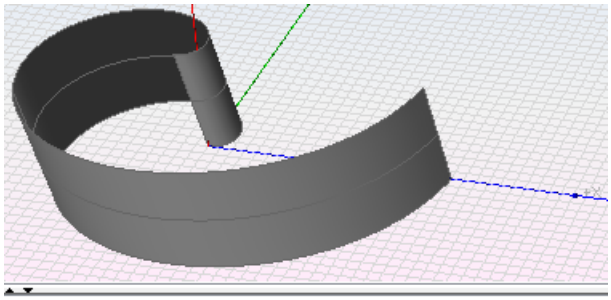


Figure 184: Spiral cross-section swept along a segment rail

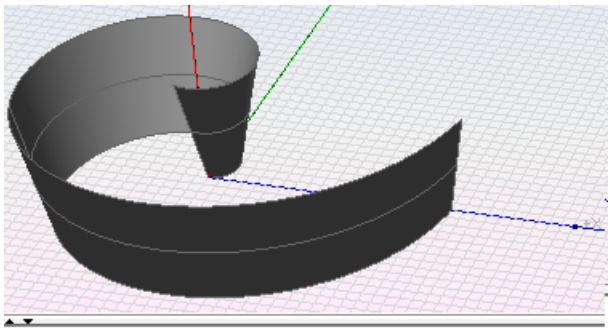


Figure 185: Segment cross-section swept along a spiral rail

Pipe

The primitive pipe generates a tube that surrounds to the given curve with a specific radius.

If the reference curve describes strong curvatures, the resulting pipe may be complex. Note that the generated pipe has no caps.

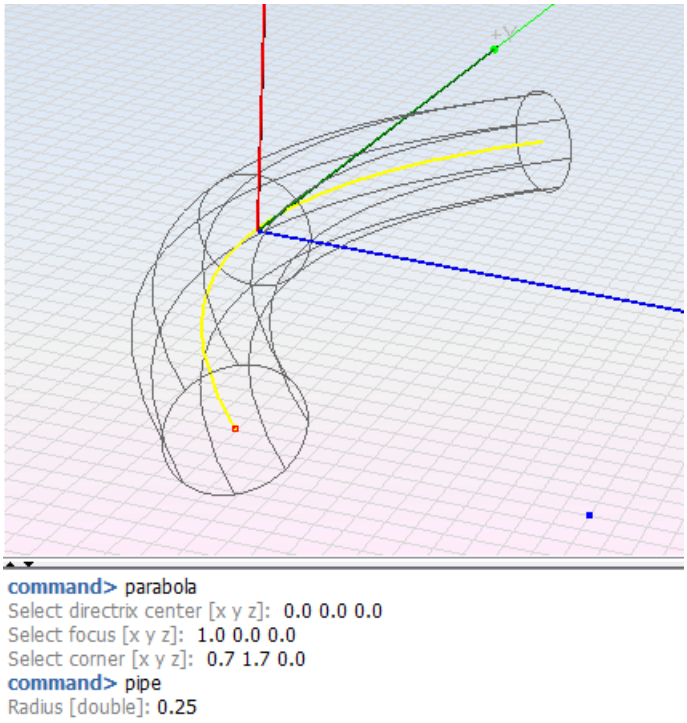


Figure 186: Example of pipe that surrounds a parabola.

Split

This submenu contains several options for splitting surfaces in different ways.

Surfaces

Select this option when multiple surfaces must be mutually split. Every surfaces will be divided in so many sections as required by the intersecting surfaces. The command may process both surfaces and objects, and the result is returned in the same format, i.e., after splitting a surface, the resulting sections is returned in multiple surfaces; however, an object is returned as the same object but with the resulting surfaces of the operation (explode it if necessary).

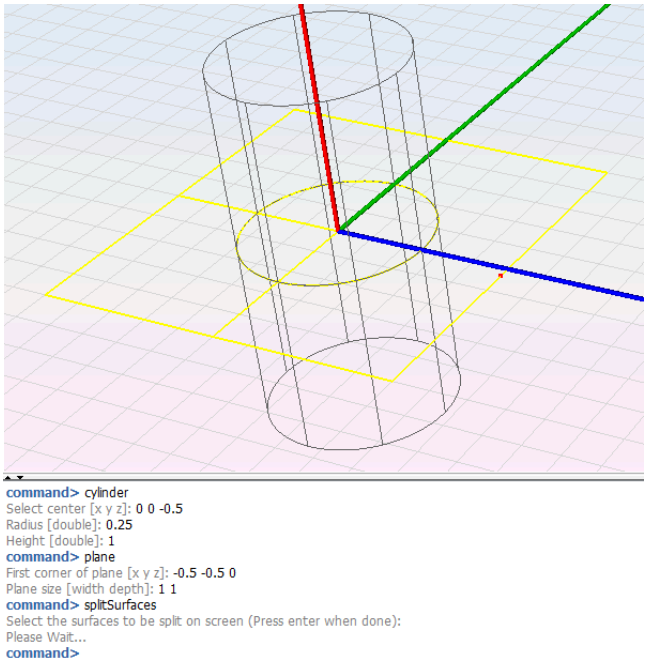


Figure 187: Mutual split of a cylinder with a plane

By Cutting Objects

Select this option when only some surfaces must be split by different objects (the command splitSurfaces with the parameter –c is used). All the surfaces to be split must be selected in the first step, and after confirming the selection, the cutting objects are selected (curves, surfaces and objects are considered).

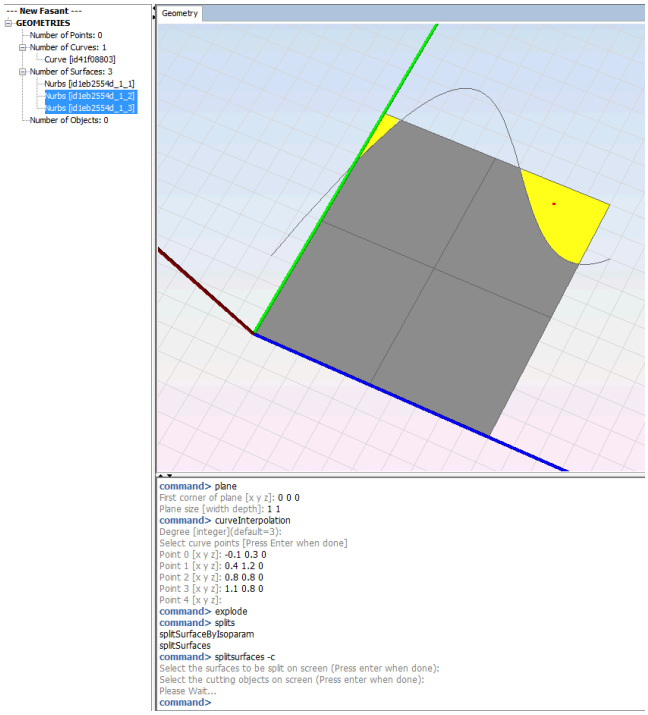


Figure 188: Exploded plane split by an interpolated curve

By Isoparam

This option is useful when a surface need to by split in a given isoparametric curve. This operation may be used to separate surfaces with extreme curvatures in different parts. Tip u or v to change between the two parametric dimensions of the surface. Then, the point is specified as a parametric coordinate (between 0 and 1), and the pick mouse functions may be useful to split the surface close to any desired point.

One or multiple parametric divisions may be inserted in the command line, all of them corresponding to the selected U or V dimension.

To select the divisions points with the pick mouse functions, two different modes may be used:

- Use the left mouse button to set an unique division coordinate. The previous values are removed and just the last one is considered.
- Use the right mouse button to add the last coordinate to the already existing values.

Confirm the selected divisions by pressing **Enter** key in the command line.

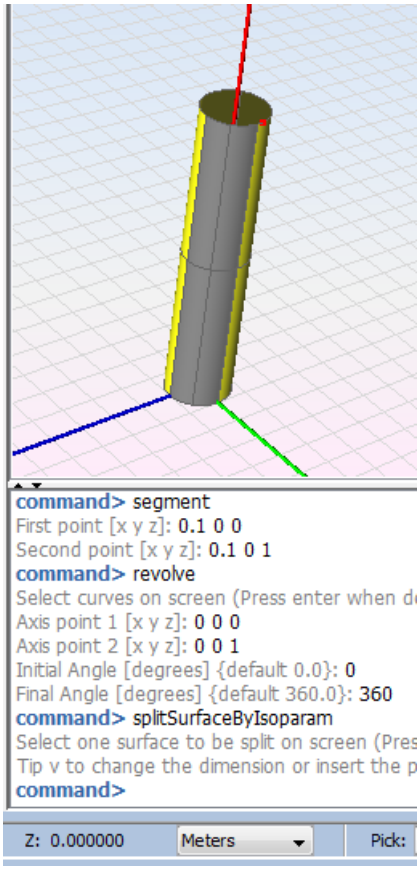


Figure 189: Split a cylindrical surface by isoparam line

By Curves Projection

This menu contains two options for splitting surfaces by using the projection of any set of curves.

To Closest Point

Every cutting curve is projected onto the selected surfaces. When multiple target surfaces are selected the curves may be projected onto all surfaces, whenever the projections are perpendicular to the normal vectors.

This projection may be slow because the cutting curves are evaluated onto all surfaces.

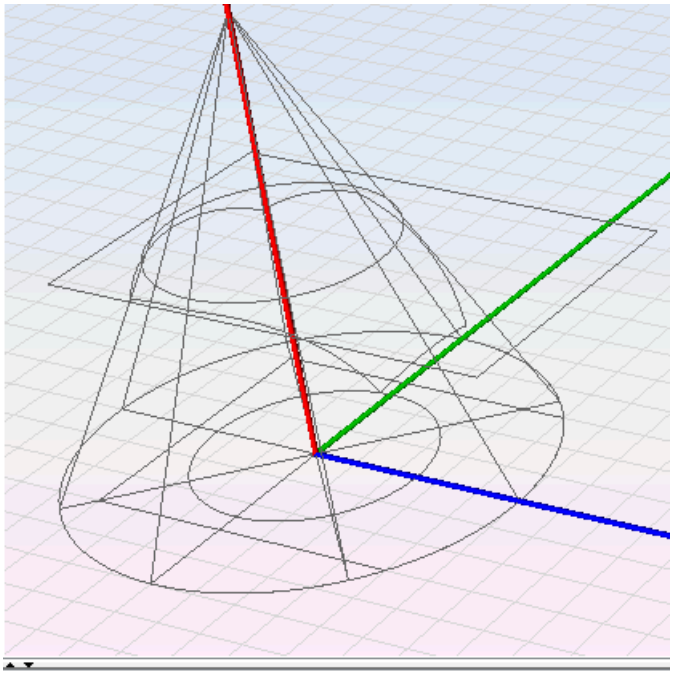


Figure 190: Split a cone with the projection of a rectangle to the closest point

By Direction

Every cutting curve is projected only onto the selected surfaces that are in the specified direction from the curve.

This projection is faster than the closest one due to the direction check.

After selecting the cutting curves and the target surfaces, the direction vector is required to project.

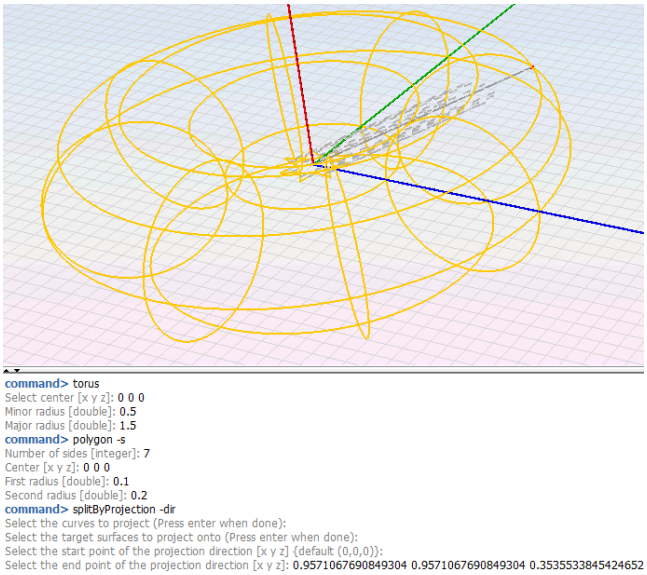


Figure 191: Split a torus with the projection of a star polygon by direction (1/2)

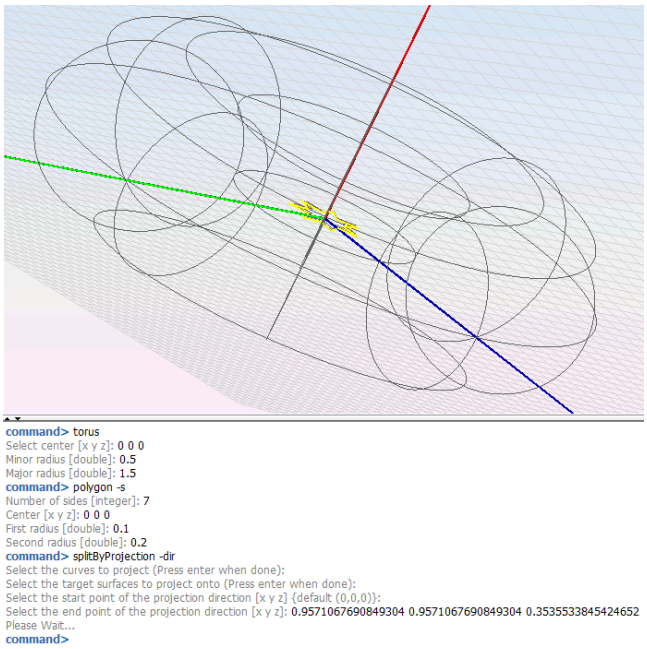


Figure 192: Split a torus with the projection of a star polygon by direction (2/2)

Solid

This menu contains the following options:

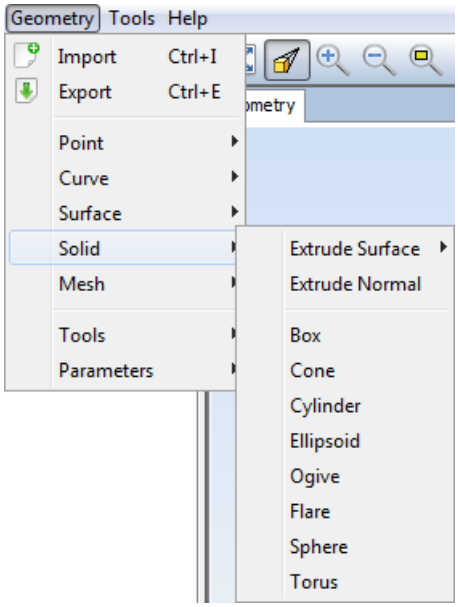


Figure 193: Solid submenu

Extrude Surface

This option allows the user to create a solid volume from a surface. There are two different forms to do this:

- **Straight**: This option creates a solid object by sweeping a surface along a direction vector. This function receives two points as parameters. The vector that connects these two points will be the direction vector considered for the extrusion.

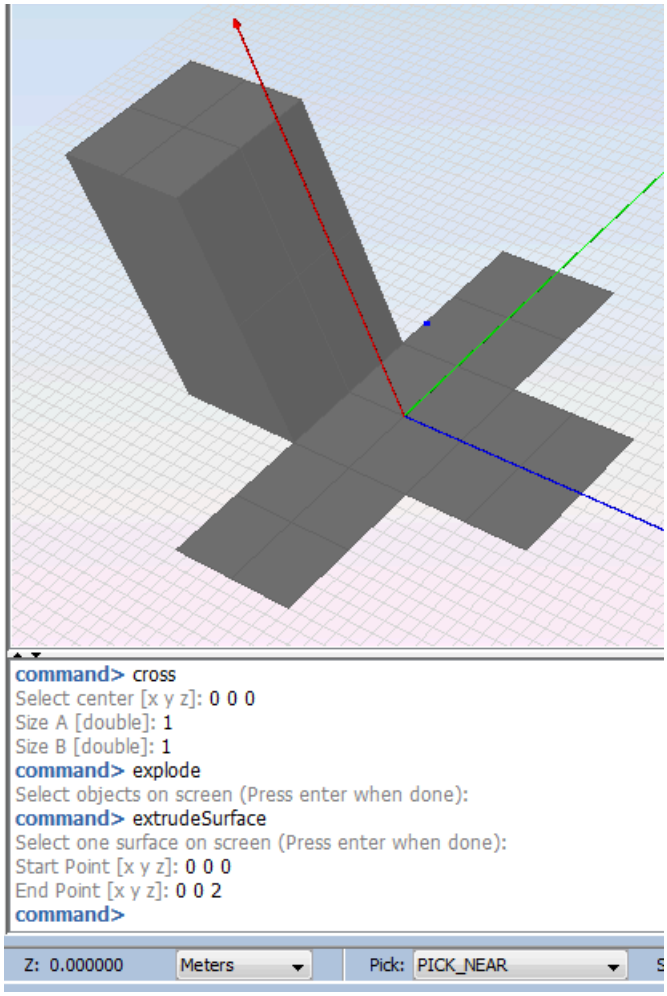


Figure 194: Result of extruding a surface

- **To point**: This option allows the user to create a solid object by connecting all points of the surface to a single point. The parameters needed for this operation are the surface to extrude and the point.

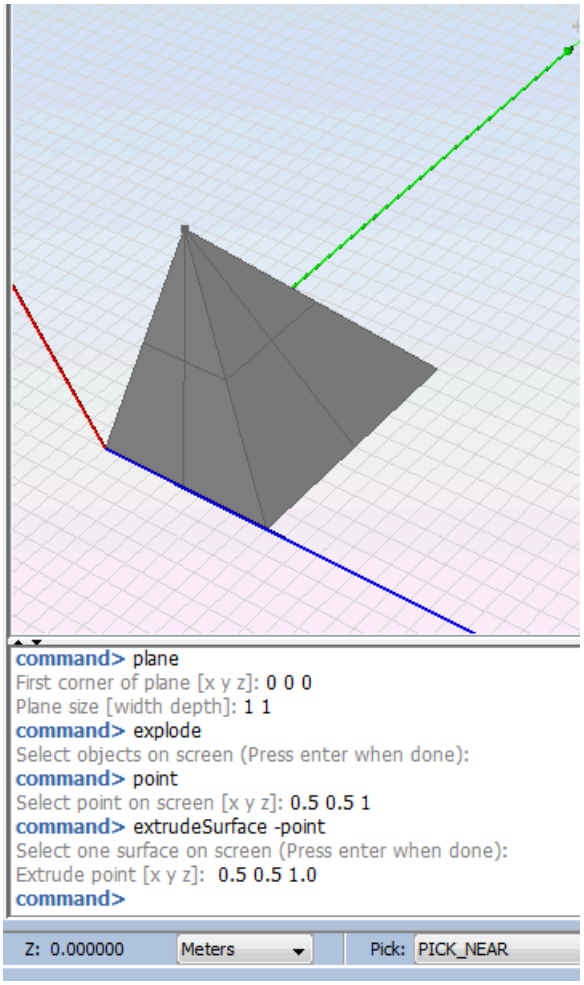


Figure 195: Result of extruding a surface to a point

Extrude Normal

This function returns an object with the surfaces resulting of extruding the input surfaces according to their local normal vectors plus the side surfaces to obtain a volume together the original surfaces.

The next figure shows and example of the resulting surfaces when this function is applied to four plane with hole surfaces.

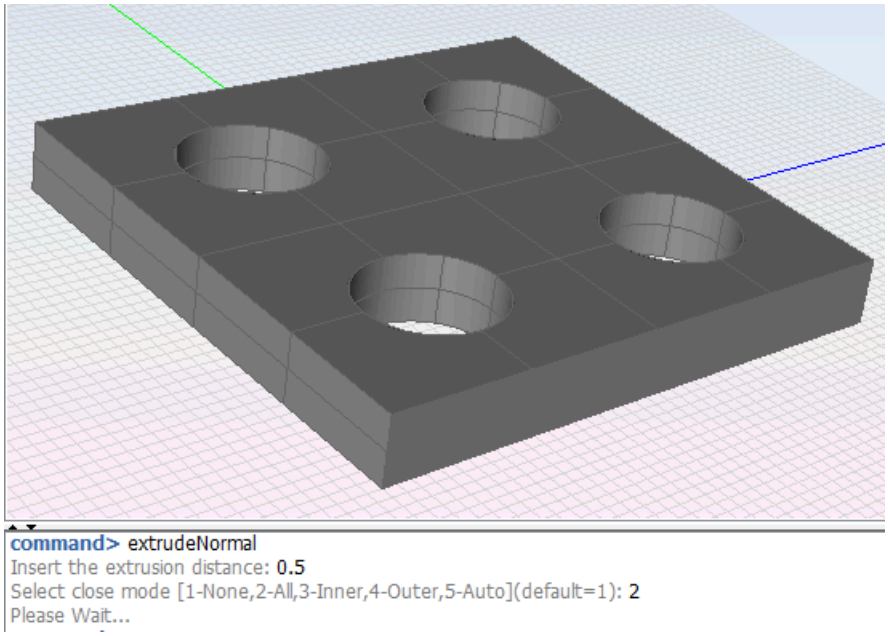


Figure 196: Generation of a closed object with the extrudeNormal command.

The first parameter is the extrusion distance that may be positive or negative.

The second parameter is the mode to close the result object.

1. **None:** Only the parallel layer is returned.
2. **All:** Connects all edges of the surfaces. If the surfaces are connected it generates duplicate surfaces on the outer edges.
3. **Inner:** Only connects the inner edges (holes) of the surfaces.
4. **Outer:** Only connects the outer edges (bounds) of the surfaces.
5. **Auto:** Study the topology and connect the naked edges. It is not recommended to use on curved surfaces with long extrusion distances because the parallel layer may not be well connected.

Box

This option allows the user to create a solid box. This function receives two corner points of the base of the box (these must be two points in the same diagonal and share the same Z coordinate) and the height of the box.

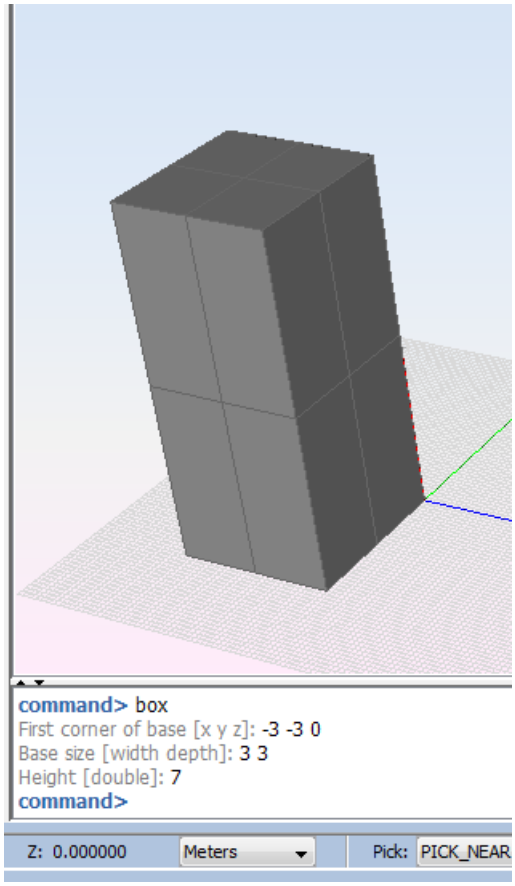


Figure 197: Creating a box

Cone

This option creates a solid cone. The required parameters are the center point of the base of the cone, the radius of the bottom base, the height and the radius of the top base of the cone.

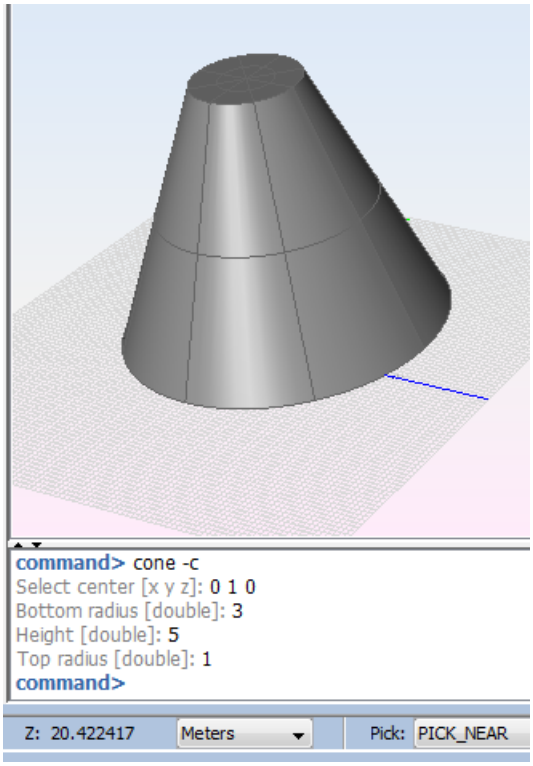


Figure 198: Creating a cone

Cylinder

This option allows the user to create a solid cylinder. Parameters needed for this function are the center point of the base, its radius and its height.

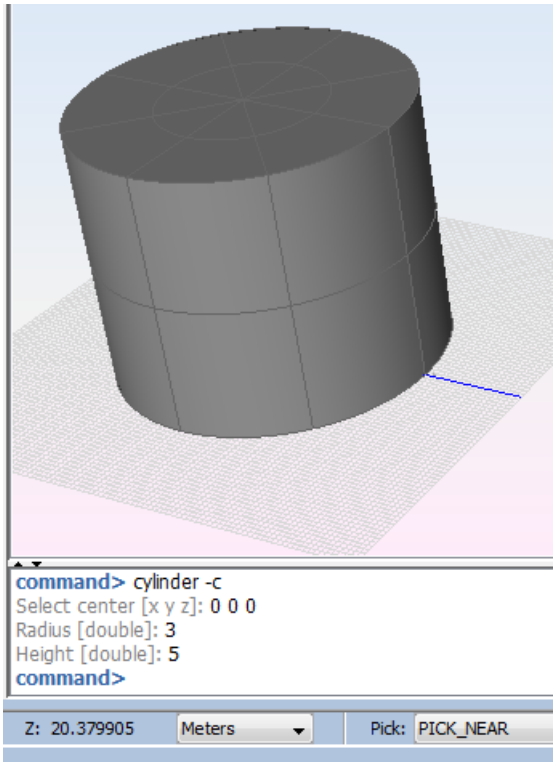


Figure 199: Creating a solid cylinder

Ellipsoid

This option allows the user to create an ellipsoid. The parameters required in this function are the center point, major radius and minor radius.

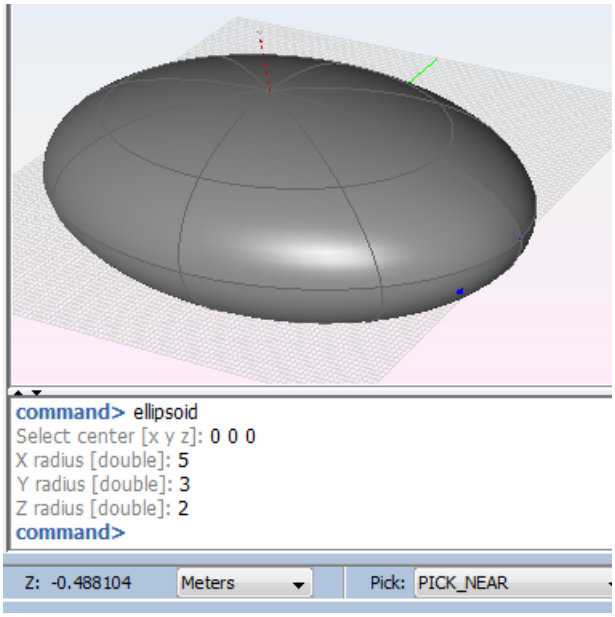


Figure 200: Creating an ellipsoid

Ogive

This option allows the user to create a spherically blunted-tangent ogive. It is a tangent ogive with its upper side smoothed by an sphere.

The base **center**, the **bottom radius** (measured in the the base center), the total **height** (measured for the tangent ogive without being spherically blunted) and the **top radius** of the spherical side are the parameters required.

A spherically blunted-tangent ogive is shown in next figure.

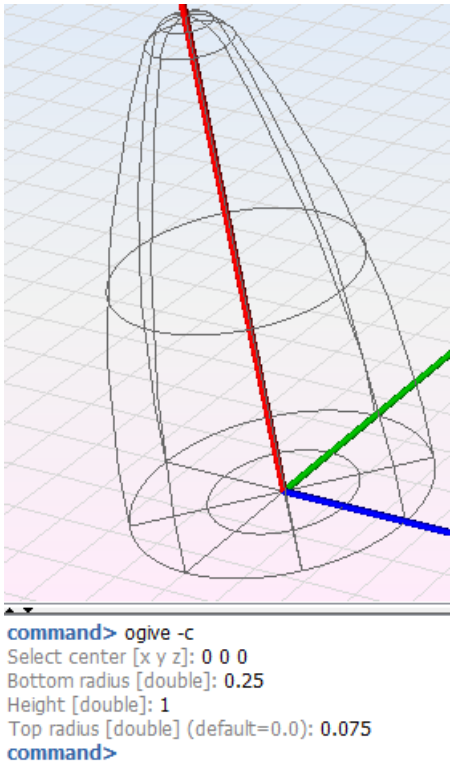


Figure 201: Spherically blunted-tangent ogive

A tangent ogive with the same parameters than the spherically blunted is shown in figure below. Note that the **height** of the generated ogive only agrees with the specified one when the **top radius** is set to 0 (tangent ogive), otherwise the ogive is shorter (spherically blunted-tangent ogive).

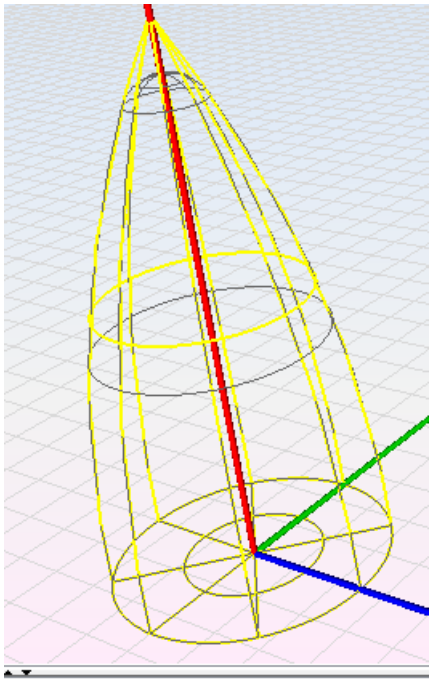


Figure 202: Tangent ogive compared to the spherically blunted-tangent ogive

Flare

This option allows the user to create a trapezoidal prism. This function receives the following parameters two corner points of the base (these must be in the same diagonal), the height and two corner points of the top base (again, in the same diagonal).

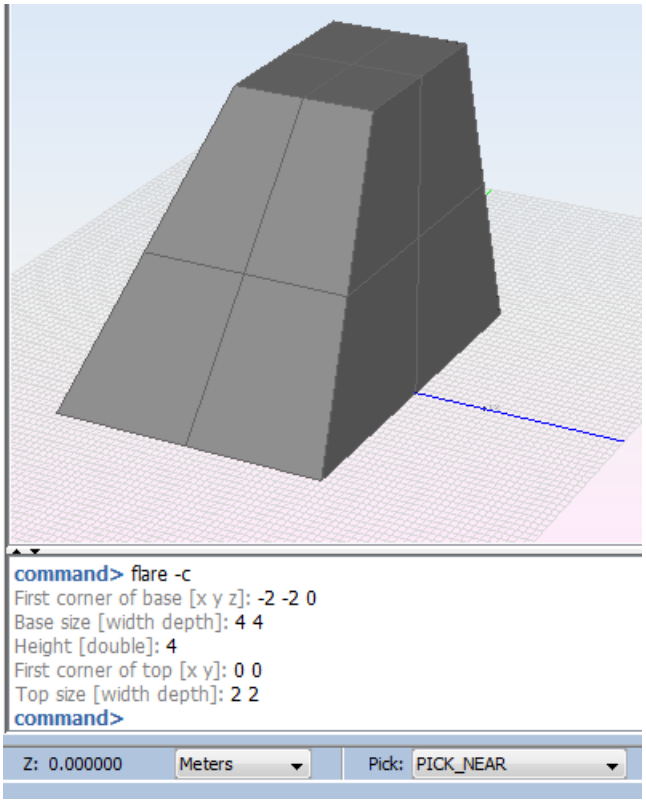


Figure 203: Creating a flare

Sphere

This option allows creating a sphere with both the center and the radius specified by the user.

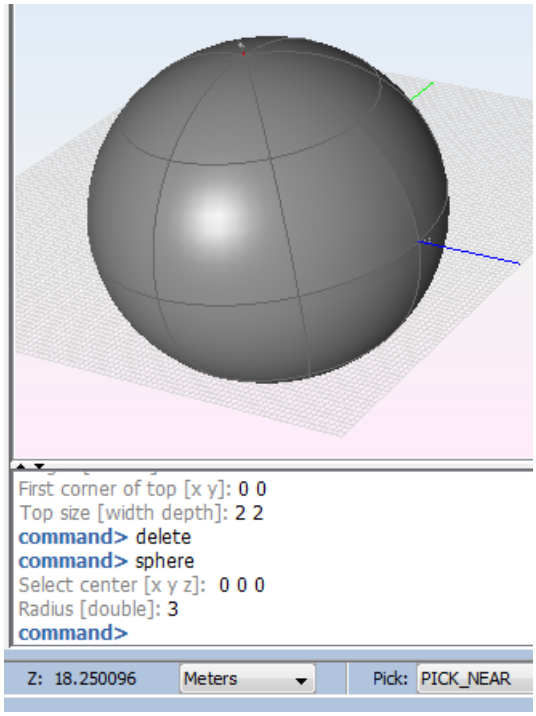


Figure 204: Creating a sphere

Torus

This option allows creating a torus with the center, the minor radius and the major specified by the user.

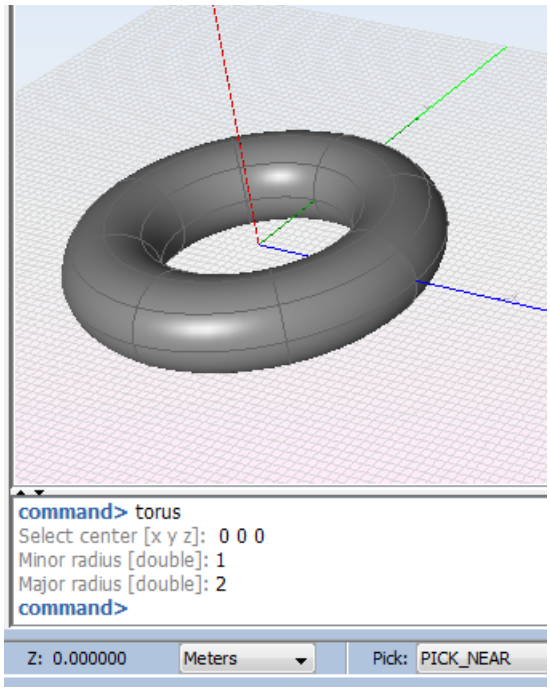


Figure 205: Creating a torus

Mesh

This menu contains the following options:

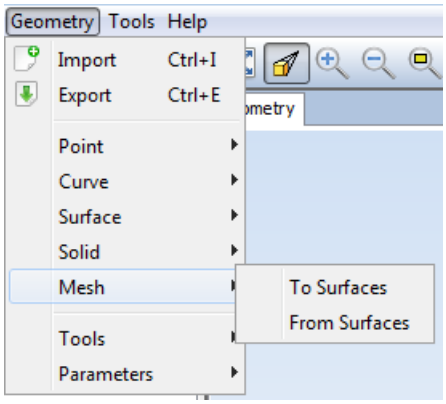


Figure 206: Mesh menu options

To Surfaces

This option allows the user to explode a mesh object into several open surfaces.

Note that high density meshes may overload the GUI behaviour after being converted to surfaces.

From Surfaces

This option allows the user to create a mesh object by joining a set of surfaces.

Note that the surfaces to be converted to a mesh must have degree 2 in their U and V dimensions without trimmed areas (which is equivalent to surfaces defined by three or four corner points). Surfaces with higher degrees are not allowed, so the command would returns an error.

2.5.3 Tools

This submenu contains several utility tools which can be used through the command line. Most of functions are useful to analyze parameters on the existing geometry, such as the dimensions of the objects, the curvatures, or the overlapped areas.

The available tools are shown in the following figure:

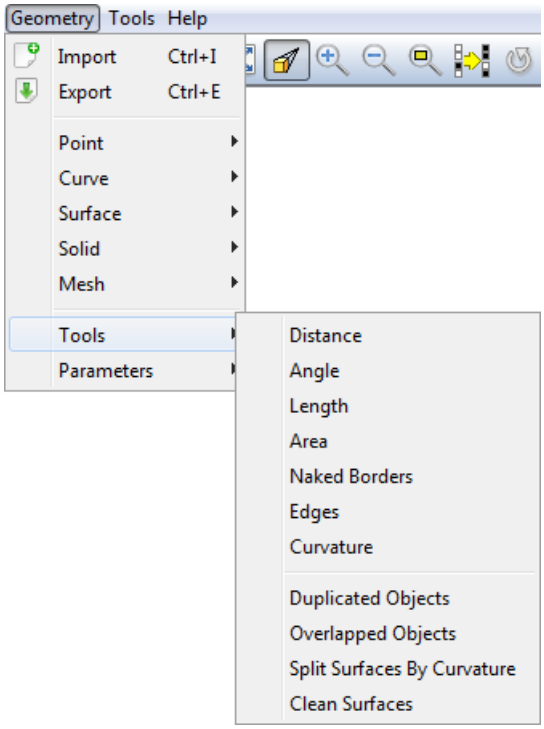


Figure 207: Geometry - Tools menu

Distance

This option allows the user to calculate the Euclidean distance between two given points.

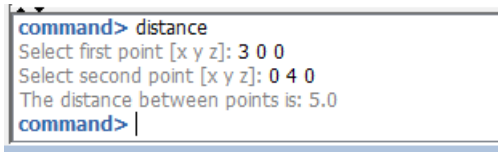


Figure 208: Calculating the distance between two points

Angle

This tool allows the user to calculate the angle (in degrees) between two given vectors. Vectors are specified by an initial point and an end point.

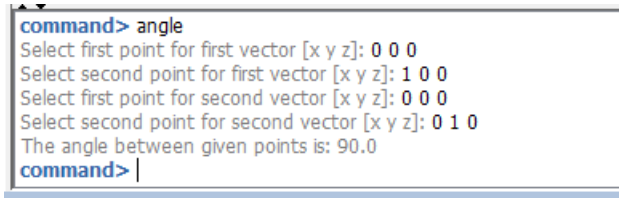


Figure 209: Calculating the angle between two vectors

Length

This tool computes the length of the selected curves. When multiple curves are selected, the length computed is the total sum of all the curves lengths.

```
command> circle
Center [x y z]: 0 0 0
Radius [double]: 1
command> length
The length of the selected curves is: 6.283185307179587 meters
command>
```

Figure 210: Length of a circle

Area

This tool computes the approximate area of the selected surfaces. When multiple surfaces are selected, the area computed is the total sum of all the surfaces areas.

```
command> sphere
Select center [x y z]: 0 0 0
Radius [double]: 1
command> area
Please Wait...
The area of the selected geometry is: 12.566370615048587 square meters
command>
```

Figure 211: Area of a sphere

Naked Borders

This tool shows the borders of the surfaces in the geometry that are not connected with another ones. The command only shows the naked borders, but it asks to the answer whether the detected borders must be added as new curves in the geometry or not, as shown in next figure.

Two different options may be used to use this command:

- If no parameters (analyzeNakedBorders) is introduced, the whole geometry is considered.
- If the parameter –s is added (analyzeNakedBorders –s), only the geometry specified by the user is considered.

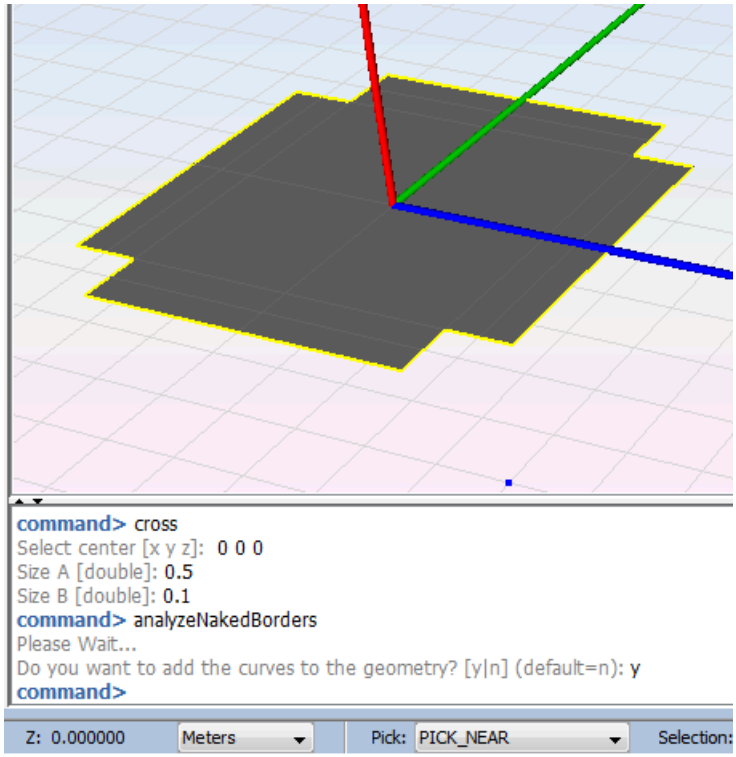


Figure 212: Analyzing the naked borders of a cross

Edges

This tool shows the borders of the surfaces in the geometry that are considered as Edges, it means that they are not connected with another ones or even being connected, the transition between the normal vectors of the connected surfaces is discontinuous (define an angle greater than zero). The command only shows the edges, but it asks to the answer whether the detected edges must be added as new curves in the geometry or not, as shown in figure below (in this case, the 12 resulting curves has been added and selected to be highlighted).

Two different options may be used to use this command:

- If no parameters (analyzeEdges) is introduced, the whole geometry is considered.
- If the parameter –s is added (analyzeEdges –s), only the geometry specified by the user is considered.

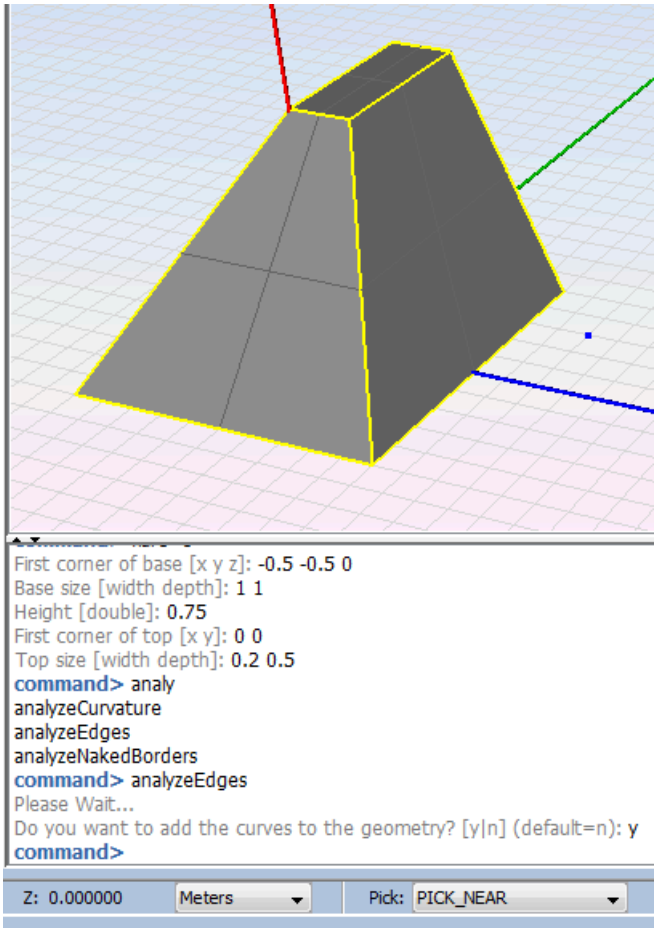


Figure 213: Analyzing the naked borders of a flare

Curvature

This option identifies approximately when the curves or surfaces vary their direction at a given angle. It may be useful for splitting strong curvatures, so the meshing-analyzing processes should be simpler. The command only shows the points where the curves varies the desired angles, but it also asks to the user whether the detected points must be added to the geometry or not.

The command only asks for the desired reference angle (in degrees).

The following figure shows an example of application:

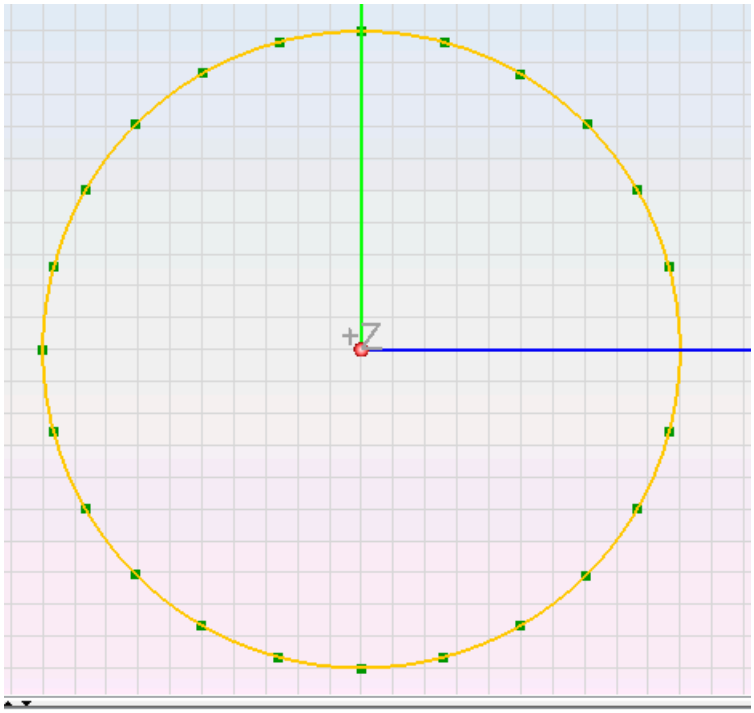


Figure 214: Example of curvature analysis on a circle

Duplicated Objects

This tool identifies the elements duplicated in the existing geometry. The duplicated elements are listed and the user must decide how to proceed with them.

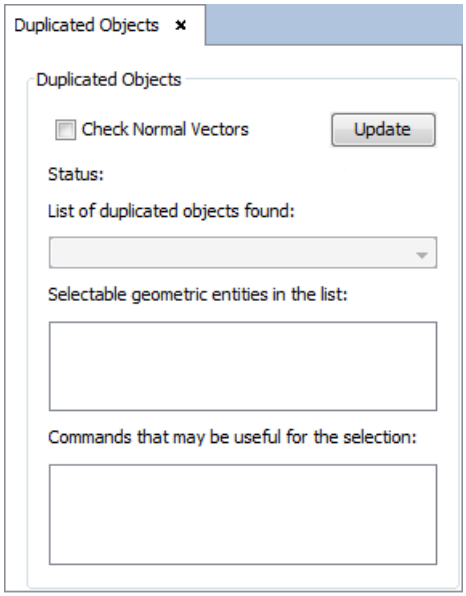


Figure 215: Duplicated Objects panel

The elements and options available in the tool are:

- **Check Normal Vectors** If this option is selected, the tool identifies as duplicated elements only those whose normal vectors of the duplicated surfaces (or direction for curves) are coherent.
- **Update Button** Press it to repeat the identification of the duplicated elements in the current geometry. It is necessary when the Check Normal Vector option or some elements of the geometry have been modified.
- **Status** Indicates if there are duplicated objects or not.
- **List of duplicated objects found** In case of identifying some duplicated elements, they are separated in different lists. When the current selected list is changed, its details are updated and its geometric elements are selected. If some geometric entity is deleted, the next time that the current list is selected, the deleted element won't be included. Two different duplicated elements may be identified:
- ◦ Full duplicated objects, that is, identical elements. If there is one or more of this type, the first list element contains all the duplicated entities identified (without including the original ones) with the name "All duplicated elements". However, the user also can inspect every duplicated shape (including both the original shape and its duplications), as they are listed in the next elements as "Duplicated XXX counter", where XXX is the type of elements (point, curve, surface or object) included, and counter is just a counter of each type.
  - ◦ Partial duplicated objects, that is, objects with some of their surfaces duplicated with other ones. They are listed as "Partial duplicated object Counter", where Counter is just a counter of partial duplicated elements.
- **Selectable geometric entities in the list** All the geometric elements included in a given list of duplicated objects are shown. These elements may be selected from this list. The index of the surface within any partial duplicated object is also shown.

- **Commands that may be useful for the selection** By depending on the type of the duplicated objects selected in the first list and the existing geometry in it, some commands are suggested to be applied. Click twice quickly on the desired command to run it.

In the following, an example of a box and two identical planes is analysed to test this tool.

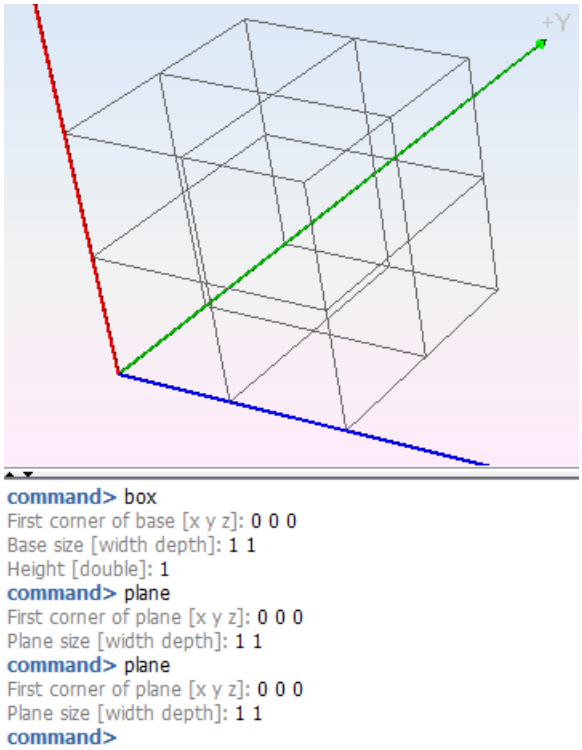
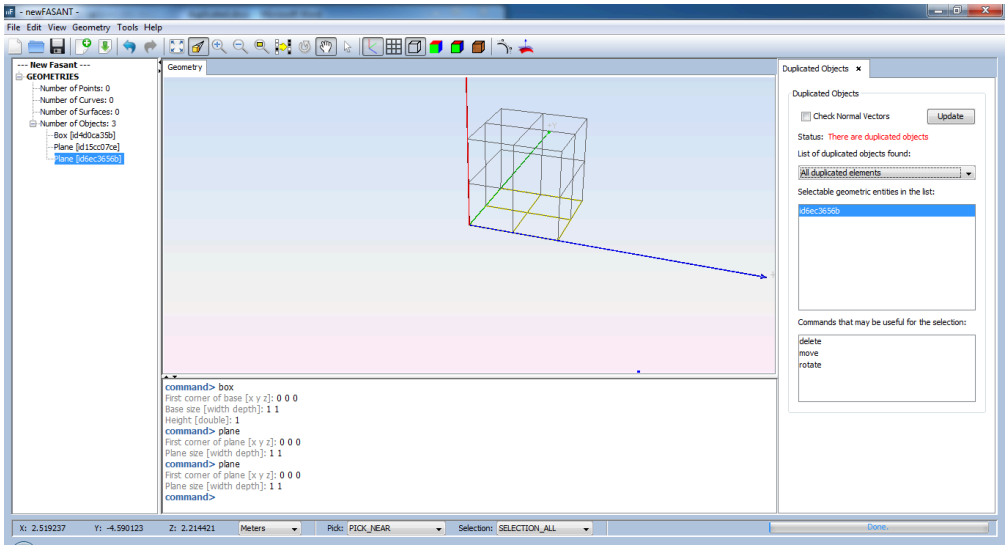


Figure 216: Example for duplicated objects identification.

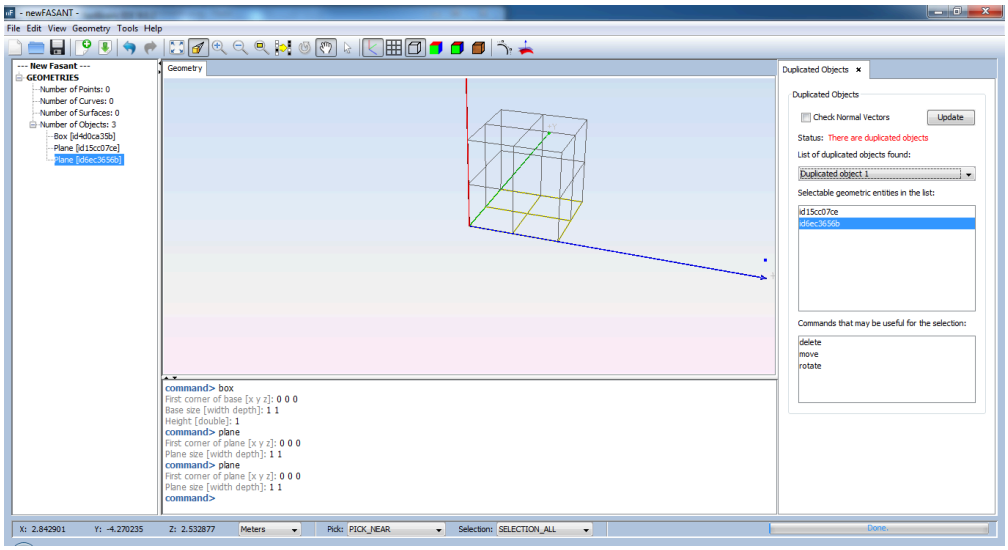
Click on **Geometry – Tools – Duplicated Objects** to check the duplicated elements identified in the geometry.

With the default configuration, the next elements have been identified:

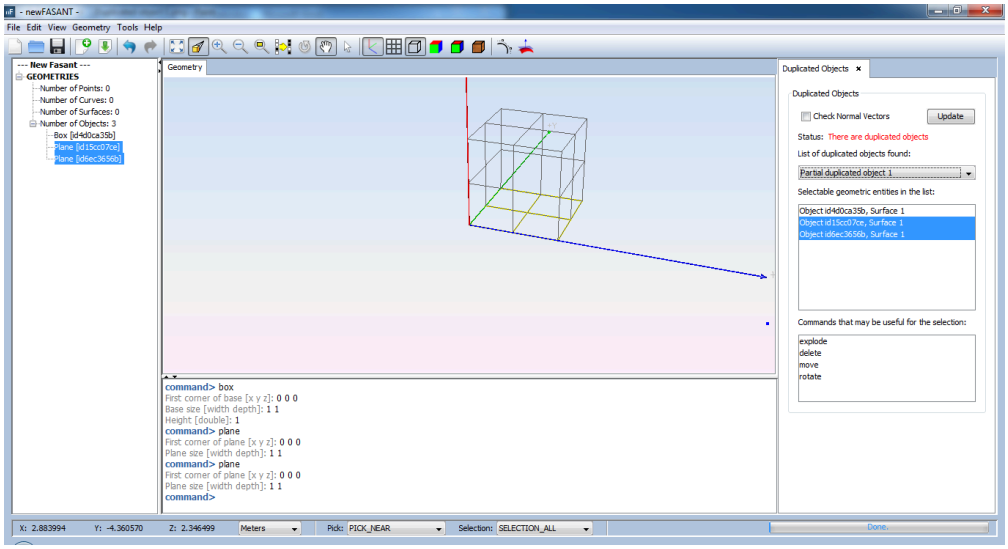
- **All duplicated elements**. Only the second plane is listed. All the duplicated elements included in this list could be deleted to solve the identical entities duplication.



- **Duplicated object 1**. The two planes are listed. The user should decide whether changing or not one of the involved elements.

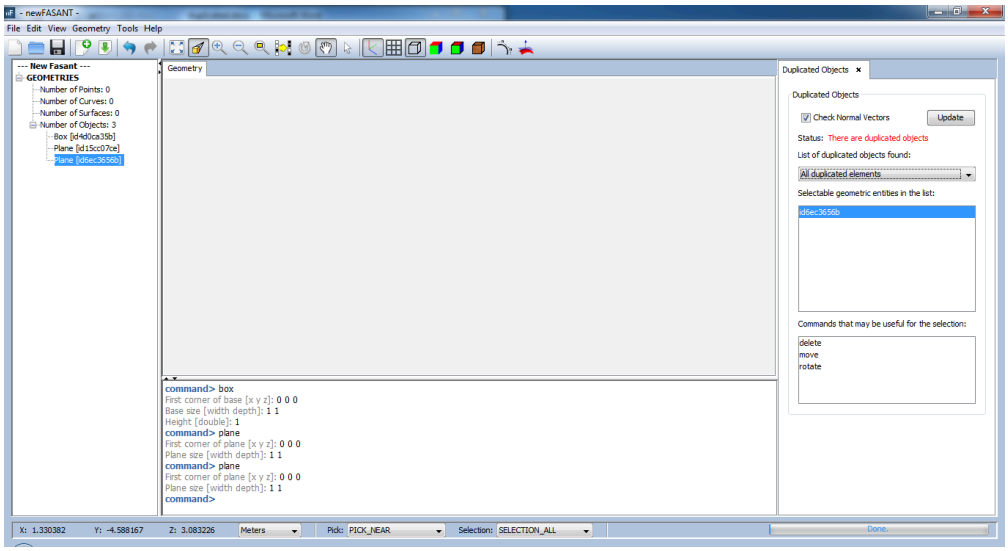


- **Partial duplicated object 1**. The local index of the surface duplicated within every object is shown.

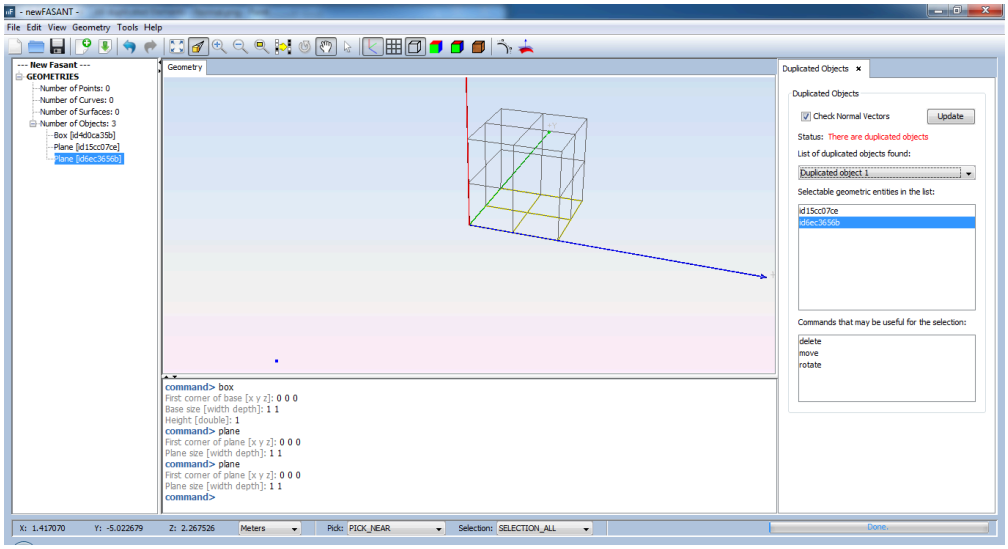


Without changing the existing geometry, select the option Check Normal Vector and then click on the Update button to re-run the identification. The next lists are shown:

- **All duplicated elements**. Only the second plane is listed.



- **Duplicated object 1**. The two planes are listed because their normal vector is coherent.



- **Partial duplicated objects** are not listed because the normal vectors of the planes are opposite to the duplicated surface of the box.

Without changing the existing geometry again, unselect the option Check Normal Vector and then click on the Update button to re-run the identification and come back to the first configuration. Then select the Partial duplicated object 1 in the list. Note that the two planes are directly selected, so click on delete command. After deleting the two planes, the lists contain the following elements:

- **All duplicated elements**. Now the list is empty.
- **Duplicated object 1**. Now the list is also empty.
- **Partial duplicated object 1**. Only the box is listed because the two planes have already been removed.

Click on **Update** button to re-run the identification. The **Status** has changed and no duplicated objects are identified.

Overlapped Objects

This tool identifies the elements overlapped in the existing geometry. The overlapped elements are listed and the user must decide how to proceed with them.

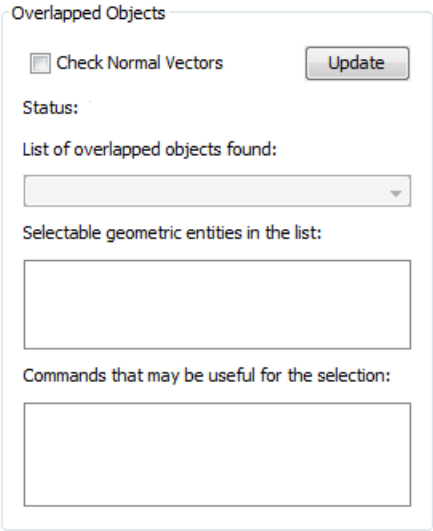


Figure 217: Overlapped Objects panel

The elements and options available in the tool are:

- **Check Normal Vectors** If this option is selected, the tool identifies as overlapped elements only those whose normal vectors of the overlapped surfaces (or direction for curves) are coherent.
- **Update Button** Press it to repeat the identification of the overlapped elements in the current geometry. It is necessary if the Check Normal Vector option or some elements of the geometry have been modified.
- **Status** Indicates whether there are overlapped objects or not.
- **List of overlapped objects found** In case of identifying some overlapped elements, they are separated in different lists. When the current selected list is changed, its details are updated and its geometric elements are selected. If some geometric entity is deleted, the next time that the current list is selected, the deleted element won't be included.
- **Selectable geometric entities in the list** All the geometric elements included in a given list of overlapped objects are shown. These elements may be selected from this list. The index of the surface within any partial overlapped object is also shown.
- **Commands that may be useful for the selection** By depending on the type of the overlapped objects selected in the first list and the existing geometry in this list, some commands are suggested to be applied. Click twice quickly on the desired command to run it.

In the following, an example of a box and a plane is analysed to test this tool.

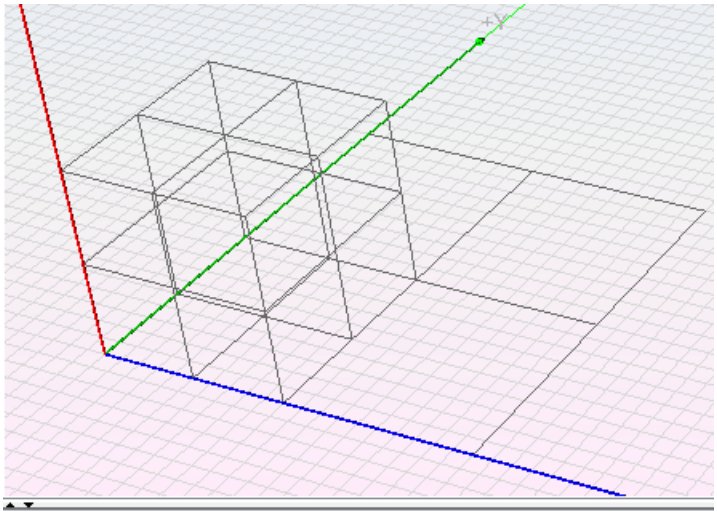
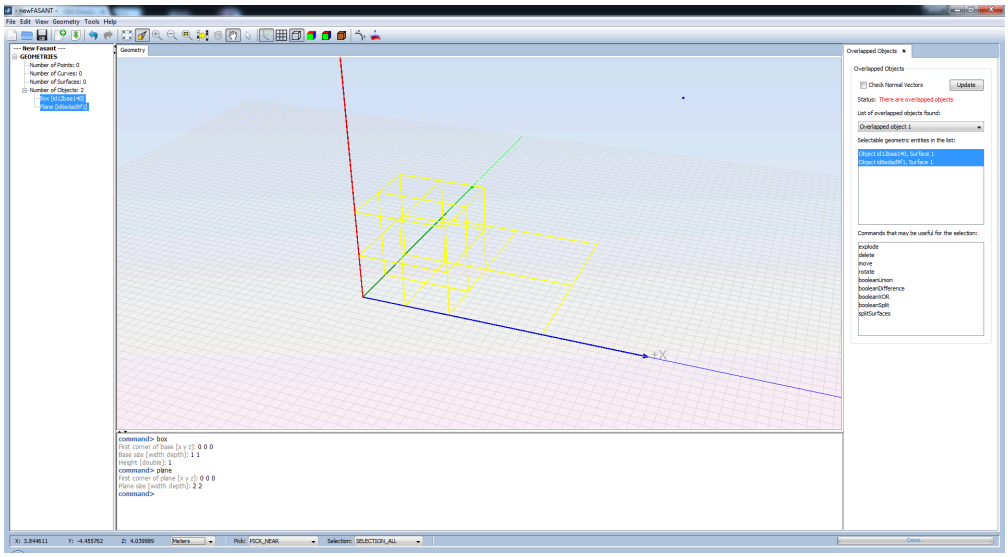


Figure 218: Example for overlapped objects identification.

Click on **Geometry – Tools – Overlapped Objects** to check the overlapped elements identified in the geometry.

With the default configuration, the next elements have been identified:

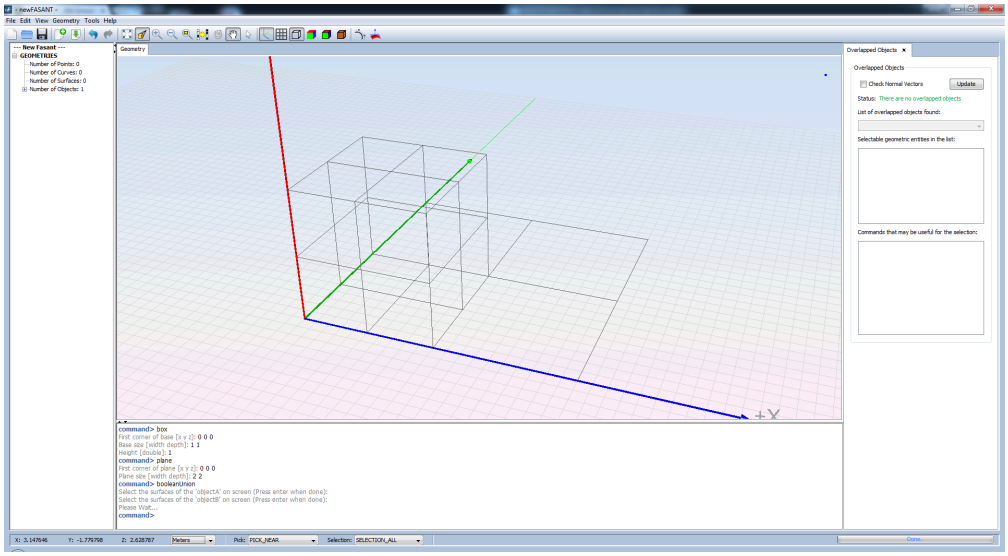
- **Overlapped object 1**. The box and the plane are listed (both are objects). The local index of the surface duplicated within every object is shown.



Without changing the existing geometry, select the option Check Normal Vector and then click on the Update button to re-run the identification. Then no overlapped objects are identified, as the vector normal of the plane is in the opposite direction to the lower surface of the box.

Without changing the existing geometry again, unselect the option **Check Normal Vector** and then click on the **Update** button to re-run the identification and come back to the first configuration. Then we are going to solve the overlap issue by using one of the suggested commands. Click twice on booleanUnion, and then select the box as "objectA", and the plane as "objectB".

After finishing the booleanUnion operation, click on **Update** button to re-run the identification. The **Status** has changed and no overlapped objects are identified, as the overlapped part of the plane has been removed.



Split Surfaces by Curvature

This tool analyzes the curvature of the selected surfaces according to a specified angle in degrees. The surfaces with such curvatures are listed with the list of parametric coordinates where this criterion may be satisfied.

By default, not all the parametric coordinates will be always enabled because the analysis is performed taking into account the neighboring surfaces, so more divisions may appear because of the adjacent surfaces. The tool enables only the divisions that provide the angular step.

This tool only takes into account NURBS surfaces, so objects must be exploded (explode command) to be analyzed and meshes must be converted (meshToSurfaces command). Note that meshes are made of almost planar elements, so analyzing its curvature would not have real interest.

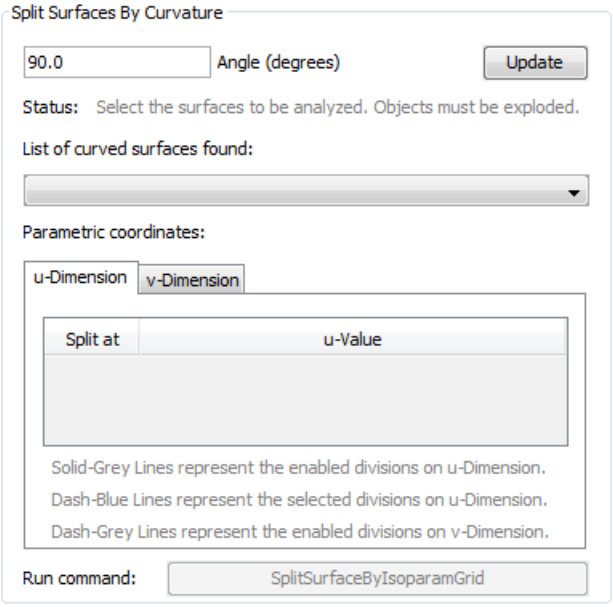


Figure 219: Split Surfaces By Curvature panel

The elements and options available in the tool are:

- **Angle** The reference angle for the curvatures detection.
- **Update Button** Press it to repeat the identification of the curved surfaces in the selected list. It must be pressed if the Angle is modified.
- **Status** Indicates if there are curved surfaces or not.
- **List of curved surfaces found** In case of identifying some curved surfaces, they are separated in this list. The list is updated whenever some of the considered surfaces is removed of the geometry or modified with some opperations such as group, split or boolean operations,...
- **Parametric coordinates** Table with the U and V coordinates where a curvature deviation has been detected for the selected surface in the list. The detected parametric coordinates can be selected to be visualized in Dash-Blue lines, but only the enabled coordinates are always represented in grey lines.
- **SplitSurfaceByIsoparamGrid button** The curved surface selected in the list may be split by the isoparam lines corresponding to the parametric points enabled in the U-V tables. The button only can be used when at least one parametric coordinate is enabled.

In the following, an example of a sphere is analyzed to test this tool. Remember that the sphere must be exploded to be analyzed with the tool.

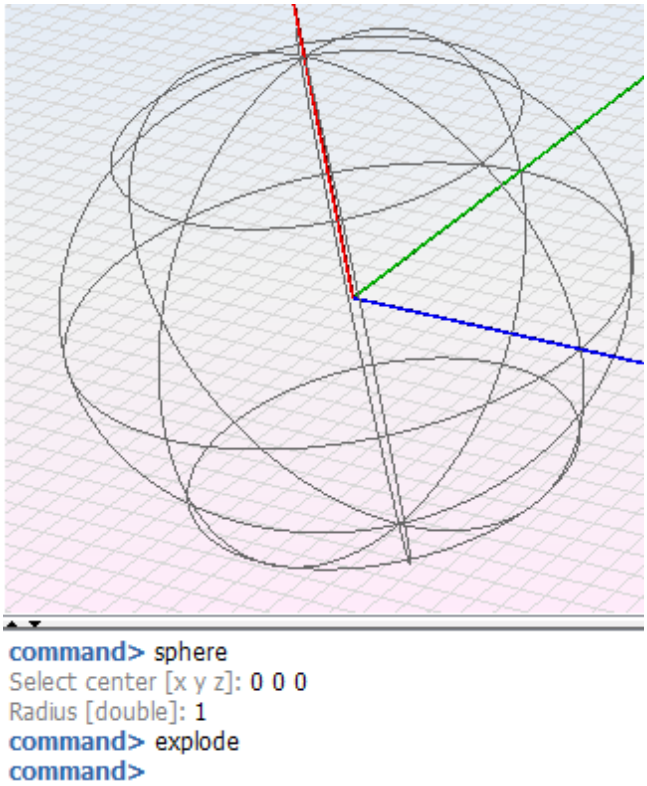


Figure 220: Example for splitting an sphere by curvature

Click on **Geometry – Tools – Split Surfaces by Curvature** to open the panel.

By selecting all the surfaces of the exploded sphere, the next two test are performed:

- **Angle of 30 degrees**. Select all the surfaces, insert a curvature angle of 30 degrees and then click on **Update** button.

As all the surfaces of the sphere has the same shape and a curvature of 90 degrees in each parametric dimension, they are listed with 2 parametric divisions in U and V dimensions each one. The next figure shows the representation of the second surface in the list with all the parametric coordinates on the table enabled and only one of then selected.

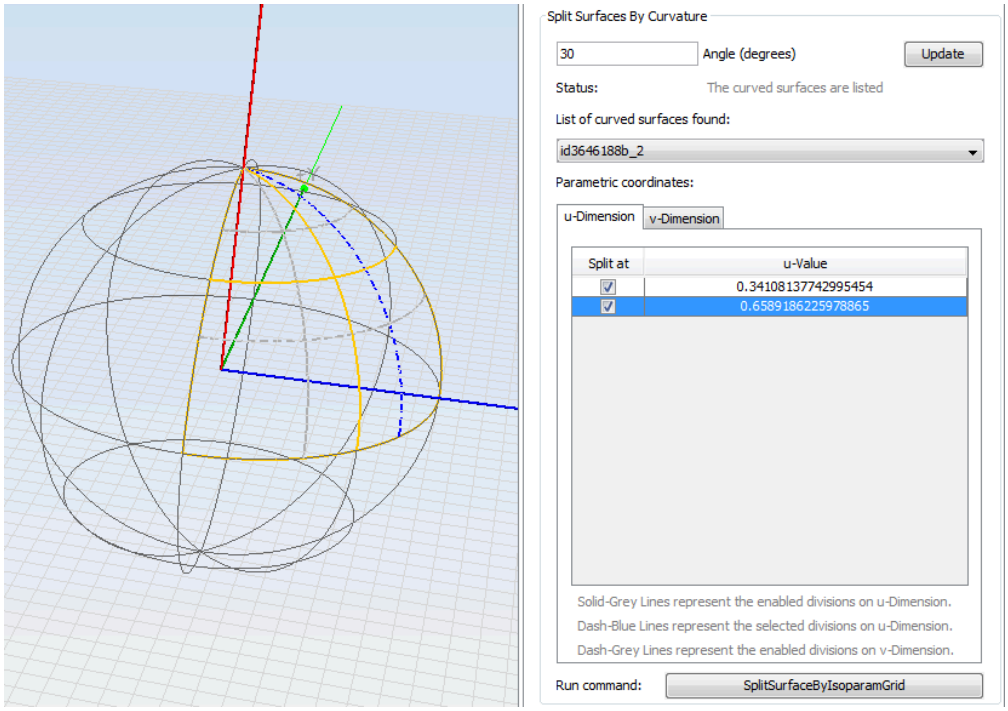


Figure 221: Analysis of curvature on the sphere performed with 30 degrees

- **Angle of 20 degrees**. Select all the surfaces, insert a curvature angle of 20 degrees and then click on update button.

As all the surfaces of the sphere has the same shape and a curvature of 90 degrees in each parametric dimension, and the selected angle is not a divisor of 90, the curvature of 20 degrees may be different according to the start reference point that is considered. The next figure shows the representation of the second surface in the list with the default parametric coordinates on the table enabled and the disabled ones in V-dimension selected. In this figure is shown that the curvature analysis provides two different solutions the enabled and the disabled divisions.

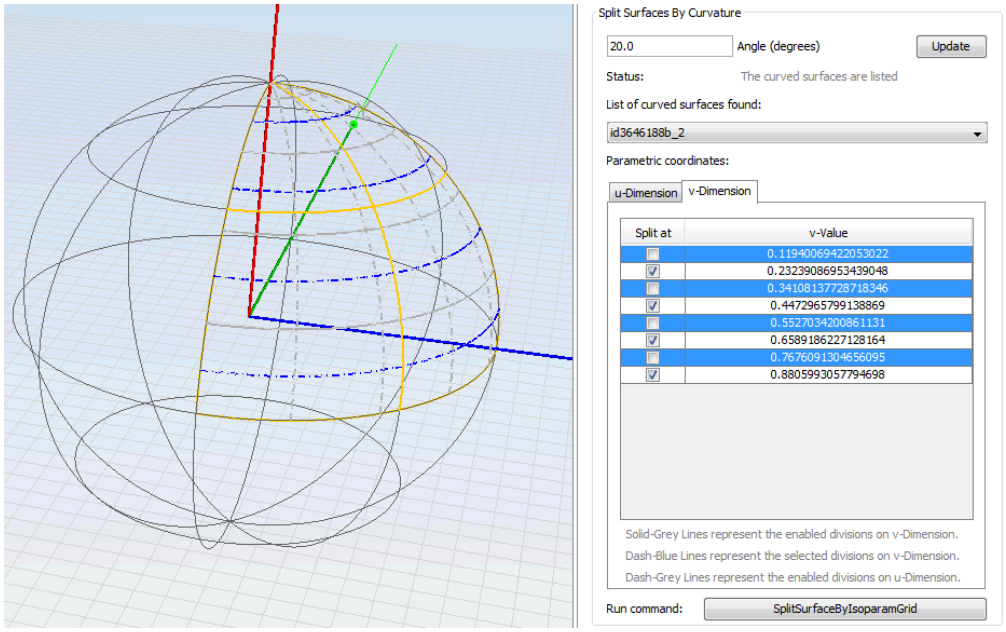


Figure 222: Analysis of curvature on the sphere performed with 20 degrees.

Click on **SplitSurfaceByIsoparamGrid** button to finish the example by dividing the surface with the enabled parametric coordinates. After run the command, the list of curved surfaces is updated and the first one is selected, as shown in next figure.

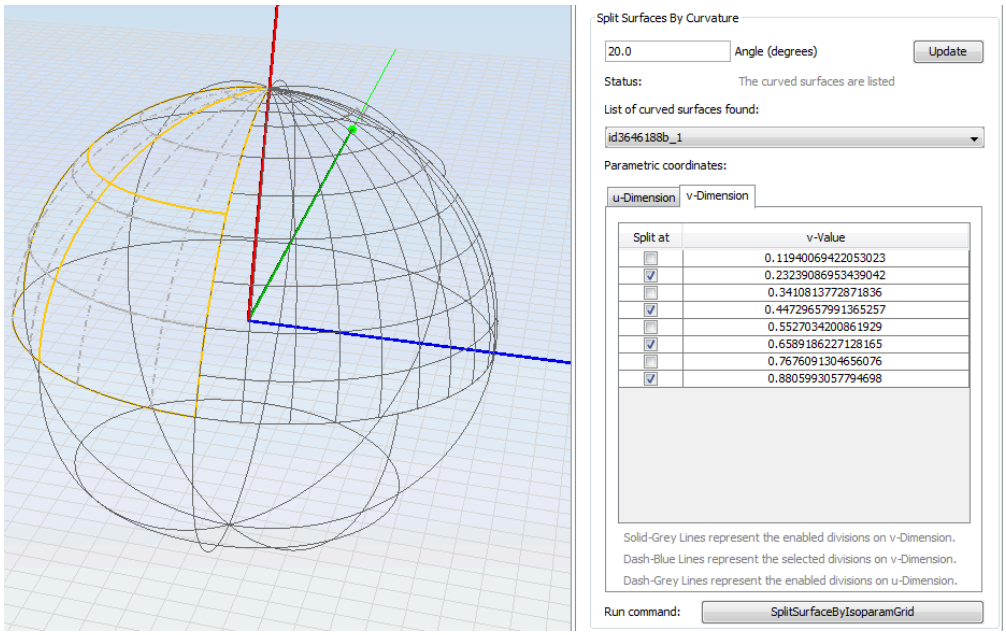


Figure 223: Surface of the sphere divided by the default points for 20 degrees

Clean Surfaces

This tool allows to filter the selection of multiple surfaces by applying different criterions. Use the tool if the geometry may contain tiny surfaces.

The selected surfaces are filtered according to the selected **Criterion**:

- **Smallest dimension** only the surfaces with their shortest side lower than the **Smaller than** parameter (in **Units**) are kept in the original selection.
- **Area** only the surfaces with their area lower than the **Smaller than** parameter (in square **Units**) are kept from the original selection.
- **Perpendicular to direction** only the surfaces with their normal vector perpendicular to the direction given by the **XYZ** fields are kept from the previous selection. The direction from the origin of coordinates to any point given by the mouse-click may be computed by selecting the **Pick** option.

After choosing the desired **Criterion** and its parameter, select the surfaces to be analyzed (**Objects** must be exploded) and click on **Update** button.

The **Status** label shows tips for filtering surfaces or information about the filtered surfaces.

The filtered surfaces are listed in the **Detected tiny surfaces** table. Any surface within the list may be selected, and also the multiple selection is enabled.

- Click on **Delete** button to remove the surfaces selected within the **Detected tiny surfaces** list.
- Click on **Delete All**button to remove the all the surfaces listed in the **Detected tiny surfaces** table.

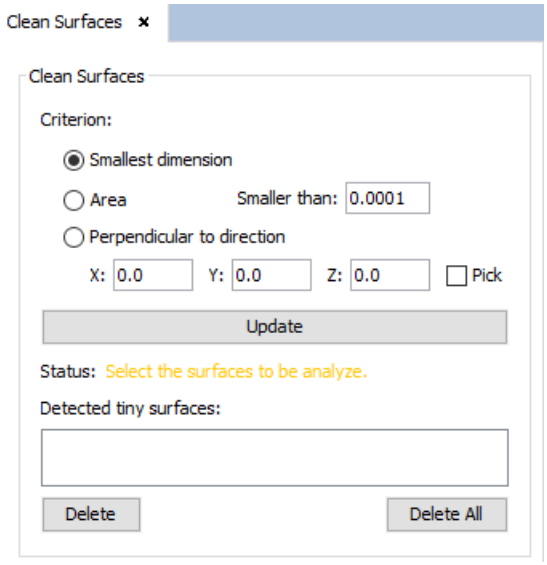


Figure 224: Clean Surfaces panel

2.5.4 Parameters

The options in this submenu allow the user to specify certain parameters that act as variables for the geometry. This allows parametrizing our geometries in order to be able to run several simulations

more easily by tuning the different parameters of the geometry (thus not being necessary to redo the geometry for different simulations).

This procedure works in the following way the user defines several parameters and their value arrays (i.e. the possible values for the parameters) and assigns them to the geometry. This creates several states for the geometry, which we will call “steps”. A step is a geometry state in which each of the parameters takes the value of a specific index of its set of values. As an example, in step 1, every parameter takes the value of the first value of its value array; in step 2 the second value and so on. This means that there will be as many steps as the length of the larger value array of those parameters used in the geometry.

In this way, the user can use parameters while defining the geometry. For example, the following figure shows how to define a cube with a side length depending on a parameter called “myLength”.

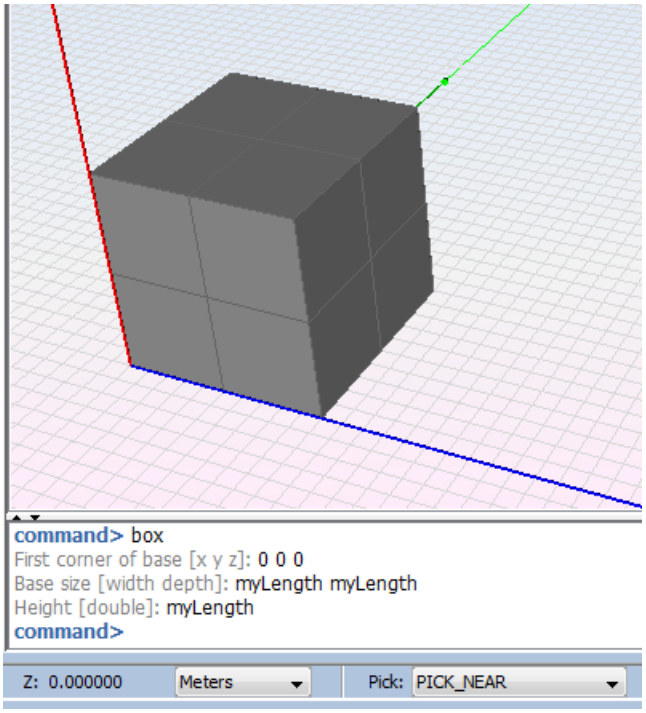


Figure 225: Creating a parameterized cube

When using a parametrized geometry, the user will be able to view the results for each step in the options of the Show Results menu.

The options explained below will allow adding parameters and visualizing the effect of these parameters in the geometry view.

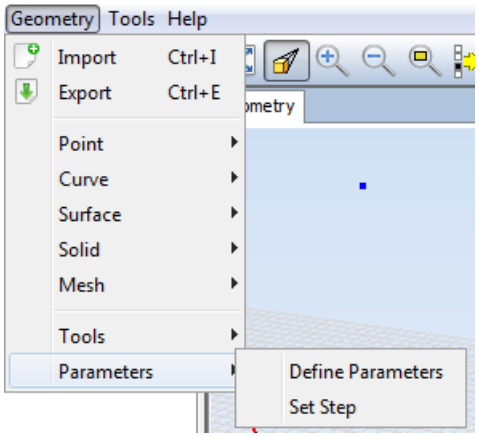


Figure 226: Options in the Parameters submenu

Define Parameters

After selecting this option, the following panel will be shown:

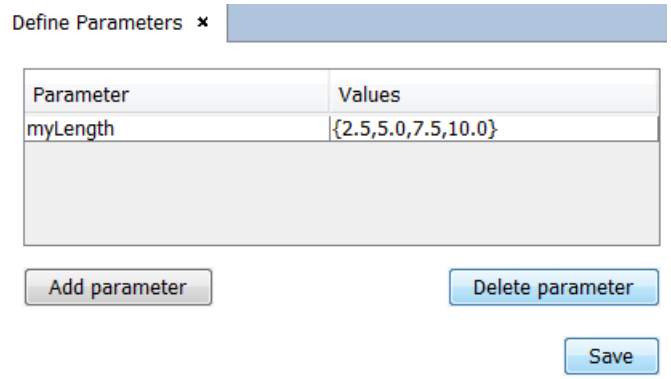


Figure 227: Define Parameters panel

In this panel the user can define the parameters needed to parametrize the geometry. In the figure above, we have defined a parameter with a value varying in the array of values (2.5, 5.0, 7.5, 10.0). The user can define new parameters with the **Add parameter** button, which will add a new row in the parameters table.

The **name of the paramaters** may satisfy the following **requirements**:

- Blank spaces are not allowed. For example, "new parameter" is not a valid name.
- No empty names are allowed.
- The name must be made-up only by alphanumeric characters and the "_" character. For example, both "parameter1", "parameter_1", "d" and "newParameter" are valid names, while both "new parameter", "parameter&1", "", and "?" are not valid names.

The user can set the array of values of each parameter manually. There are three ways to define parameters:

- **Implicit linear values** with this way to parametrize the geometry, the user indicates the values of the parameter. The number of values introduced will define the number of steps that the objects parametrized obtain with this parameter. The syntax required for the implicit linear parameters definition is a list of values, separated with commas and surrounded between "{" and "}" characters, as in the below example.

{2, 5, 12.5, 15}

- **Range of linear values** with this option, the program will assign a new array of values to the parameter. The values of this array will be uniformly chosen between the intervals, with as many values as the number introduced after the range. The syntax required for the linear parameters definition is an interval defined by its initial and final values, separated with commas and surrounded between brackets, and followed by the number of samples (integer positive value) to be taken. See the following example to parametrize in this way:

[0.0,3.0] 4 The program will assign an array of values (0, 1, 2, 3)

Values defined by functions *with this option, the user can specify an expression using other parameters and built-in functions*

. If the expression is a constant value (i.e. does not use parameters) the set of values generated for the parameter will be a singleton set containing that constant value. Otherwise, i.e. the function depends on previously defined parameters, the set of values will contain as many values as the smallest array used, and the actual values will be the result of the expression. See the following examples to parameterize in this way:

cos(90) Array of a singleton set.

sin(a) Array of different values using the "a" parameter created previously.

The auxiliary parameters are automatically defined when an operation is performed on an existing parameter, and their name starts always with the ' **$**' character to be identified. In this way, for example, given a parameter named 'Length', the using of 'Length/2.0' at any operation creates (or uses the previously created the following times) an auxiliary parameter named '$x', where 'x' notation is an internal numeration.

The user can also delete existing parameters by selecting them and clicking the **Delete parameter** button.

Finally, the user may save the changes in the parameter list by pressing the **Save** button.

Set Step

This option allows the user to visualize an animation of the geometry as the value of each parameter cycles through the set of values of the parameter. When the user selects the Set Step option, the following panel will show:

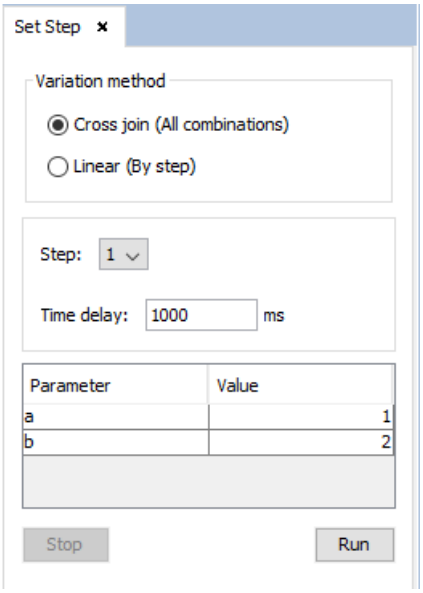


Figure 228: Set Step panel

This panel allows the user to select a step to view on the geometry view. There are two available variation methods used to generate the steps:

- **Cross join** Generates a step for every combinations of the possible values of the parameters used in the geometry. Parameters that are not used in the geometry will not be not considered.
- **Linear** The values of the parameters are the values at a specific index (determined by the step) of the array of values. There will be as many steps as the maximum length of the value arrays of those parameters used in the geometry.

It is also possible, as mentioned before, to see an animation of the geometry view on each step. In order to do this, the user may change the time delay between steps and press the **Run** button.

2.6 Tools

When you select a new module, the following menu appears at the top of the screen:

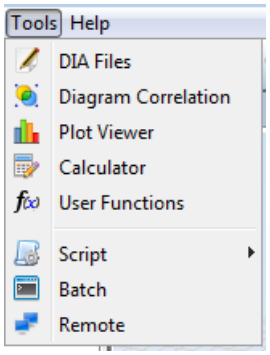


Figure 229: Tools Menu

2.6.1 DIA Files

In this menu we can see a set of options used to feed an antenna with the data represented in the table below.

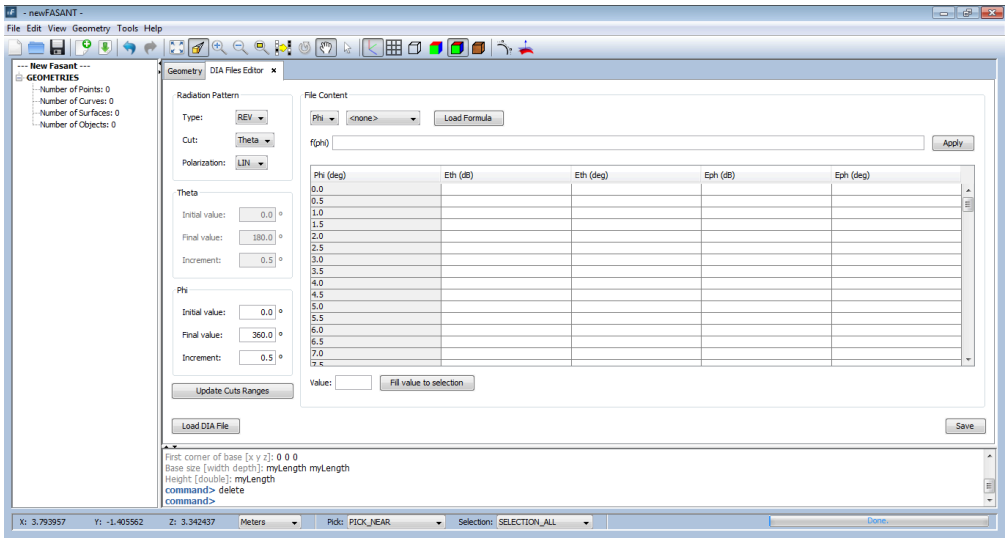


Figure 230: DIA Files Editor

The **radiation pattern** section defines the three types that can be chosen and, depending on each one, a different table is then displayed REV, RV2 and 3DE. The general features of each one of these options are explained below:

Radiation pattern: In this section we can see three options the type of radiation (mentioned above), angular range to be considered, only visible in REV type, and Polarization, to indicate a linear or circular polarization.

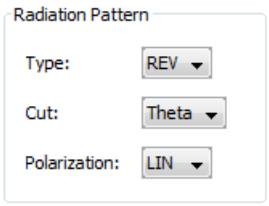


Figure 231: Radiation Pattern

Theta and Phi sections used to mark the initial and final values in degrees that define the angular sweeps.

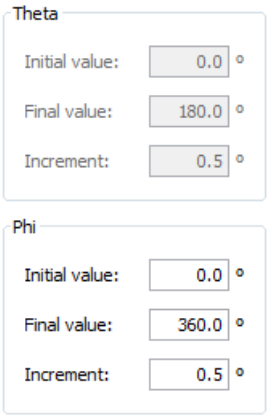


Figure 232: Theta and Phi Menu

File Content In this section the user can select an operation and load in the formula using the button **Load Formula**, and introduce a customized mathematical formula. To do this the Theta o Phi variables are required, as well as the operation, shown in the drop down menu.

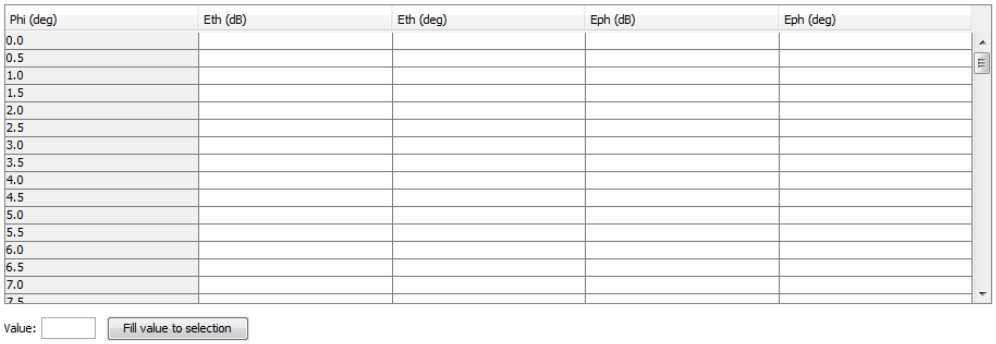


Figure 233: File Content

- Operations of File Content operations available to be applied to a given formula are as follows
- *abs*
  (absolute value)

  acos
  (arccosine)

acosr
(arccosine in radians)

asin
(arcsine)

asinr
(arcsine in radians)

atan
(arctangent)

atanr
(arctangent in radians)

ceil

cos
(cosine)

cosr
(cosine in radians)

deg
(value in degrees with a circumference of 360 parts)

exp
(elevate a number to an exponent)

floor

ln
(natural logarithm)

log
(logarithm with the introduced base)

log10
(logarithm in 10 base)

max
(maximum)

min
(minimum)

mod
(module)

pow
(to raise a number to the power)

rad
(value in degrees with a circumference of 360 parts)

random
(random value)

round
(round up of value)

sin
(sine)

sinr
(sine in radians)

sqrt
(square root of value)

tan
(tangent)

tanr
(tangent in radians)

After creating the formula, the user needs to select the cells in the table where the formula is going to be applied and press **Apply.**

To insert multiple values in the table, insert the number in Value and select the required cells and then press **Fill value to selection**.

In order to reset the initial values, press **Update Cut Ranges**.

Finally to save the data calculated using the formula, press **Save** or load previous data with the button **Load DIA File**.

After seen the basics, let us overview each type options:

REV The user can select **Phi** degrees on **Radiation pattern** and the table will include the following information:

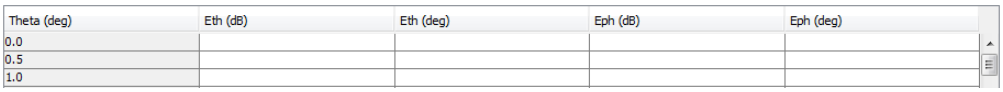


Figure 234: REV File Content

The attribute **Theta** describes the angle in degrees, **Etheta (dB)** and **EPhi (dB)** describe the module and **Etheta(deg)** and EPhi(deg) the phase values.

RV2 With this option it is not possible to select **Phi** degrees on Radiation pattern. In this case the table will contain the following information:

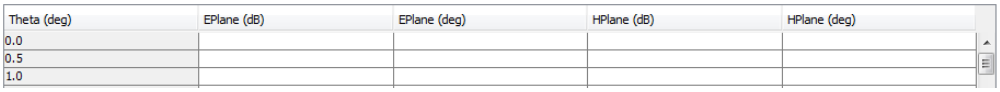
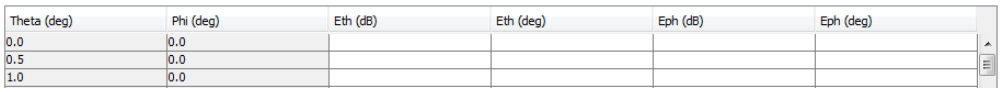


Figure 235: RV2 File Content

In this case the field is expressed in the E-plane components and the H-plane components (module in dB and phase in degrees), instead of using the theta and phi components as in the previous case.

3DE With this option it is not possible to select **Phi** degrees on **Radiation pattern**. In this case the table will contain the following information:



Theta (deg)	Phi (deg)	Eth (dB)	Eth (deg)	Eph (dB)	Eph (deg)
0.0	0.0				
0.5	0.0				
1.0	0.0				

Figure 236: 3DE File Content

The table values in this case are the same as those shown for the **REV** option. In this case the full pattern is provided and no revolution condition is assumed.

DIA File Format

File Structure

This section describes the structure of the radiation pattern files in newFASANT.

These files are plain text files whose format depends on the kind of radiation pattern. The radiation pattern file contains:

Title	Description	Possible Values
File description	Three lines describing the file.	Any. These lines will be interpreted as comments.
Cut	**Only if radiation pattern is REV (revolution).** The cut in theta or phi.	TH=<angle> PH=<angle> Comments <angle> is a decimal number representing the angle in degrees.
Polarization	**Only for linearly polarized antennas.** Polarization of the radiated field.	Eth Eph Eth, Eph
Polarization	**Only for circularly polarized antennas.** Polarization of the radiated field.	Ui Ud Ui, Ud Comments Use Ui for LHCP polarized antennas, Ud for RHCP polarized antennas, and Ui, Ud for both LHCP and RHCP polarized antennas.

Title	Description	Possible Values
Blank line	A blank, empty line (necessary).	None.
Sweep	Sweep in the direction of theta and phi.	<initial_theta> <num_points_theta> <step_theta> <initial_phi> <num_points_phi> <step_phi> Comments All initial angles and steps are specified in degrees. The values are delimited by, at least, one blank space.
Blank line	A blank, empty line (necessary).	None.

And then, for each sampled direction (note brackets denote optional values and must not appear in the radiation pattern files):

Title	Description	Possible Values
Data (REV)	**If radiation pattern of the antenna is REV (revolution).** Value of the field in the direction.	<theta> [<phi>] [<eth_amplitude><eth_phase>] [<phi_amplitude><phi_phase>].
Data (RV2)	**If radiation pattern of the antenna is RV2 (semi-revolution).** Value of the field in the direction.	<theta> [<phi>] <eplane_amplitude> <eplane_phase><hplane_amplitude> <hplane_phase> Comments HPlane phase must be equal to EPlane phase + 180 degrees.
Data (3DE)	**If radiation pattern of the antenna is 3DE (generic).** Value of the field in the direction.	<theta> [<phi>] [<eth_amplitude> <eth_phase>] [<phi_amplitude> <phi_phase>]

It is necessary to define the field values for each point defined in the **Sweep** line.

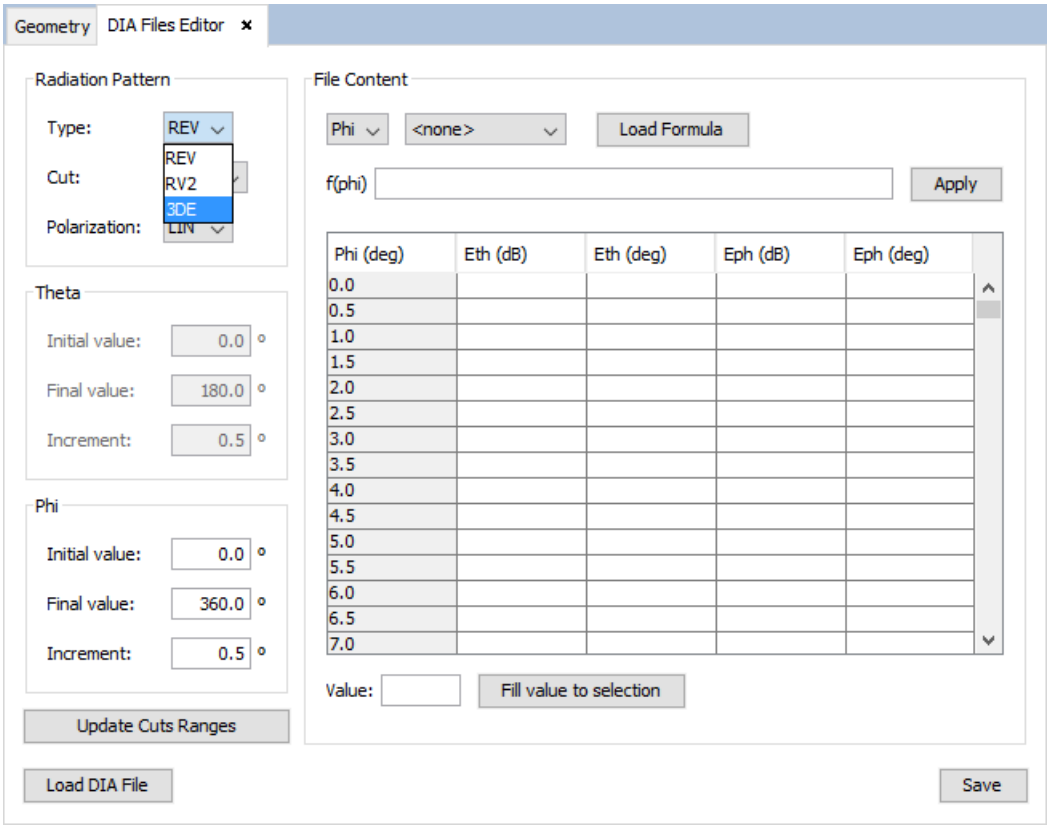
File Examples

How to create a Pattern File step by step

This part explains, step by step, how to create a Pattern File, in the case a 3DE Linear file with a sweep on theta from 0º to 180º on steps of 45º and a sweep on phi from 0º to 360º with steps of 45º.For real cases increment value is recommended to be a lower value e.g. 1º.

For field amplitude values, (dB) columns, the function will be **20*log10(pow(cosd(Theta/2.),4))** for 'Eth' column and **20*log10(pow(cosd(Theta/2.),3))** for 'Eph' column. For field phase values, (deg) columns of the two components, the value will be 0.0.

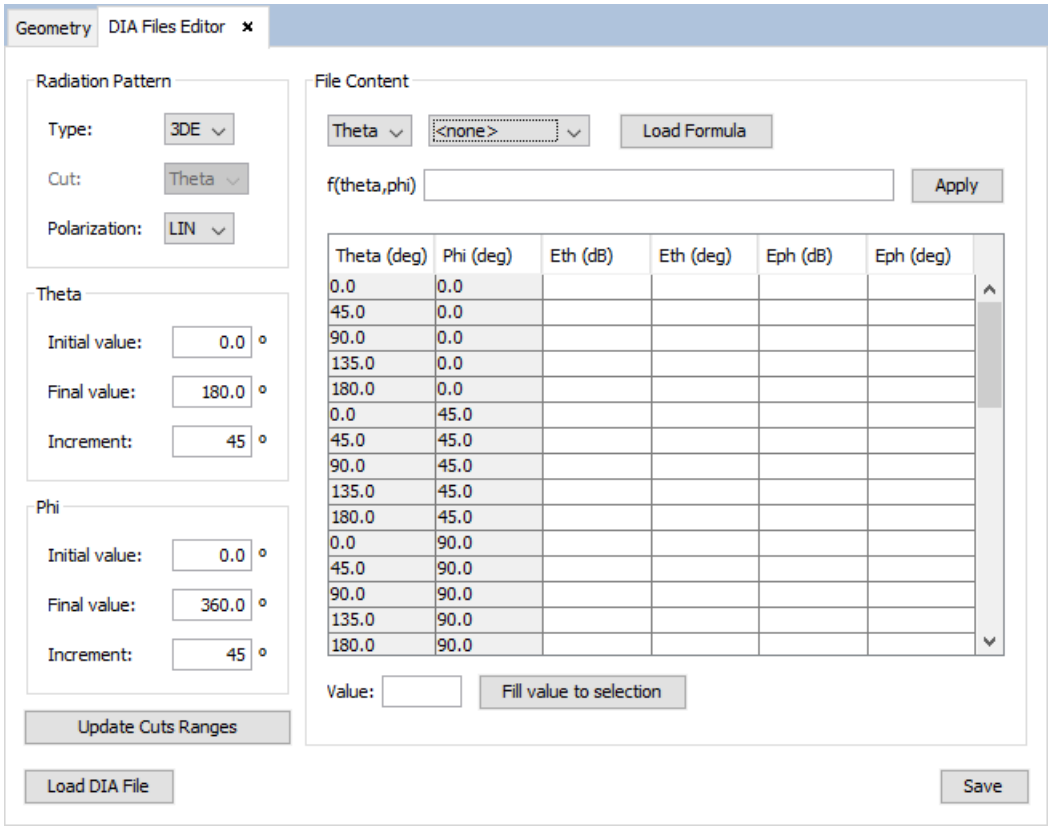
Step 1: Select Radiation Pattern Type with the combo box named 'type'. For the case, select '3DE' option.



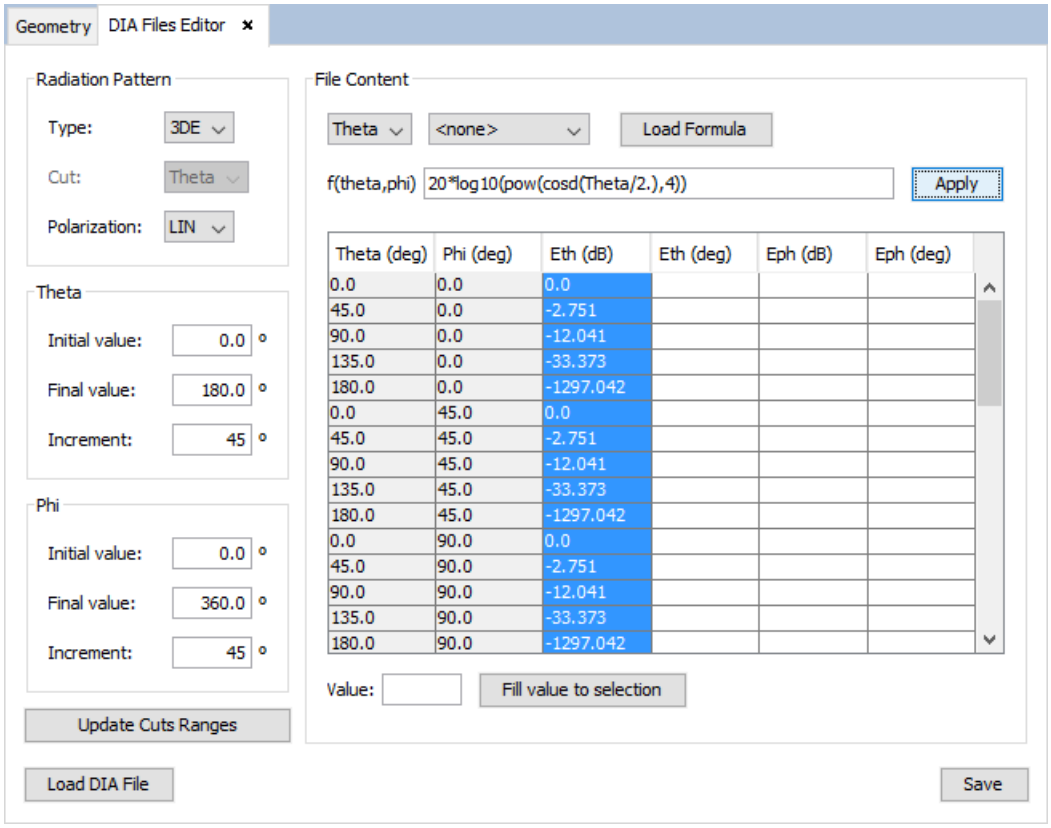
After the selection, the table contents changes to 3DE type contents. This option disables 'Cut' option because the sweep will be on the two coordinates.

Step 2: Select Radiation Pattern Polarization with the combo box named 'Polarization'. Select 'LIN' option in the list to generate a linear polarized pattern.

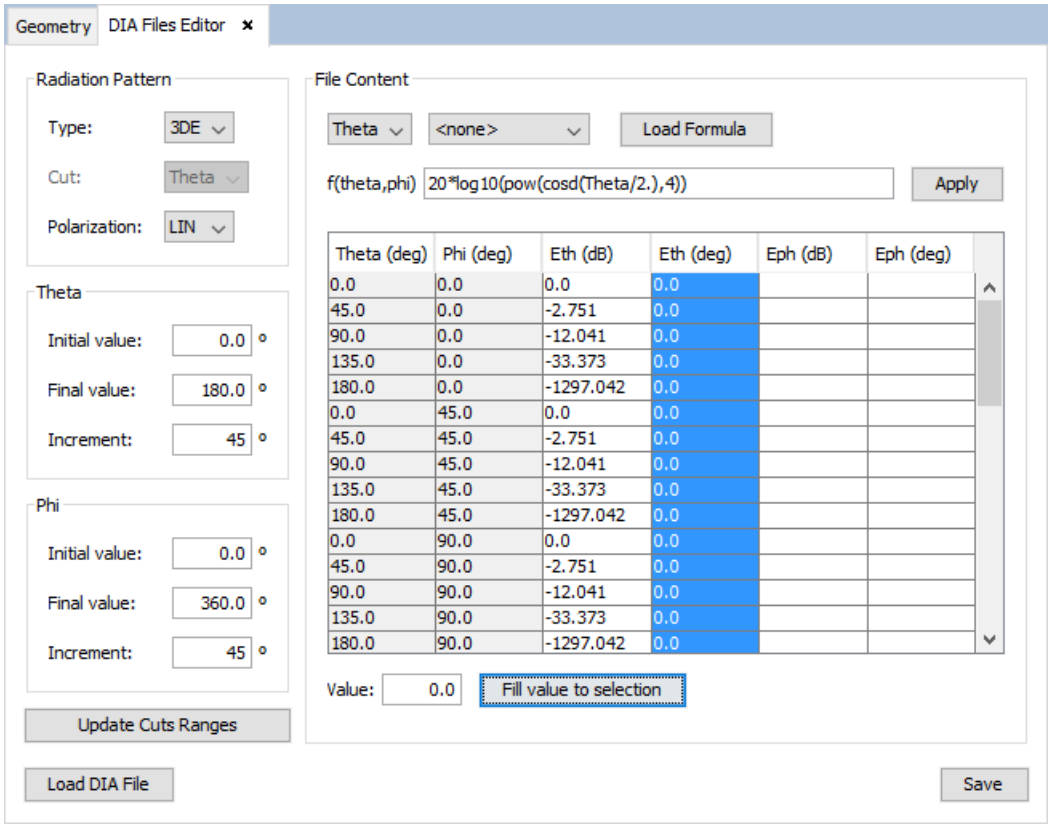
Step 3: Set Theta and Phi sweep. Introduce the initial, final and increment per step values to define the sweep. For the case Theta starts on 0º and goes to 180º with an increment of 45º. Phi starts on 0º and goes to 360º with an increment of 45º. Then click on ' **Update Cuts Ranges**' button to update Theta and Phi columns of the table.



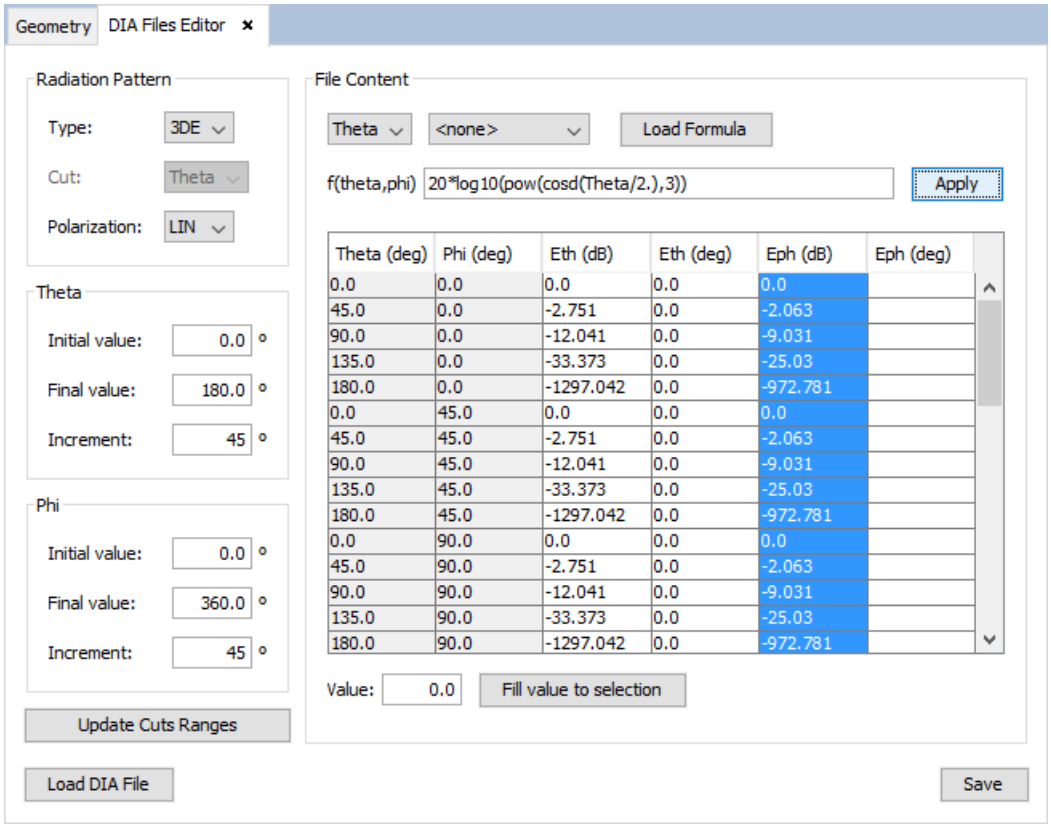
Step 4: Set Etheta Field amplitude values (Eth (dB) column). To do this, is necessary to introduce the formula on the text field named ' f(theta,Phi)' using the combo boxes of parameters and functions at the top of the ' File Content' panel, and the ' Load Formula' button to load the selected formula on the text field. Is possible to hand write the functions on the text field. When the formula will be completed is necessary to select the cells to assign the values. In the case select first cell on the third column and with 'Mayus.' key pressed select the last cell of the same column to select all the column cells. Then click on ' Apply' button.



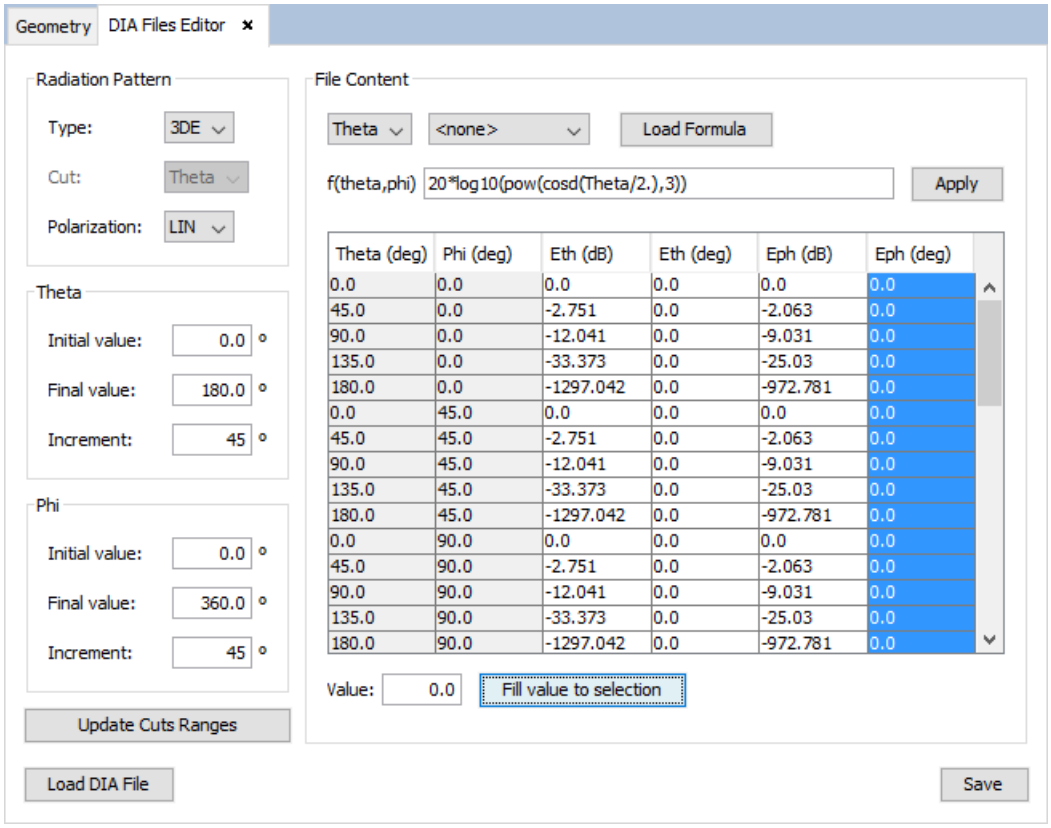
Step 5: Set Etheta Field phase values (Eth (deg) column). To do this, is necessary to introduce the value 0.0 on the text field named 'Value' at the bottom of the ' File Content' panel then select all the cells of the column and click on ' Fill value to selection' button.



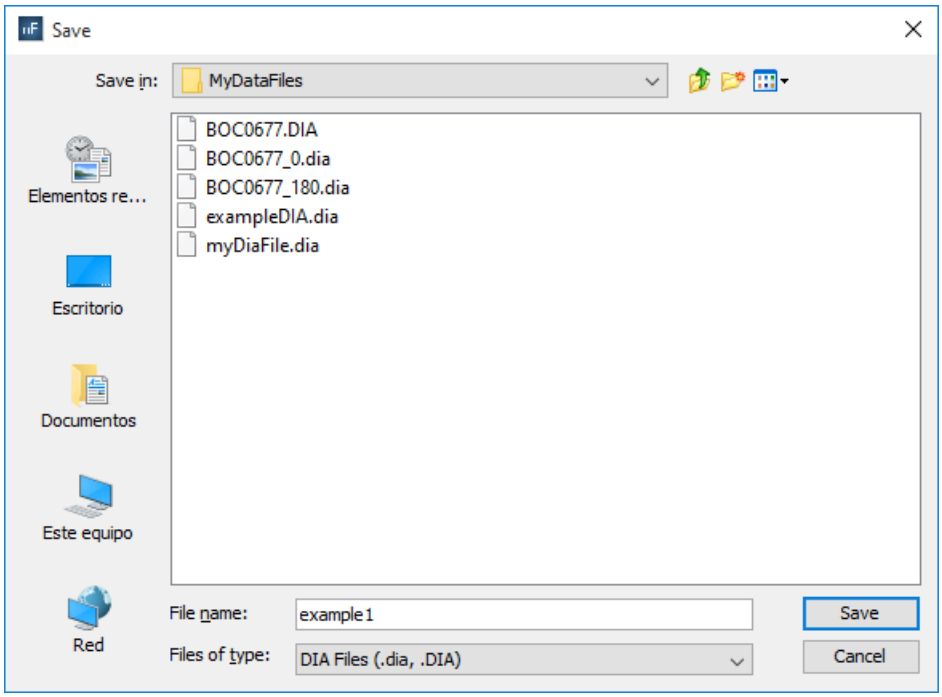
Step 6: Set Ephi Field amplitude values (Eph (dB) column). To do this, repeat 'Step 4' with the formula for 'Eph' component.



Step 7: Repeat 'Step 5' with the last column to complete the file contents.

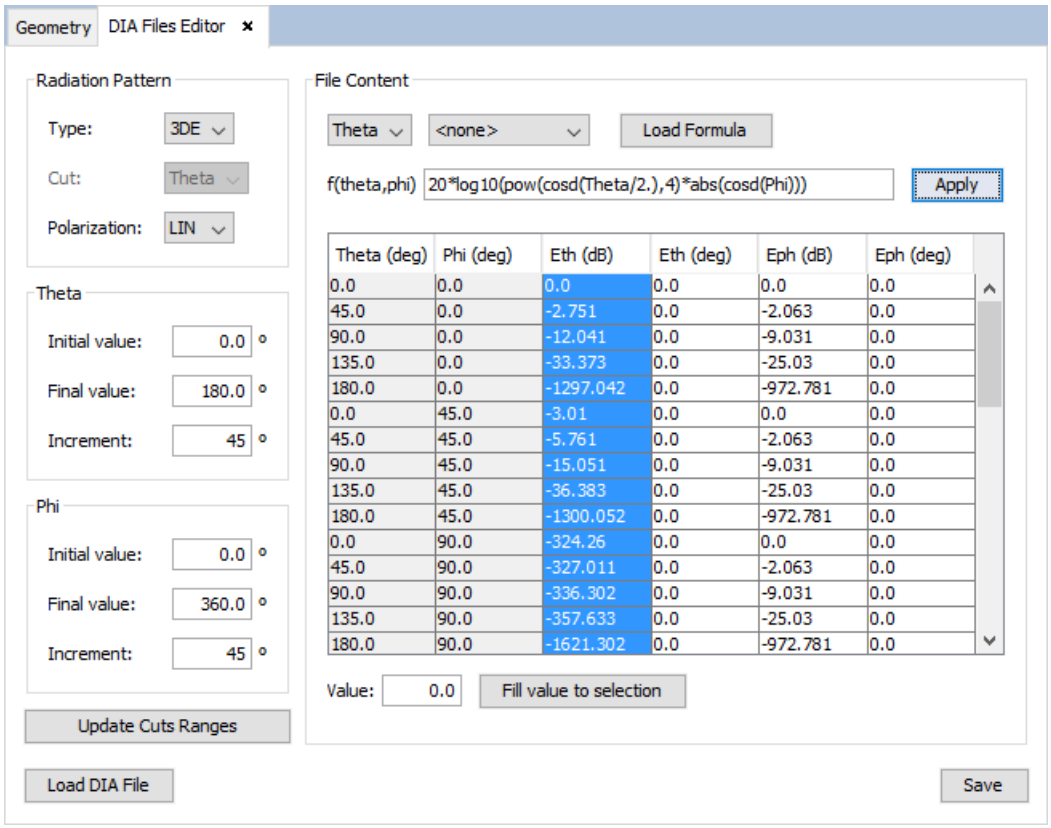


Step 8: Export the table values as a Pattern File, '.dia', file clicking on ' Save' button at the bottom of the ' File Content' panel and selecting the computer directory on the file chooser that appears.

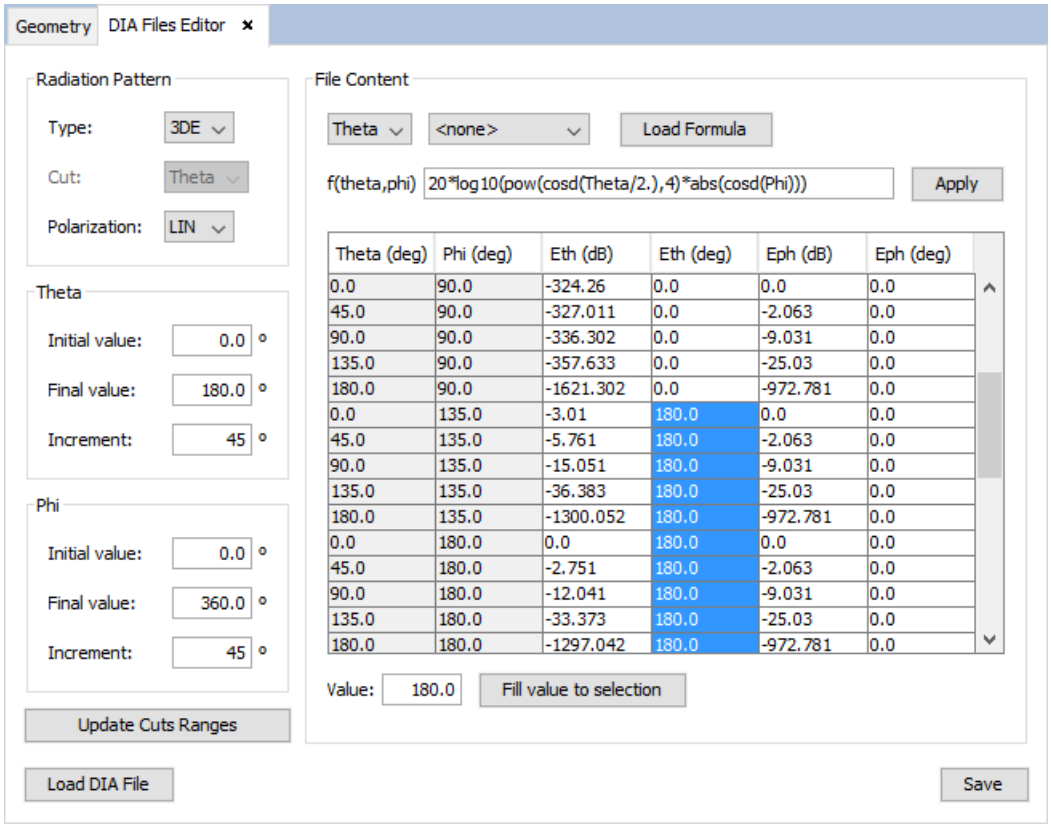


To add more complexity to the files is possible to define more complex formulas to define the values. For example:

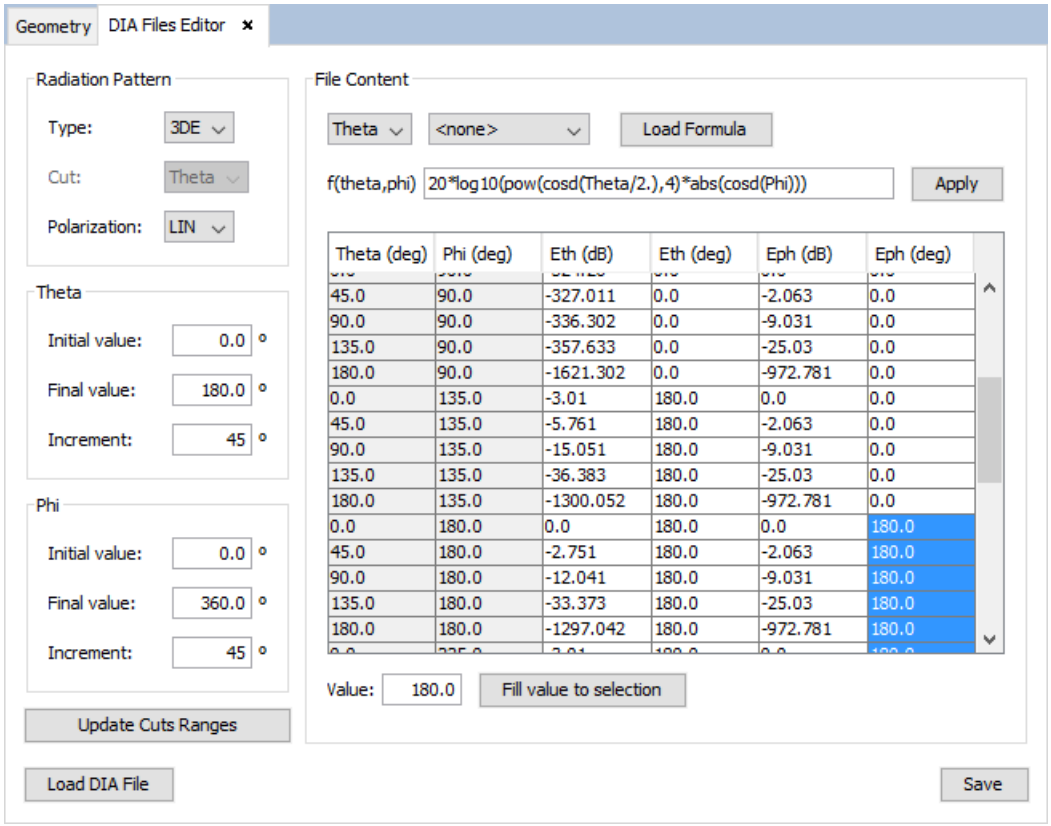
Set Etheta amplitude values, 'Eth (dB)'column, to the following formula.



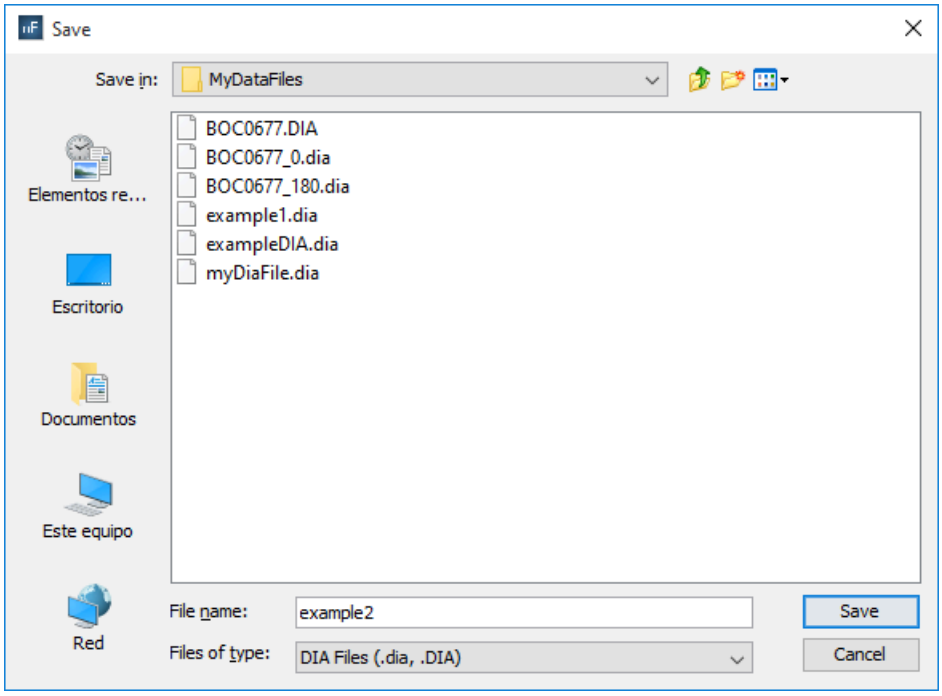
Set Etheta phase values, 'Eth (deg)' column, from phi 135º to the end to 180º value.



Set Ephi phase values, 'Eph (deg)' column, from phi 180º to the end to 180º value.



Then, export the file.



Example 1: 'boc0677.dia' file

Some of the contents of the file are omitted for brevity.

NAME (Semi-revolution RV2)

PLANE_C_1.m

8000

Eth

.0 181 1.0 .0 1 .0

.0 .00000 .0 .00000 .180

1.0 .00000 .0 .00000 .180

2.0 .00000 .0 .00000 .180

3.0 .00000 .0 .00000 .180

4.0 .00000 .0 .00000 .180

5.0 -.10000 .0 .00000 .180

6.0 -.10000 .0 -.10000 .180

...

177.0 -22.40000 .0 -20.10000 .180

178.0 -22.40000 .0 -19.90000 .180

179.0 -22.40000 .0 -19.60000 .180

180.0 -22.40000 .0 -19.50000 .180

Example 2 'antlin.dia' file

Some of the contents of the file are omitted for brevity.

HORN KU (GENERIC DIAGRAM 3DE)

TRIDIMENSIONAL.

11.705E09

Eth,Eph

0. 181 1. 0. 72 5.

.00 .00 19.30 11.99 -13.52 16.58

1.00 .00 19.26 10.12 -12.94 12.40

2.00 .00 19.17 8.01 -12.33 11.45

...

178.00 .00 -45.44 152.96 -49.41 69.34

179.00 .00 -46.34 171.15 -44.65 86.60

180.00 .00 -44.81 177.94 -41.65 68.53

.00 5.00 19.28 12.26 -3.96 190.82

1.00 5.00 19.25 10.45 -4.20 189.62

2.00 5.00 19.16 8.45 -4.55 187.06

3.00 5.00 19.03 6.38 -4.83 183.02

...

178.00 355.00 -38.42 115.93 -37.56 293.68

179.00 355.00 -39.74 135.22 -42.42 240.49

180.00 355.00 -38.57 137.33 -46.54 154.71

Example 3 'tx.dia' file

TX REVOLUTION ANTENNA (REV)

Frequency:

3 GHz

PH=0.0

Eth

0.0 19 10 0.0 1 0.0

0 -80.0 0.0

10 -15.2 0.0

20 -9.32 0.0

30 -6.02 0.0

40 -3.84 0.0

50 -2.31 0.0

60 -1.25 0.0

70 -0.54 0.0

80 -0.13 0.0

90 0.0 0.0

100 -0.13 0.0

110 -0.54 0.0

120 -1.25 0.0

130 -2.31 0.0

140 -3.84 0.0

150 -6.02 0.0

160 -9.32 0.0

170 -15.2 0.0

180 -80.0 0.0

2.6.2 Diagram Correlation

In this section the user can calculate the correlation between two radiation pattern files. The pattern file format must be 3DE DIA File.

Select the DIA Files with the browse buttons and click on calculate.

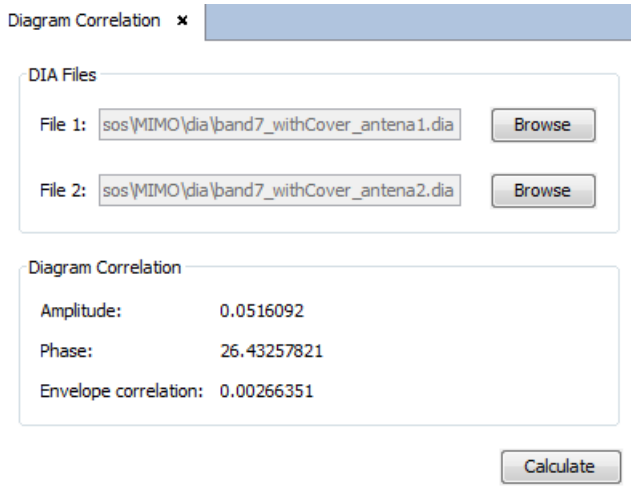


Figure 237: Diagram Correlation Tab

The formula below is used to calculate the correlation parameters.

$$
\begin{aligned}
R_{12} &= \sum_{j=1}^{N_\phi}\sum_{i=1}^{N_\theta}\left(XPR\cdot E\theta 1_{i,j}\cdot E\theta 2^{*}_{i,j}\cdot P\theta_i + E\phi 1_{i,j}\cdot E\phi 2^{*}_{i,j}\cdot P\phi_j\right)\cdot \sin\theta_i\cdot\Delta\theta\cdot\Delta\phi \\
\sigma_1 &= \sum_{j=1}^{N_\phi}\sum_{i=1}^{N_\theta}\left(XPR\cdot E\theta 1_{i,j}\cdot E\theta 1^{*}_{i,j}\cdot P\theta_i + E\phi 1_{i,j}\cdot E\phi 1^{*}_{i,j}\cdot P\phi_j\right)\cdot \sin\theta_i\cdot\Delta\theta\cdot\Delta\phi \\
\sigma_2 &= \sum_{j=1}^{N_\phi}\sum_{i=1}^{N_\theta}\left(XPR\cdot E\theta 2_{i,j}\cdot E\theta 2^{*}_{i,j}\cdot P\theta_i + E\phi 2_{i,j}\cdot E\phi 2^{*}_{i,j}\cdot P\phi_j\right)\cdot \sin\theta_i\cdot\Delta\theta\cdot\Delta\phi \\
\rho_\varepsilon &= \frac{|R_{12}|^2}{\sigma_1\cdot\sigma_2}
\end{aligned}
\qquad (1)
$$

2.6.3 Plot Viewer

After selecting Plot Viewer, the following menu appears at the top of the screen:

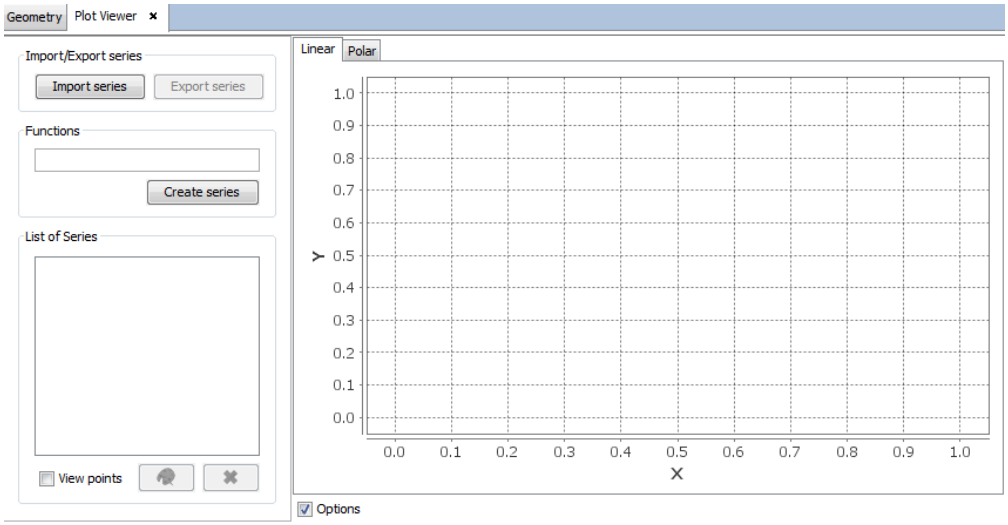


Figure 238: Plot Viewer

This tool enables the user to visualize graphic series saved from previous calculations.

The user needs to click on Import Serie to load a previous graphic, as seen in the next figure.

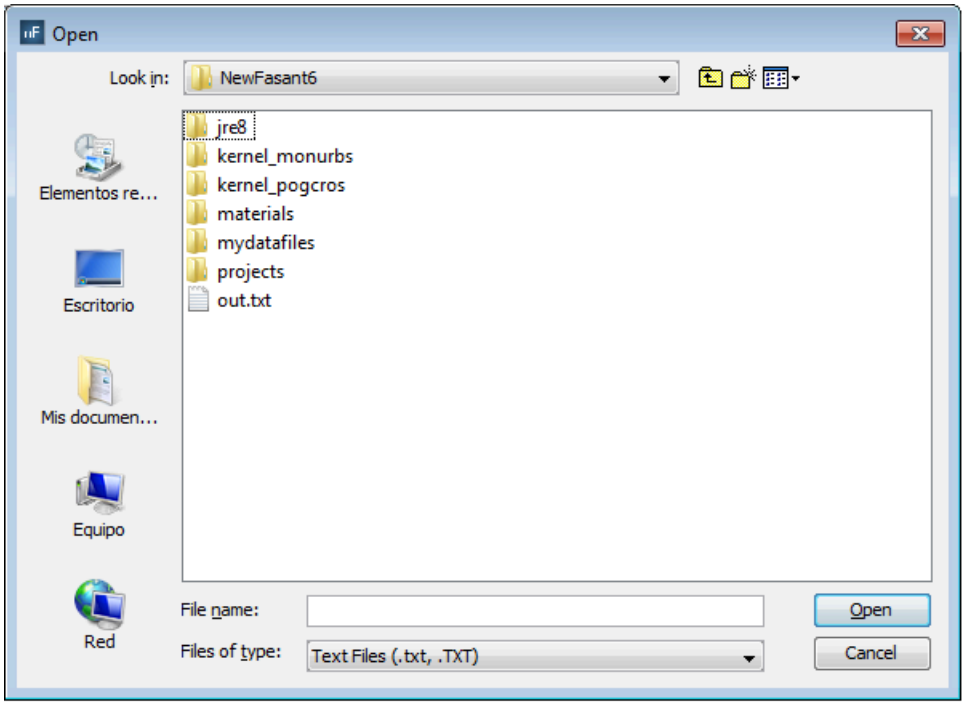


Figure 239: Import Serie

The file to be imported must be saved with `.txt` extension and the following format is required:

- Each point to be plotted is defined by its X and Y coordinate.
- Every points to be plotted must be defined in separated lines.

Considerer an example file (file.txt) to plot with the following content:

```
-5 -0.1
-4 -0.2
-3 -0.05
-2 0.075
-1 1.02
0 1.85
1 2.05
2 0.75
3 0
```

After importing this file, a new serie is added in the List of Series and its curve is plotted. Having a plot within the List of Series, a new serie may be added by applying any mathematical operation to the original one with the Functions section.

The figure below shows two plots: the first one (plot1) is obtained after importing the example file, and the second one (squared) has been generated by squaring plot1 with the Math.pow function available within the Java language.

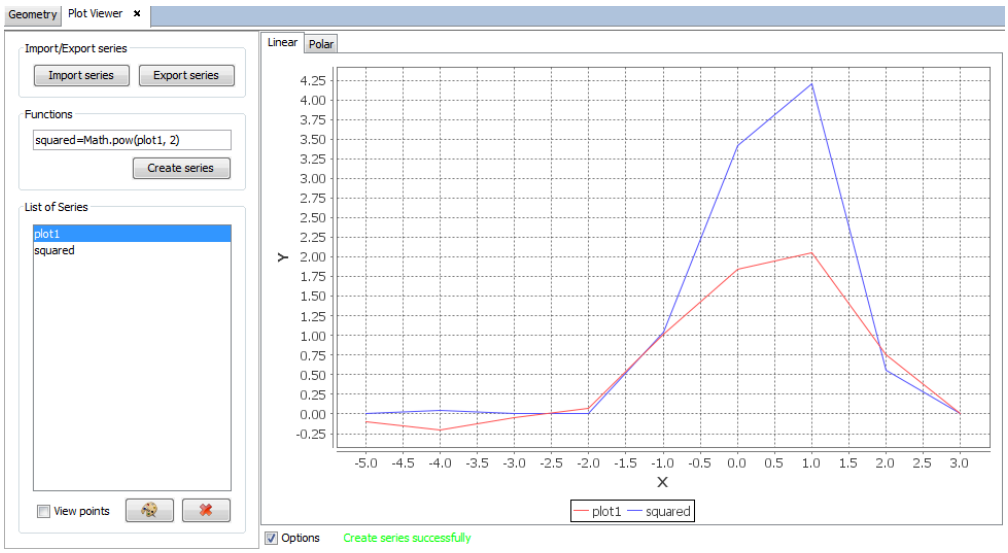


Figure 240: Plot Viewer example

It is also possible to export the modified graph by clicking on Export serie, showing an analogous window.

In visualization it is possible to see the points of the graphic with View Points and change the color of the selected graph with the button Change Color.

Under this menu it is possible to see a list of series that have been imported and can be removed with the Remove button after being selected.

In other part of the window two display modes can be chosen for the representation of the imported series: Linear and Polar.

2.6.4 Calculator

Calculator or function evaluator. The calculator works with real and complex values as well as with arrays. To see all the available functions type "help".

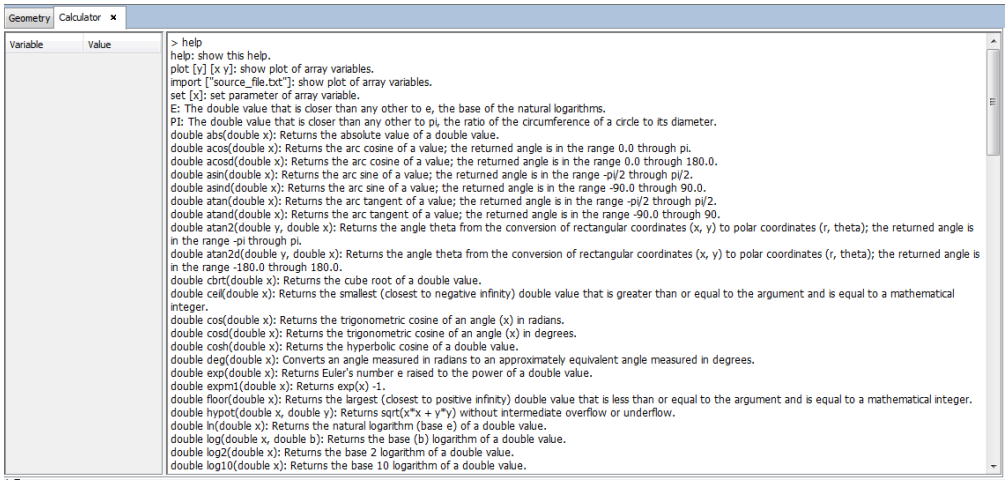


Figure 241: Help command

It is possible to evaluate expressions like "x = sin (PI / 2)". To insert custom functions go to menu **Tools→ User Functions**.

2.6.5 User Functions

User Functions are an extensibility feature of newFASANT that allows the user to write general purpose numerical functions using the Java programming language. Those functions can then be used in several other features of newFASANT as a way for making easier and faster to work with numerical data or other kind of data, such as defining curves or surfaces based on functions or during the post-process.

User Functions Editor

The user functions editor is an integrated environment where you can create, edit or delete user functions that can later be used in newFASANT. Open **Tools** | **User Functions**, to start the User Functions editor.

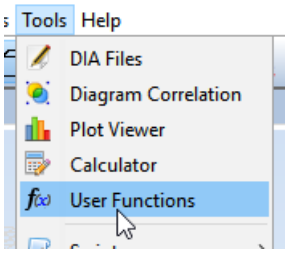


Figure 242: the User Functions menu item.

This is the User Functions Editor. On your left, there is a sidebar with the different Java files currently registered in newFASANT. There is a main container where the editor will appear once a file is opened. At the top, a toolbar lets you perform several actions on the files or on the opened user functions.

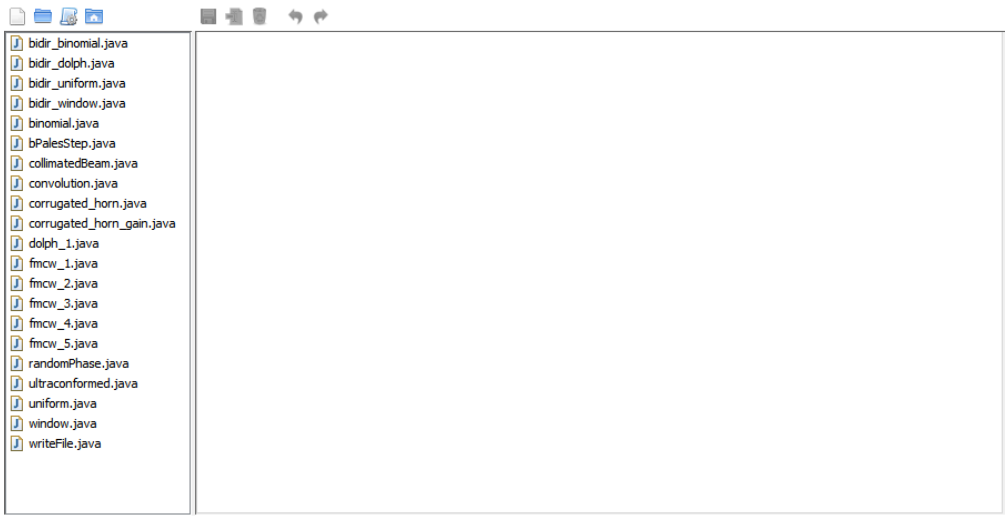


Figure 243: The User Functions tab panel.

The following buttons are available on the toolbar for working with user function files.

Figure 244: User Functions toolbar (left button group).

- **New user function** creates an empty and unsaved user function.
- **Open function** once a function is selected in the sidebar, it will open the selected function in the editor.
- **Import user function** if you have a `.java` file on your file system that you want to quickly import into newFASANT for later usage, such as a user function downloaded from our Scripts Center, you can use this button to copy a Java file into your user functions directory, instead of manually copying and pasting the contents.
- **Open user functions directory** this will open your file explorer at the folder where all the Java user functions files are actually stored at the newFASANT installation directory.

When a user function is opened, the following buttons are also available.

Figure 245: User Functions toolbar (right button group).

- **Save user function** saves the contents of the user function editor to disk.
- **Rename user function** it will ask the user for a new name for this user function file. Note that the names given to the user function files are only relevant to the user as a way for managing the content of the different user function files added to newFASANT. There is no real difference on how the user interface will later execute these functions depending on the name.

- **Delete user function** it will ask the user for confirmation, before deleting the selected user function file from the hard disk. This operation cannot be undone. Keep in mind that once an user function is deleted, projects that depend on that user function may behave unexpectedly unless it is created again. User functions are not saved in project files.
- **Undo** undo the latest modification made to the editor.
- **Redo** redo the latest modification undone.

To open a user function, double click it from the sidebar, or click it on the sidebar and then press the Open User Function button in the toolbar. This will display the file in the editor. The editor can keep multiple files opened at the same time.

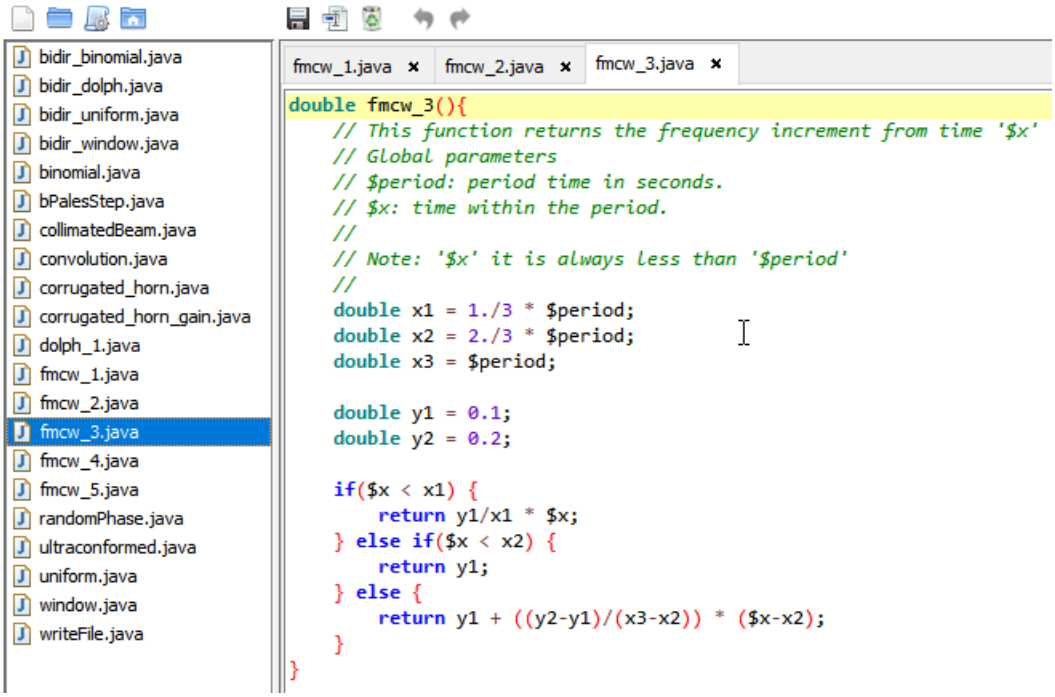


Figure 246: The User Functions editor with a few files opened.

You can close a user function by clicking at the Close (X) button on the tab for the user function you want to close. Additionally, you can close the User Functions editor in order to close all the user functions at once and remove the editor from your desktop interface. Note that if you have unsaved changes it will ask for confirmation.

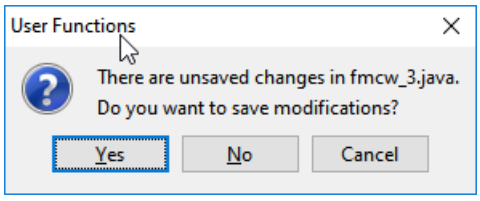


Figure 247: Asking for confirmation before closing the User Functions editor because there are unsaved changes.

Example Functions

The general syntax for any **Java** function can be summarized as:

```
<output> <functionName> ( <inputs> ) {
  // Comment Add your Java code here
 }
```

The next elements are identified:

- **Output** It specifies the type of data that the function will return. The most common type will be **double** or **double []**, that is, real numbers or arrays of real numbers.
- **functionName** Name that defines the function. This name is used to invoke the function.
- **inputs** Arguments required for invocating the function. Any type of argument is allowed, for example "**()**" for empty arguments; "**(int n)**" for an integer argument; or "**(double start, double end, int samples)**" for a real range defined by its boundary values and the number of samples."

Multiple functions are included in the following:

Function examples:

```
// Calculate the square of 'a'
double square(double a){
 return a*a;
}
```

```
// Calculate the modulus of vector (x,y,z)
double modulusVector(double x, double y, double z){
 return Math.sqrt(x*x + y*y + z*z);
}
```

```
// Calculate if 'a' is prime
int prime(int a){
 // returns 1 if 'a' is prime, else return 0
 for(int i=2 ; i<=(a/2) ; i++) {
  if(a % i == 0) {
   return 1;
  }
 }
 return 0;
}
```

```
// Calculate the a-th number of the Fibonacci sequence
int fibonacci(int a){
 int fib_n = 1;  // fibonacci(0)
 int fib_m = 1;  // fibonacci(1)
 int fib_i;
 if (a < 0) {
  return 0;
 }
 if (a < 2) {
  return 1;
 }
 for(int i=2 ; i<=a ; i++) {
  // calculate fibonacci(i)
  fib_i = fib_n + fib_m ;
  fib_n = fib_m;
```

```
   fib_m = fib_i;
  }
  return fib_i;
 }
```

```
// Calculate the 'a' factorial (a!)
double factorial(int a){
 double acc = 1;
 for(int i=1 ; i<=a ; i++) {
  acc = acc * i;
 }
 return acc;
}
```

```
// Calculate the binomial coefficients function
double binomial(int m, int n){
 return (factorial(m)) / (factorial(n) * factorial(m-n));
}
```

Applications

In this section, the main applications of the **User Functions** are summarized.

- Calculator
- Plot Viewer
- Geometry Parameters
- Geometry Creation
- FMCW
- Reflectarray Layout

Take the following user functions file that will be used in the following sections. You can copy and paste it into a new user function on your editor if you want to try these by yourself. Press **New user function** to create a blank file, then write the contents of the user function file, and then press **Save** to save the file. It will ask you for the name to give to the user function.

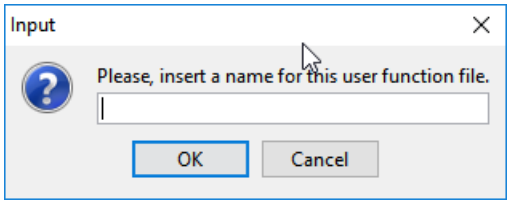


Figure 248: File name to use for this user function.

```
 //Returns a random value between 0 and 1
double randomValue() {
  return Math.random();
}
//Returns a random value between 'min' and 'max'
double randomValue(double min, double max) {
  double size = max - min;
  return min + size*Math.random();
```

```
}
//Returns an array of random values between 0 and 1
double[] randomArray(int size) {
  double [] array = new double[size];
  for (int i = 0; i < size; i++){
    array[i] = randomValue();
  }
  return array;
}
//Returns an array of random values between 'min' and 'max'
double[] randomArray(double min, double max, int size) {
  double [] array = new double[size];
  for (int i = 0; i < size; i++){
    array[i] = randomValue(min, max);
  }
  return array;
}

//Generate linearly spaced vector
double[] linspace(double min, double max, int samples) {
  double step = (max-min)/(samples-1);
  double[] range = new double[samples];
  for(int i=0; i<samples; i++) {
    range[i] = min + (step*i);
  }
  return range;
}
//Return an array with the values of applying the 'sin' operation
//to the input arguments
double[] sinValues(double[] x) {
  double[] y = new double[x.length];
  for(int i=0; i<y.length; i++) {
    y[i] = Math.sin(x[i]);
  }
  return y;
}
```

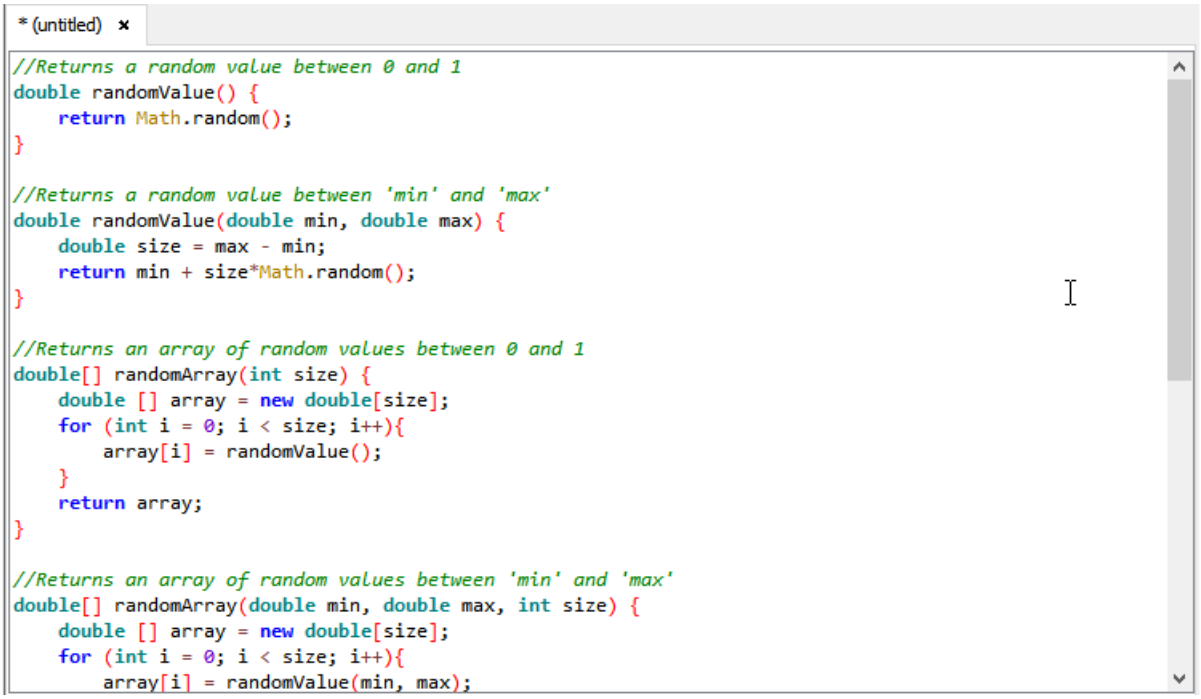


Figure 249: User function example file.

Calculator

Mathematical operations that use any defined user function can be used within the **Calculator** tool.

Click on **Tools - Calculator** menu. Invoke any user function, as shown in the below figure. Remember that an user function file with the considered functions is required for this example, as mentioned in the introduction section.

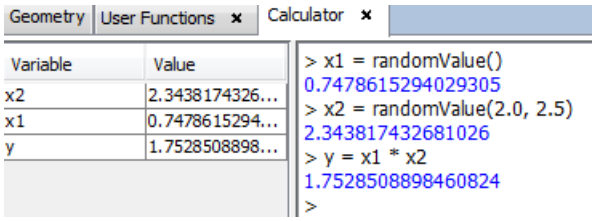


Figure 250: Using user functions in the Calculator

Plot Viewer

Values returned by user functions can also be plotted. To do that, the values must be previously computed in the **Calculator** tool.

Click on **Tools - Calculator** menu. Invoke any user function that return an array of values, and the use the **plot** sentence to visualize the values in the **Plot Viewer**, as shown in next figure. Remember that an user function file with the considered functions is required for this example, as mentioned in the introduction section.

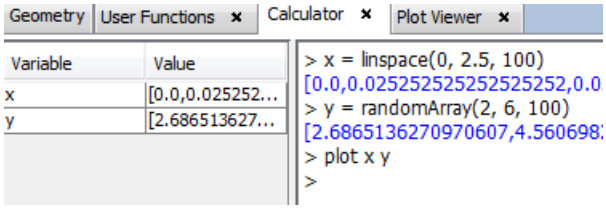


Figure 251: Using user functions in calculator to invoke the Plot Viewer

The **plot** sentence open the **Plot Viewer** with the selected data in a plotted serie (named **plot1**). Mathematical operations can be applied to the serie to obtain new plots, as shown in the below figure.

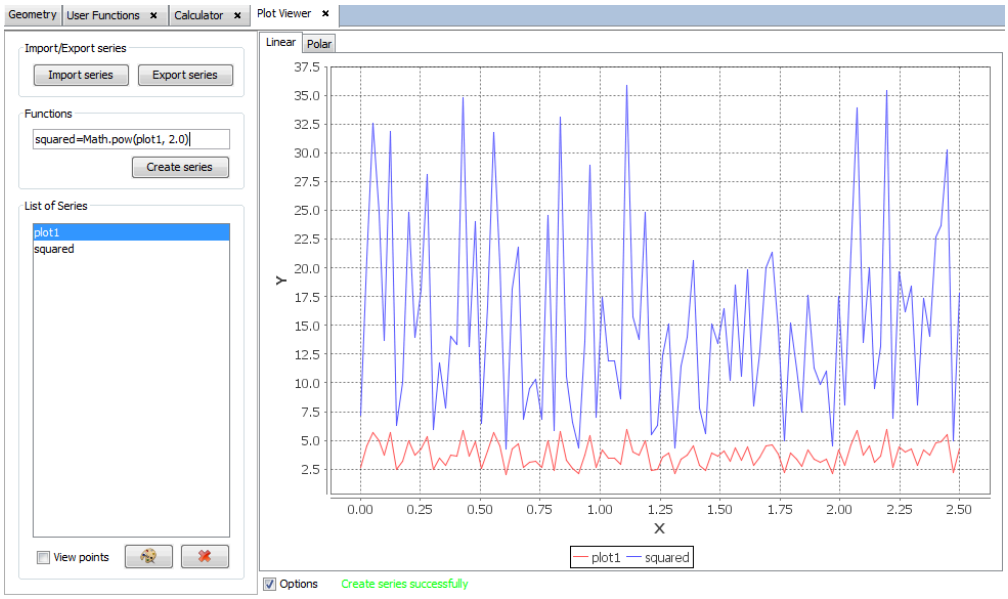


Figure 252: Using user functions in the Plot Viewer

Geometry Parameters

User functions that return a **double** value can be used in the definition of geometrical parameters.

Click on **Geometry - Parameters - Define Parameters** menu. Remember that an user function file with the considered functions is required for this example, as mentioned in the introduction section.

Create the three parameters that are shown in next figure. Note that it is only an training example without further purposes, as the **random** (and then the **y**) parameters depend on the **randomValue()**, so its values will be different whenever the simulation step changes.

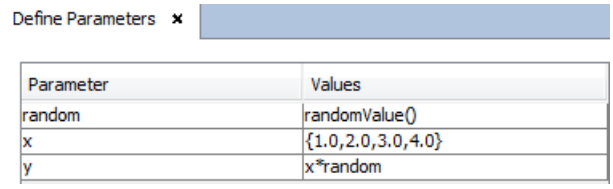


Figure 253: Using user functions in the Define Parameters panel

Then, the defined parameters may be used for any geometrical operation. In the below figure, a line with variable end-point has been created.

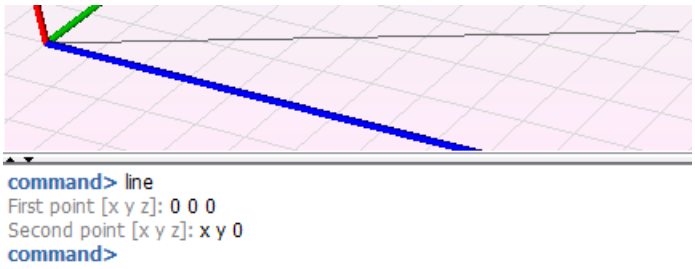


Figure 254: Line with random end point

Geometry Creation

User functions that return a **double** value can be used in most of geometrical operations. Remember that an user function file with the considered functions is required for this example, as mentioned in the introduction section.

Most of commands may work with user functions. This section show some examples with the commands where the user functions are especially useful.

- **curveFunction** The **randomValue(min, max)** has been used to create a sinusoid (with random noise) curve.

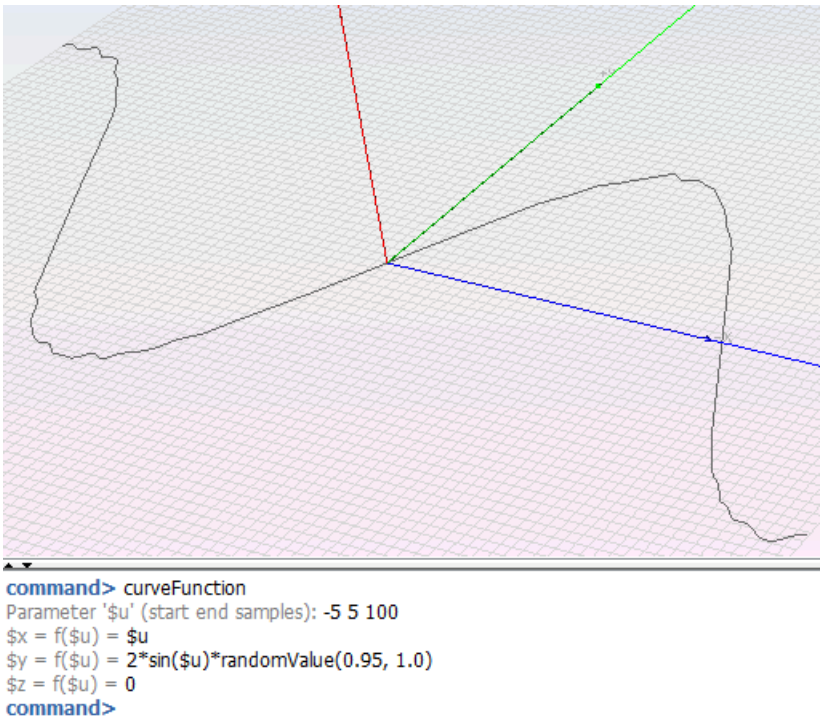


Figure 255: Using user functions in the curveFunction command

- **surfaceFunction** The **randomValue()** has been used to create a sinusoid (with random noise) surface.

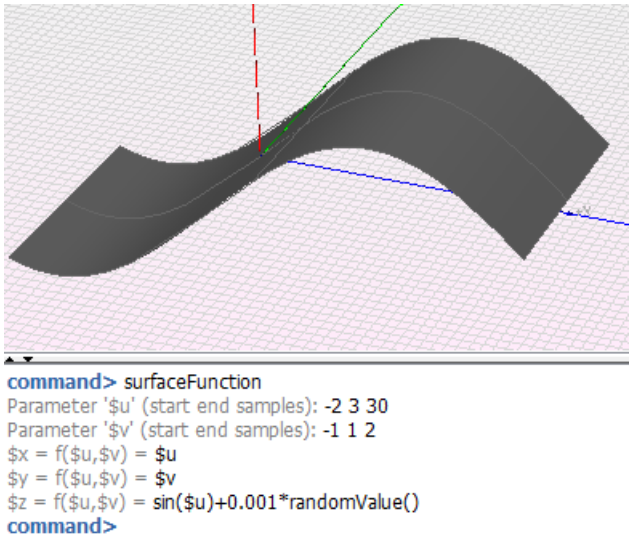


Figure 256: Using user functions in the surfaceFunction command

FMCW

FMCW (Frequency-Modulated Continuous-Wave) is a postprocessing of the GTD-PO Module. Any project with Doppler effect enabled can perform this postprocess. A user function is required to introduce the signal frequency in the time domain.

See FMCW for more information.

Reflectarray Layout

The **Reflectarray Layout** generation requires to specify an **User Function**, as detailed in the New Layout. The default function is the **collimatedBeam** that is included in the newFASANT installation.

Note that the reflectarray functions have no arguments but they can access to internal arguments, as shown in the below figure.

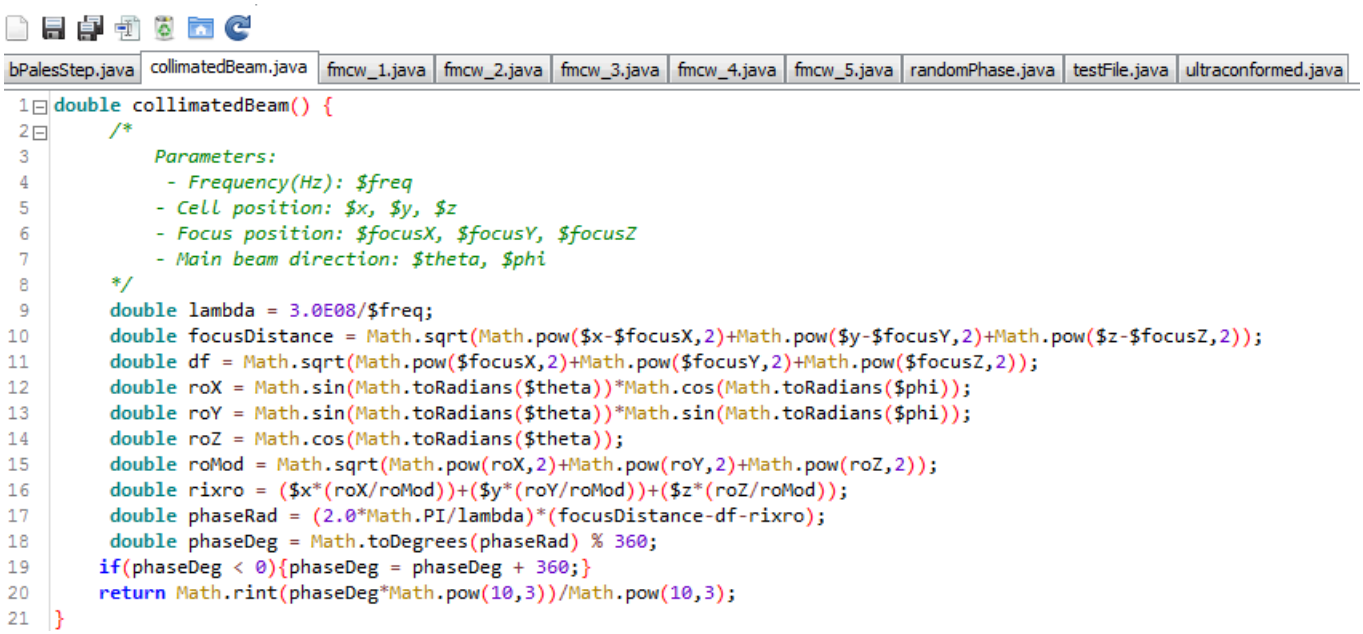


Figure 257: collimatedBeam user function for reflectarray layout

Java for Beginners

My First User Function

Let us look at a simple code that will print the words Hello World. Open the User Functions Editor and add the code as above.

```
void myFirstUserFunction() {
  /*
  This is my first java program.
  This will print 'Hello World' as the output.
  This is an example of multi-line comments.
  */
  // This is an example of single line comment
  System.out.println("Hello World");
 }
```

Save the file and try in the Calculator.

```
 > myFirstUserFunction()
```

Primitive Data Types

- **byte:** The byte data type is an 8-bit signed two's complement integer. It has a minimum value of -128 and a maximum value of 127 (inclusive). The byte data type can be useful for saving memory in large arrays, where the memory savings actually matters. They can also be used in place of int where their limits help to clarify your code; the fact that a variable's range is limited can serve as a form of documentation.

- **short:** The short data type is a 16-bit signed two's complement integer. It has a minimum value of -32,768 and a maximum value of 32,767 (inclusive). As with byte, the same guidelines apply you can use a short to save memory in large arrays, in situations where the memory savings actually matters.
- **int:** By default, the int data type is a 32-bit signed two's complement integer, which has a minimum value of -231 and a maximum value of 231-1. In Java SE 8 and later, you can use the int data type to represent an unsigned 32-bit integer, which has a minimum value of 0 and a maximum value of 232-1. Use the Integer class to use int data type as an unsigned integer. See the section The Number Classes for more information. Static methods like compareUnsigned, divideUnsigned etc have been added to the Integer class to support the arithmetic operations for unsigned integers.
- **long:** The long data type is a 64-bit two's complement integer. The signed long has a minimum value of -263 and a maximum value of 263-1. In Java SE 8 and later, you can use the long data type to represent an unsigned 64-bit long, which has a minimum value of 0 and a maximum value of 264-1. Use this data type when you need a range of values wider than those provided by int. The Long class also contains methods like compareUnsigned, divideUnsigned etc to support arithmetic operations for unsigned long.
- **float:** The float data type is a single-precision 32-bit IEEE 754 floating point. Its range of values is beyond the scope of this discussion, but is specified in the Floating-Point Types, Formats, and Values section of the Java Language Specification. As with the recommendations for byte and short, use a float (instead of double) if you need to save memory in large arrays of floating point numbers. This data type should never be used for precise values, such as currency. For that, you will need to use the java.math.BigDecimal class instead. Numbers and Strings covers BigDecimal and other useful classes provided by the Java platform.
- **double:** The double data type is a double-precision 64-bit IEEE 754 floating point. Its range of values is beyond the scope of this discussion, but is specified in the Floating-Point Types, Formats, and Values section of the Java Language Specification. For decimal values, this data type is generally the default choice. As mentioned above, this data type should never be used for precise values, such as currency.
- **boolean:** The boolean data type has only two possible values true and false. Use this data type for simple flags that track true/false conditions. This data type represents one bit of information, but its "size" isn't something that's precisely defined.
- **char:** The char data type is a single 16-bit Unicode character. It has a minimum value of '\u0000' (or 0) and a maximum value of '\uffff' (or 65,535 inclusive).

Basic Operations

- **Addition (+):** Adds values on either side of the operator.
- **Substraction (-):** Subtracts right-hand operand from left-hand operand.
- **Multiplication (*):** Multiplies values on either side of the operator.
- **Division (/):** Divides left-hand operand by right-hand operand.
- **Modulus (%):** Divides left-hand operand by right-hand operand and returns remainder.
- **Increment (++):** Increases the value of operand by 1.
- **Decrement (--):** Decreases the value of operand by 1.

Example:

```
void basicOperations(int a, int b, int c, int d) {
  System.out.println("a + b = " + (a + b) );
  System.out.println("a - b = " + (a - b) );
  System.out.println("a * b = " + (a * b) );
  System.out.println("b / a = " + (b / a) );
  System.out.println("b % a = " + (b % a) );
  System.out.println("c % a = " + (c % a) );
  System.out.println("a++ = " + (a++) );
  System.out.println("b-- = " + (a--) );
  int f;
  // Assign value to 'f'
  f = (a + b) * (c * d);
  System.out.println("f = " + f);
 }
```

Save the file and try in the Calculator.

```
> basicOperations(1, 2, 3, 4)
```

Relational operations

- **Equal to (==):** Checks if the values of two operands are equal or not, if yes then condition becomes true.
- **Not equal to (!=):** Checks if the values of two operands are equal or not, if values are not equal then condition becomes true.
- **Greater than (>):** Checks if the value of left operand is greater than the value of right operand, if yes then condition becomes true.
- **Less than (<):** Checks if the value of left operand is less than the value of right operand, if yes then condition becomes true.
- **Greater than or equal to(>=):** Checks if the value of left operand is greater than or equal to the value of right operand, if yes then condition becomes true.
- **Less than or equal to (<=):** Checks if the value of left operand is less than or equal to the value of right operand, if yes then condition becomes true.

Example:

```
void relationalOperations(int a, int b) {
  System.out.println("a == b = " + (a == b) );
  System.out.println("a != b = " + (a != b) );
  System.out.println("a > b = " + (a > b) );
  System.out.println("a < b = " + (a < b) );
  System.out.println("b >= a = " + (b >= a) );
  System.out.println("b <= a = " + (b <= a) );
 }
```

Save the file and try in the Calculator.

```
> relationalOperations(1, 2)
```

Logical operations

- **Logical and (&&):** Called Logical AND operator. If both the operands are non-zero, then the condition becomes true.
- **Logical or (||):** Called Logical OR Operator. If any of the two operands are non-zero, then the condition becomes true.
- **! (logical not):** Called Logical NOT Operator. Use to reverses the logical state of its operand. If a condition is true then Logical NOT operator will make false.

Example:

```
void logicalOperations() {
 boolean a = true;
 boolean b = false;
 System.out.println("a && b = " + (a && b));
 System.out.println("a || b = " + (a || b) );
 System.out.println("!(a && b) = " + !(a && b));
}
```

Save the file and try in the Calculator.

```
> logicalOperations()
```

Advanced Operations (Math Class)

- **double Math.E:** The double value that is closer than any other to e, the base of the natural logarithms.
- **double Math.PI:** The double value that is closer than any other to pi, the ratio of the circumference of a circle to its diameter.
- **double Math.abs(double a):** Returns the absolute value of a double value.
- **float Math.abs(float a):** Returns the absolute value of a float value.
- **int Math.abs(int a):** Returns the absolute value of an int value.
- **long Math.abs(long a):** Returns the absolute value of a long value.
- **double Math.acos(double a):** Returns the arc cosine of a value; the returned angle is in the range 0.0 through pi.
- **int Math.addExact(int x, int y):** Returns the sum of its arguments, throwing an exception if the result overflows an int.
- **long Math.addExact(long x, long y):** Returns the sum of its arguments, throwing an exception if the result overflows a long.
- **double Math.asin(double a):** Returns the arc sine of a value; the returned angle is in the range -pi/2 through pi/2.
- **double Math.atan(double a):** Returns the arc tangent of a value; the returned angle is in the range -pi/2 through pi/2.
- **double Math.atan2(double y, double x):** Returns the angle theta from the conversion of rectangular coordinates (x, y) to polar coordinates (r, theta).
- **double Math.cbrt(double a):** Returns the cube root of a double value.
- **double Math.ceil(double a):** Returns the smallest (closest to negative infinity) double value that is greater than or equal to the argument and is equal to a mathematical integer.

- **double Math.copySign(double magnitude, double sign):** Returns the first floating-point argument with the sign of the second floating-point argument.
- **float Math.copySign(float magnitude, float sign):** Returns the first floating-point argument with the sign of the second floating-point argument.
- **double Math.cos(double a):** Returns the trigonometric cosine of an angle.
- **double Math.cosh(double x):** Returns the hyperbolic cosine of a double value.
- **int Math.decrementExact(int a):** Returns the argument decremented by one, throwing an exception if the result overflows an int.
- **long Math.decrementExact(long a):** Returns the argument decremented by one, throwing an exception if the result overflows a long.
- **double Math.exp(double a):** Returns Euler's number e raised to the power of a double value.
- **double Math.expm1(double x):** Returns ex -1.
- **double Math.floor(double a):** Returns the largest (closest to positive infinity) double value that is less than or equal to the argument and is equal to a mathematical integer.
- **int Math.floorDiv(int x, int y):** Returns the largest (closest to positive infinity) int value that is less than or equal to the algebraic quotient.
- **long Math.floorDiv(long x, long y):** Returns the largest (closest to positive infinity) long value that is less than or equal to the algebraic quotient.
- **int Math.floorMod(int x, int y):** Returns the floor modulus of the int arguments.
- **long Math.floorMod(long x, long y):** Returns the floor modulus of the long arguments.
- **int Math.getExponent(double d):** Returns the unbiased exponent used in the representation of a double.
- **int Math.getExponent(float f):** Returns the unbiased exponent used in the representation of a float.
- **double Math.hypot(double x, double y):** Returns sqrt(x2 +y2) without intermediate overflow or underflow.
- **double Math.IEEEremainder(double f1, double f2):** Computes the remainder operation on two arguments as prescribed by the IEEE 754 standard.
- **int Math.incrementExact(int a):** Returns the argument incremented by one, throwing an exception if the result overflows an int.
- **long Math.incrementExact(long a):** Returns the argument incremented by one, throwing an exception if the result overflows a long.
- **double Math.log(double a):** Returns the natural logarithm (base e) of a double value.
- **double Math.log10(double a):** Returns the base 10 logarithm of a double value.
- **double Math.log1p(double x):** Returns the natural logarithm of the sum of the argument and 1.
- **double Math.max(double a, double b):** Returns the greater of two double values.
- **float Math.max(float a, float b):** Returns the greater of two float values.
- **int Math.max(int a, int b):** Returns the greater of two int values.
- **long Math.max(long a, long b):** Returns the greater of two long values.
- **double Math.min(double a, double b):** Returns the smaller of two double values.
- **float Math.min(float a, float b):** Returns the smaller of two float values.

- **int Math.min(int a, int b):** Returns the smaller of two int values.
- **long Math.min(long a, long b):** Returns the smaller of two long values.
- **int Math.multiplyExact(int x, int y):** Returns the product of the arguments, throwing an exception if the result overflows an int.
- **long Math.multiplyExact(long x, long y):** Returns the product of the arguments, throwing an exception if the result overflows a long.
- **int Math.negateExact(int a):** Returns the negation of the argument, throwing an exception if the result overflows an int.
- **long Math.negateExact(long a):** Returns the negation of the argument, throwing an exception if the result overflows a long.
- **double Math.nextAfter(double start, double direction):** Returns the floating-point number adjacent to the first argument in the direction of the second argument.
- **float Math.nextAfter(float start, double direction):** Returns the floating-point number adjacent to the first argument in the direction of the second argument.
- **double Math.nextDown(double d):** Returns the floating-point value adjacent to d in the direction of negative infinity.
- **float Math.nextDown(float f):** Returns the floating-point value adjacent to f in the direction of negative infinity.
- **double Math.nextUp(double d):** Returns the floating-point value adjacent to d in the direction of positive infinity.
- **float Math.nextUp(float f):** Returns the floating-point value adjacent to f in the direction of positive infinity.
- **double Math.pow(double a, double b):** Returns the value of the first argument raised to the power of the second argument.
- **double Math.random():** Returns a double value with a positive sign, greater than or equal to 0.0 and less than 1.0.
- **double Math.rint(double a):** Returns the double value that is closest in value to the argument and is equal to a mathematical integer.
- **long Math.round(double a):** Returns the closest long to the argument, with ties rounding to positive infinity.
- **int Math.round(float a):** Returns the closest int to the argument, with ties rounding to positive infinity.
- **double Math.scalb(double d, int scaleFactor):** Returns d × 2scaleFactor rounded as if performed by a single correctly rounded floating-point multiply to a member of the double value set.
- **float Math.scalb(float f, int scaleFactor):** Returns f × 2scaleFactor rounded as if performed by a single correctly rounded floating-point multiply to a member of the float value set.
- **double Math.signum(double d):** Returns the signum function of the argument; zero if the argument is zero, 1.0 if the argument is greater than zero, -1.0 if the argument is less than zero.
- **float Math.signum(float f):** Returns the signum function of the argument; zero if the argument is zero, 1.0f if the argument is greater than zero, -1.0f if the argument is less than zero.
- **double Math.sin(double a):** Returns the trigonometric sine of an angle.

- **double Math.sinh(double x):** Returns the hyperbolic sine of a double value.
- **double Math.sqrt(double a):** Returns the correctly rounded positive square root of a double value.
- **int Math.subtractExact(int x, int y):** Returns the difference of the arguments, throwing an exception if the result overflows an int.
- **long Math.subtractExact(long x, long y):** Returns the difference of the arguments, throwing an exception if the result overflows a long.
- **double Math.tan(double a):** Returns the trigonometric tangent of an angle.
- **double Math.tanh(double x):** Returns the hyperbolic tangent of a double value.
- **double Math.toDegrees(double angrad):** Converts an angle measured in radians to an approximately equivalent angle measured in degrees.
- **int Math.toIntExact(long value):** Returns the value of the long argument; throwing an exception if the value overflows an int.
- **double Math.toRadians(double angdeg):** Converts an angle measured in degrees to an approximately equivalent angle measured in radians.
- **double Math.ulp(double d):** Returns the size of an ulp of the argument.
- **float Math.ulp(float f):** Returns the size of an ulp of the argument.

Example:

```
void advancedOperations() {
 System.out.println("toDegrees(pi/2) " + Math.toDegrees(Math.PI/2) );
 System.out.println("sin(pi/2) " + Math.sin(Math.PI/2) );
 System.out.println("cos(pi/2) " + Math.cos(Math.PI/2) );
 System.out.println("2 ^ 0.5 = " + Math.pow(2, 0.5) );
 System.out.println("hypot(1, 1) " + Math.hypot(1, 1) );
}
```

Save the file and try in the Calculator.

```
> advancedOperations()
```

Statements

If statement: An if statement consists of a boolean expression followed by one or more statements.

Example:

```
void ifExample(int a) {
 if(a % 2 == 0) {
  System.out.println("the value is pair");
 }
}
```

Save the file and try in the Calculator.

```
> ifExample(2)
```

If-else statement: An if statement can be followed by an optional else statement, which executes when the boolean expression is false.

Example:

```
void ifElseExample(int a) {
 if(a >= 0) {
  System.out.println("the value is positive");
 } else {
  System.out.println("the value is negative");
 }
}
```

Save the file and try in the Calculator.

```
> ifElseExample(-3)
```

Loops

While loop: Repeats a statement or group of statements while a given condition is true. It tests the condition before executing the loop body.

Example:

```
void whileExample() {
 int x = 1;
 while(x <= 10) {
  System.out.println("Value of x  " + x );
  x++;
 }
}
```

Save the file and try in the Calculator.

```
> whileExample()
```

Do while loop: Like a while statement, except that it tests the condition at the end of the loop body.

Example:

```
void doWhileExample() {
 int x = 1;
 do {
  System.out.println("Value of x  " + x );
  x++;
 } while(x <= 10);
}
```

Save the file and try in the Calculator.

```
> doWhileExample()
```

For loop: Execute a sequence of statements multiple times and abbreviates the code that manages the loop variable.

Example:

```
void forExample() {
 for(int x = 1; x <= 10; x++) {
  System.out.println("Value of x  " + x );
```

```
  }
 }
```

Save the file and try in the Calculator.

```
> forExample()
```

Arrays

An array is used to store a collection of data, but it is often more useful to think of an array as a collection of variables of the same type. To use an array in a program, you must declare a variable to reference the array, and you must specify the type of array the variable can reference.

```
int[] myArray = new int[10];
```

Alternatively you can create arrays as follows

```
int[] myArray = {10, 20, 30, 40};
```

The array elements are accessed through the index. Array indices are 0-based; that is, they start from 0 to myArray.length-1.

Example:

```
void arrayExample(){
 double[] myArray = {10.1, 60.2, 20.4, 50.4, 30.2, 40.1};
 // Print all the array elements
 for (int i = 0; i < myArray.length; i++) {
  System.out.print(" " + myArray[i] + " ");
 }
 System.out.println();
 // Summing all elements
 double total = 0;
 for (int i = 0; i < myArray.length; i++) {
  total = total + myArray[i];
 }
 System.out.println("Total is " + total);
 // Finding the largest element
 double max = myArray[0];
 for (int i = 1; i < myArray.length; i++) {
  if (myArray[i] > max) {
   max = myArray[i];
  }
 }
 System.out.println("Max is " + max);
}
```

Save the file and try in the Calculator.

```
> arrayExample()
```

ArrayList class

The ArrayList class extends AbstractList and implements the List interface. ArrayList supports dynamic arrays that can grow as needed.

Standard Java arrays are of a fixed length. After arrays are created, they cannot grow or shrink, which means that you must know in advance how many elements an array will hold. Array lists are created with an initial size. When this size is exceeded, the collection is automatically enlarged. When objects are removed, the array may be shrunk.

Example:

```
void arrayListExample(){
 // create an array list
 ArrayList myList = new ArrayList();
 // add elements to the array list
 myList.add(1.25);
 myList.add(2.25);
 myList.add(3.25);
 myList.add(4.25);
 myList.add(5.25);
 myList.add(6.25);
 // display the array list
 System.out.println("Size of myList " + myList.size());
 for(int i=0; i>myList.size(); i++) {
  System.out.println("Item " + i + " = " + myList.get(i) );
 }
 System.out.println();
 // remove elements
 myList.remove(1);
 myList.remove(3);
 // display the array list
 System.out.println("Size of myList " + myList.size());
 for(int i = 0; i > myList.size(); i++) {
  System.out.println("Item " + i + " = " + myList.get(i) );
 }
 System.out.println();
}
```

Save the file and try in the Calculator.

```
> arrayListExample()
```

HashMap Class

The HashMap class uses a hashtable to implement the Map interface. This allows the execution time of basic operations, such as get() and put(), to remain constant even for large sets.

Example:

```
double[] generateRandomArray(int size){
 double[] arrayDouble = new double[size];
 for(int i = 0; i < size; i++) {
  arrayDouble[i] = Math.random();
 }
 return arrayDouble;
}
HashMap searchMin(double[] arrayValues){
 int indexMin = 0;
 double valMin = arrayValues[0];
 for(int i = 1; i < arrayValues.length; i++) {
  if(arrayValues[i] < valMin) {
   valMin = arrayValues[i];
   indexMin = i;
```

```
    }
   }
   HashMap map = new HashMap();
   map.put("index", indexMin);
   map.put("value", valMin);
   return map;
  }
  void hashMapExample() {
   // generate a random array
   double[] values = generateRandomArray(10);
   // print array
   System.out.println("Values:");
   for(int i = 0; i < values.length; i++) {
    System.out.println(i + " " + values[i]);
   }
   System.out.println();
   // search minimum value and index
   HashMap map = searchMin(values);
   System.out.println("Minimum value " + map.get("value"));
   System.out.println("Index " + map.get("index"));
  }
```

Save the file and try in the Calculator.

```
> hashMapExample()
```

Cmplx class

Cmplx is a class which implements complex numbers in Java. It includes basic operations that can be performed on complex numbers such as, addition, subtraction, multiplication, conjugate, modulus and squaring.

- **Cmplx(double real, double imag):** Returns new complex (real,imag).
- **Cmplx(double real):** Returns new complex (real, 0).
- **double Cmplx.real(Cmplx c):** Returns the real part of a complex value.
- **double Cmplx.imag(Cmplx c):** Returns the imaginary part of a complex value.
- **Cmplx Cmplx.conjugate(Cmplx c):** Returns the conjugate of a complex value.
- **Cmplx Cmplx.opposite(Cmplx c):** Returns the opposite of a complex value.
- **double Cmplx.mod(Cmplx c):** Returns the module of a complex value.
- **double Cmplx.abs(Cmplx c):** Returns the absoute of a complex value. Corresponds with the module.
- **double Cmplx.phase(Cmplx c):** Returns the phase (argument) of a complex value; the returned value is in the range 0 through 360.
- **double Cmplx.arg(Cmplx c):** Returns the phase (argument) of a complex value; the returned value is in the range 0 through 2*pi.
- **double Cmplx.argd(Cmplx c):** Returns the phase (argument) of a complex value; the returned value is in the range 0 through 360.
- **Cmplx Cmplx.sum(Cmplx c1, Cmplx c2):** Returns the sum of two complex values.
- **Cmplx Cmplx.sum(Cmplx c, double d):** Returns the sum of complex value by a double value.
- **Cmplx Cmplx.sub(Cmplx c1,Cmplx c2):** Returns the subtraction of two complex values.

- **Cmplx Cmplx.sub(Cmplx c, double d):** Returns the subtraction of complex value by a double value.
- **Cmplx Cmplx.mul(Cmplx c1, Cmplx c2):** Returns the multiplication of two complex values.
- **Cmplx Cmplx.mul(Cmplx c, double d):** Returns the multiplication of complex value by a double value.
- **Cmplx Cmplx.div(Cmplx c1, Cmplx c2):** Returns the division of two complex values.
- **Cmplx Cmplx.div(Cmplx c, double d):** Returns the division of complex value by a double value.
- **Cmplx Cmplx.exp(Cmplx c):** Returns Euler's number e raised to the power of a complex value.
- **Cmplx Cmplx.sqrt(Cmplx c):** Returns the correctly rounded positive square root of a double value.
- **Cmplx Cmplx.sqrt(Cmplx c):** Returns the correctly rounded positive square root of a complex value.
- **double Cmplx.amplitude20(Cmplx c):** Returns the amplitude, in db, of a complex value from a lineal magnitude.
- **double Cmplx.amplitude10(Cmplx c):** Returns the amplitude, in db, of a complex value from a squared magnitude.
- **double Cmplx.db20(double d):** Returns the amplitude, in db, of a module of a complex value from a lineal magnitude.
- **double Cmplx.db10(double d):** Returns the amplitude, in db, of a module of a complex value from a squared magnitude.
- **Cmplx Cmplx.nat20(Cmplx c):** Returns the complex natural value of a complex value in (dB,degrees) from a lineal magnitude.
- **Cmplx Cmplx.nat10(Cmplx c):** Returns the complex natural value of a complex value in (dB,degrees) from a squared magnitude.
- **double Cmplx.idb20(double d):** Returns the module of a complex natural value of a value in dB from a lineal magnitude.
- **double Cmplx.idb10(double d):** Returns the module of a complex natural value of a value in dB from a squared magnitude.

Example:

```
void complexExample(double real, double imag){
 Cmplx complex = new Cmplx(real, imag);
 System.out.println("Modulus = " + Cmplx.mod(complex));
 System.out.println("Argument (in radians) = " + Cmplx.arg(complex));
 System.out.println("Argument (in degrees) = " + Cmplx.argd(complex));
}
```

Save the file and try in the Calculator.

```
> complexExample(1, -1)
```

Import Libraries

Files that require any additional library must start with the loading sentence, that is, "**import library**".

For newFASANT tool, the main libraries that can be required are:

java.io:
import java.io

java.lang:
import java.lang

java.net:
import java.net

java.util:
import java.util

Further Java Information

For more information about the Java language programming you can visit the following external links:

- https://docs.oracle.com/javase/tutorial/
- https://docs.oracle.com/javase/8/docs/index.html
- http://docs.oracle.com/javase/8/docs/api/

2.6.6 Script

newFASANT scripts are text files (in ASCII code) with `.nfs` extension that allow to save or load all the geometrical commands that have been run.

Two type of data are generated in script files:

- Geometrical parameters defined that can be visualized within the **Geometry- Parameters** menu.
- Geometric operation that have been performed that can be seen within the **Edit -History** menu.

By using this tool, a geometry that depends on multiple parameters may be generated and exported to be used in a different project with the same or even modified parameters. Use it whenever you need to set-up the configuration parameters of the existing geometry.

Be careful after loading an existing script, as the original units are not taken into account for the script loading/saving and the geometrical objects are generated in the working units.

Usage Examples

This sections show an example of application of the scripts tool. We will show how to create a dihedral with parametric angle.

The **Centimeters Units** have been choosen and **default Reference Plane** is valid.

- **First step Parameters definition**. Click on **Geometry - Parameters - Define Parameters** menu, and create the parameters that are shown in the figure below.

Define Parameters ×

Parameter	Values
x	{0.0}
y	{0.0}
z	{0.0}
L	{1.0}
angle	[45.0,315.0] 7

Figure 258: Defined parameters

- **Second step: Geometry creation**. The dihedral consists on two square planes that share a common edge, and the angle between them is given by the parameter **angle** that has been previously defined. The next figure show the commands performed to create the geometry.

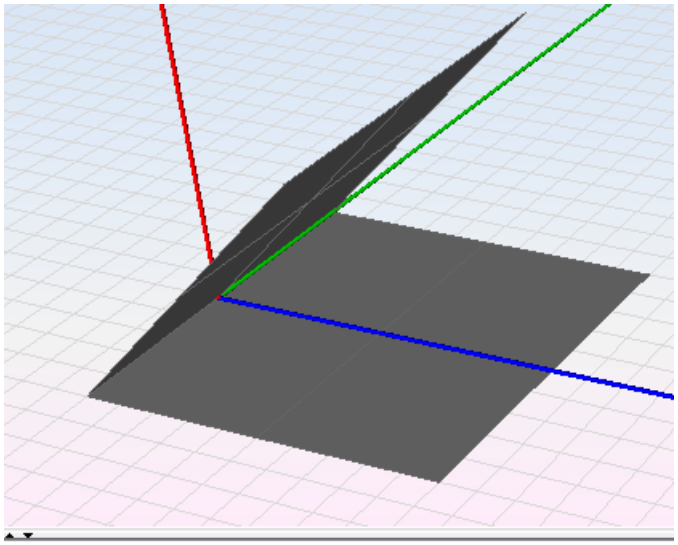
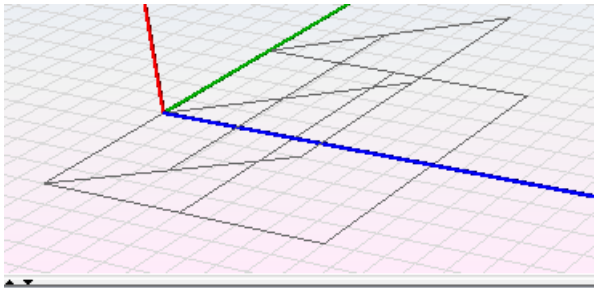


Figure 259: Commands that have been run to create a dihedral.

- **Third step Script saving**. Export the script by clicking on **Tools - Script - Save** menu.
- **Fourth step Script visualization**. Open the exported .nfs file with a text editor to visualize its content. Note that a new parameter (named **$1**) has been automatically generated for saving a mathematical operation. Edit the **angle** parameter, for example, change it from" [45.0,315.0] 7" to "[15.0,75.0] 5".
- **Fifth step Script loading**. Open a new instance of newFASANT. Note that a new instance is suggested to restart the default parameters (such as the **Units**) and remove both the **History** and the **Defined Parameters**. Click on **Tools - Script - Load** menu and select the modified script. Note that although the script was generated in **Centimeters**, the new geometry has been created in **Meters**, as they are the current working **Units**. Now, the angle of the new dihedral is set to 15 degrees, as it is the first value defined within the script file.



```
command> script -load "C:\Users\newfasant\Desktop\script.nfs"
command> set x {0.0}
command> set z {0.0}
command> set L {1.0}
command> set y {0.0}
command> set $1  = (y-(L/2.0))
$1 = -0.5
command> set angle [15.0,75.0] 5
command> plane -n id216397ab -p x $1 z L L
command> duplicate -s id216397ab -n id7af8c3e4
command> rotate -s id7af8c3e4_1 -p x y z x $1 z angle
command>
```

Figure 260: Modified script loading

Load

With this option the user can execute a list of commands written in a text file. The commands written in the file will be executed in order, one by one. This option is useful to execute a group of operations by a command.

Example, script that creates a geometry with copy and rotate commands:

```
#
#  newFASANT script file
#
#    Copy this content to a .nfs file and load it.
#
box -n myGeometry -p -1.0 -1.0 0.0 2.0 2.0 2.0
move -s myGeometry -p 0.0 0.0 0.0 6.0 0.0 0.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
copy -s myGeometry -n myCopy -p 0.0 0.0 0.0 0.0 0.0 0.0
rotate -s myGeometry -p 0.0 0.0 0.0 0.0 0.0 1.0 45.0
#
#  End script file
#
```

Results of the previous script.

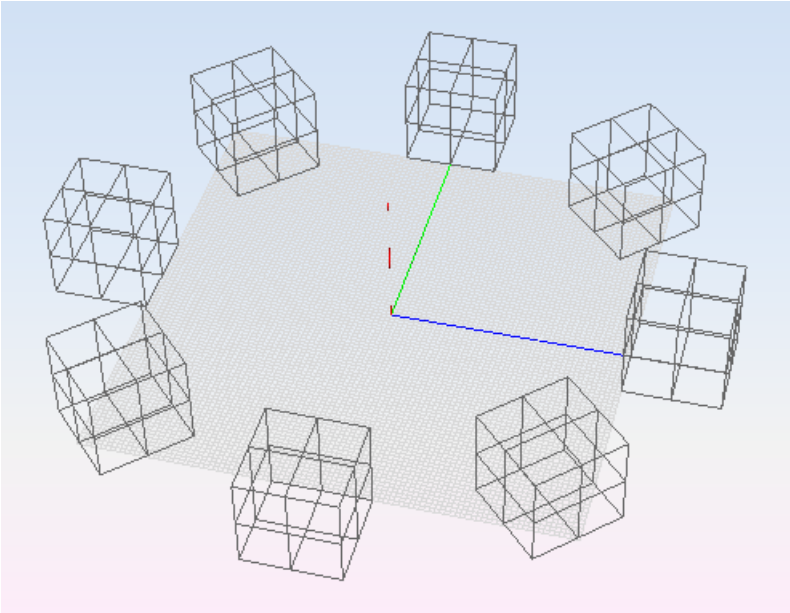


Figure 261: Geometry after loading the script

Save

The user can export all executed commands with this option. To see a list of executed commands, enter into the menu **Edit**→ **History**.

Advanced Description

This section is detailed for advanced users that need to write the scripts by themselves. After reading this section, the user may be able to understand the script examples available at https://altairone.com/.

Regarding general information about newFASANT scripts, the next considerations must be taken into account:

- Blank lines may be inserted to get a more readable file.
- Comments start with "#" character. Every sentences of the current line that are written after this character will be omitted.
- Every line run a new order. However, a line may be split in so many lines as desired, as stated in the below specification.
- Character " \" is the Escape character, so the next character will be omitted. It is usually inserted to split an order in multiple lines and then to obtain a more readable file.
- Order lines start with the command name.

The following specifications are applied to most of commands that may be loaded in script files:

- **set** command creates new parameters, specified by its name and values (it may be fixed values, range values or even functions)

- **-n** option set the output name of the resulting elements within the current command. Be careful to provide unique names to every command; otherwise the console will not load the script after detecting repeated output names.
- **-s** option list the selected objects as input arguments to the current command. It is used in commands that require the selection of objects as input.
- **-p** option list the required parameters for the current command.
- **-t** option list the transformation matrix of the reference plane required for using the current command. It is the next 4x4 matrix with the below specifications:

	column0	**column1**	**column2**	**column3**
row0	m00	m01	m02	m03
row1	m10	m11	m12	m13
row2	m20	m21	m22	m23
row3	m30	m31	m32	m33

  - the matrix is written in rows as: *-t m00 m01 m02 m03 m10 m11 m12 m13 m20 m21 m22 m23 m30 m31 m32 m33*
  - the elements *m00 m10 m20* within *column0* defines the X unitary vector-direction of the reference plane.
  - the elements *m01 m11 m22* within *column1* defines the Y unitary vector-direction of the reference plane.
  - the elements *m02 m12 m22* within *column2* defines the Z unitary vector-direction of the reference plane.
  - the elements *m00 m10 m20* within *column3* defines the X Y Z point where the reference plane is centered.
  - the elements *m30 m31 m32 m33* withinrow3 are always set to 0 0 0 1 respectively.
- **-c** option is only available in some commands, and it may be used with different purposes:
  - It indicates that the current command will generate a ***copy***of the input objects. It may be used in commands such as: *symmetric*,...
  - It indicates that the current command will generated a ***closed***object. Otherwise, only the side surfaces are generated. It may be used in commands such as: *cylinder*, *cone*, *ogive*,...
  - It indicates that the current command will have ***cutting***objects that will not be modified. It is only used in split options, such as ***splitSurfaces***, ***splitCurves***,...

2.6.7 Batch

This option lets you select multiple projects and execute them one by one.

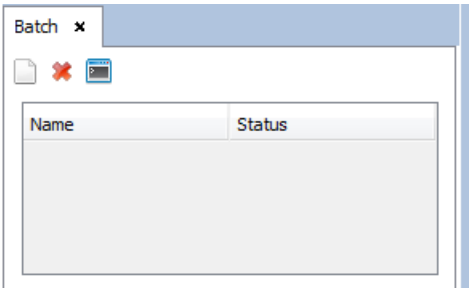


Figure 262: Batch

To add projects to the table press the button **Add project**. To delete multiple projects, select them in the table and press the button **Remove selected projects**. Pressing the **Run** button to execute all projects inserted into the table.

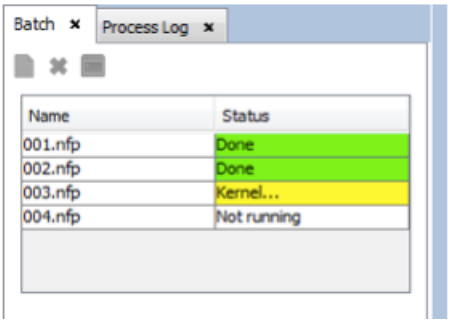


Figure 263: Run Batch

To stop the Batch simulations, go to **Process Log** tab and press **Abort** button. To finish the Batch simulation, the user can be visualize the log.

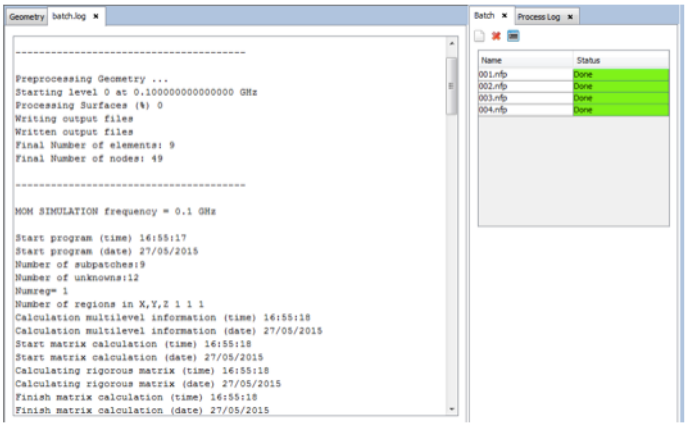


Figure 264: Batch Log

2.6.8 Remote

For more information on how to set up a remote server, see Remote Server.

2.7 Help Menu

When selecting Help, the following menu appears:

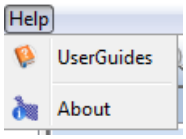


Figure 265: Help menu.

2.7.1 About

In this menu some specific information about newFASANT is shown. Important information about the software release being used is included as well.

2.8 Annex 1: Graphics Advanced Options

This section explains the advanced options menu that appears when the user clicking with a right button.

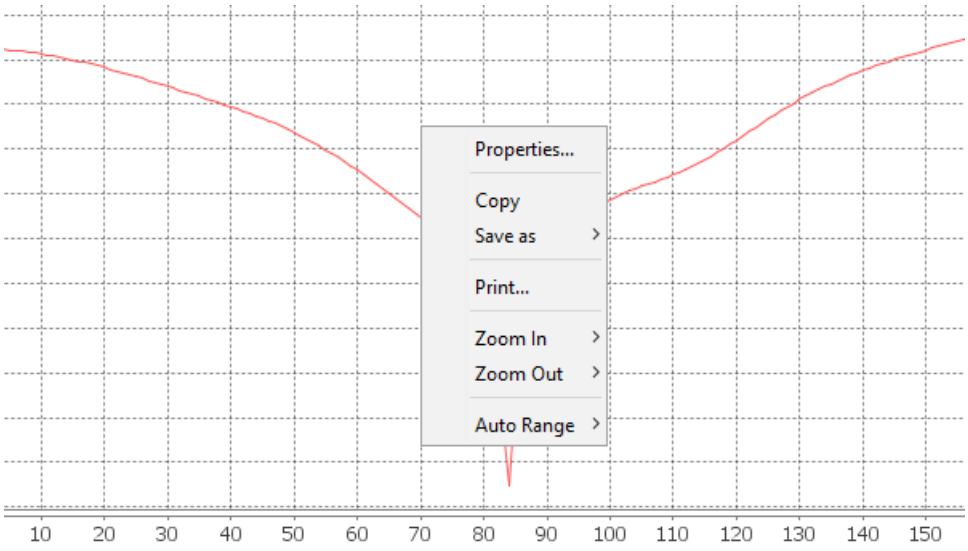
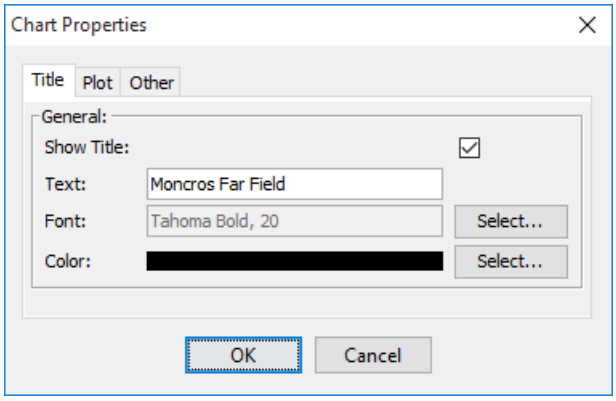


Figure 266: Chart Advanced Options menu

2.8.1 Properties

This section explains the following properties panel, that appears clicking on "Properties..." option on the main menu:



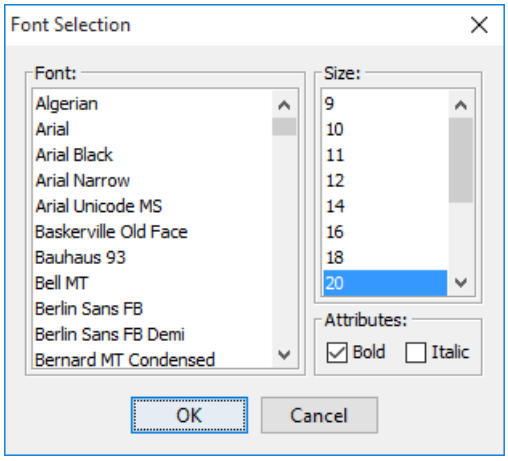
This panel is divided on three tabs:

Title

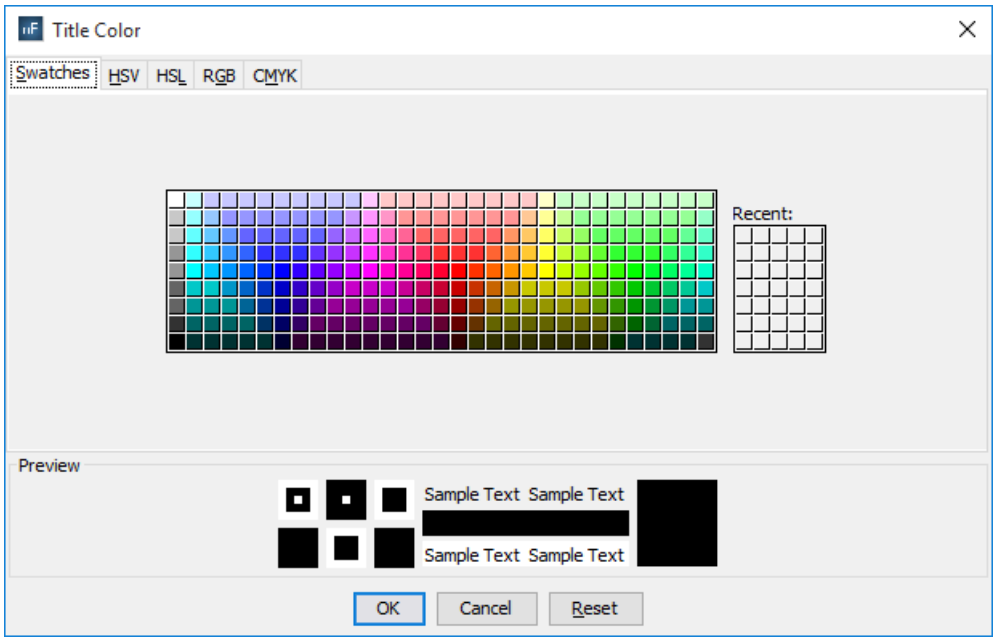
This tab allows the user to configure the following parameters referenced to the title of the chart:

- Show Title. When this check box is selected, the title will be visible on the chart panel, otherwise there is no visible title.

- Text. This text field set the name for the chart title.
- Font. This option allows the user to configure the font settings. The text field resumes the current configuration and "Select..." button displays a panel to customize the configuration.



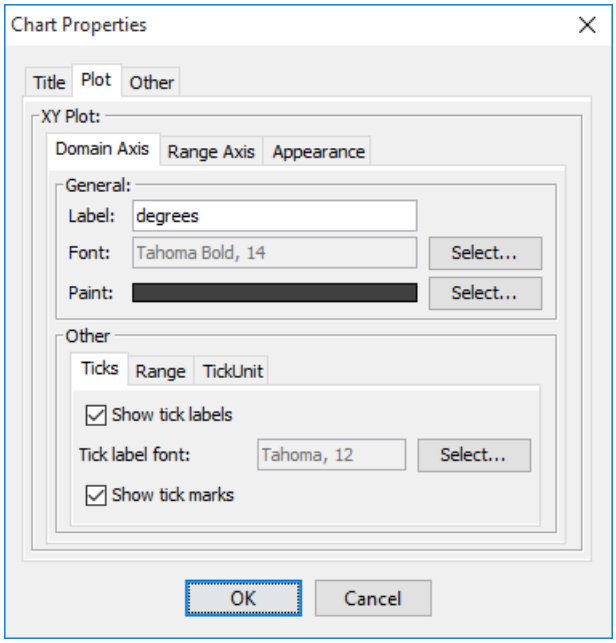
  - Font. With this panel the user select the font family for the title.
  - Size. With this panel the user select the font size for the title.
  - Attributes. With this panel the user can apply bold and italic effects to the font.
- Color. This option allows the user to configure the font color. The text field shows the current color and "Select..." button displays a panel to select a new color.



Plot

This tab allows the user to configure the parameters for the axis and the appearance, divided on three tabs:

- Domain Axis. With this tab the user configures the parameters for the horizontal axis (X):

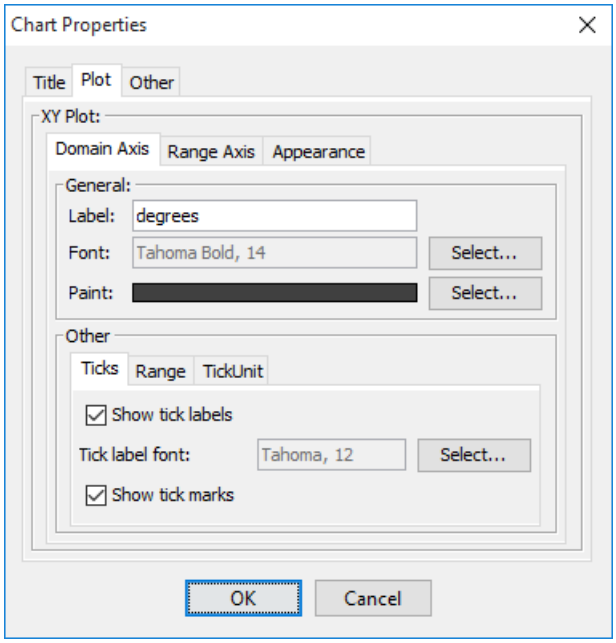


"General" panel has the following parameters:

- Label. This text field set the name for the axis.
- Font. This option allows the user to configure the font settings. The text field resumes the current configuration and "Select..." button displays a panel to customize the configuration. The panel that appears is the same as the 'Font' option explained on 'Title' tab.
- Paint. This option allows the user to configure the font color. The text field shows the current color and "Select..." button displays a panel to select a new color. The panel that appears is the same as the 'Font' option explained on 'Title' tab.

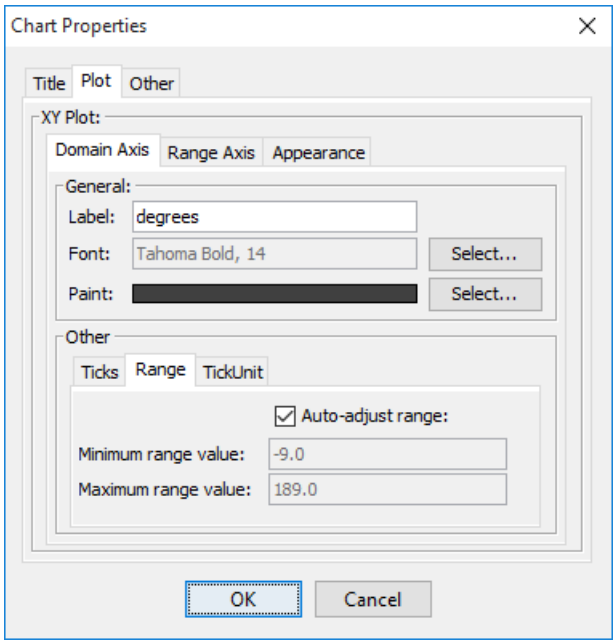
"Other" panel has the following parameters, divided on three tabs:

- Ticks. This tab configures the parameters to the ticks of the axis:



- Show tick labels. When this check box is selected, ticks values will be visible over the axis.
- Tick Label Font. This option allows the user to configure the font settings. The text field resumes the current configuration and "Select..." button displays a panel to customize the configuration. The panel that appears is the same as the other 'Font' option explained on this section.
- Show tick marks. When this check box is selected, ticks will be visible over the axis.

- Range. This tab configures the parameters to the ticks of the axis:

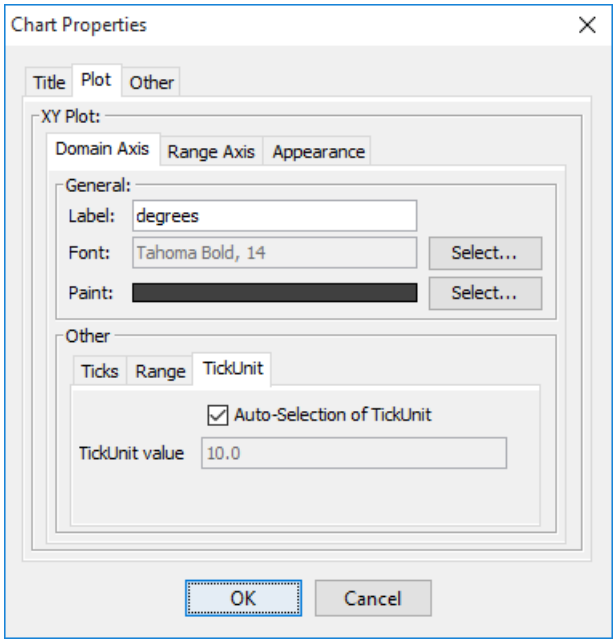


- Auto-adjust range. When this check box is selected, the maximum and minimum visible values for the axis on the chart will be calculated automatically over the range of values. This option

calculates the total range with a margin before the first value and after the last value of the represented plot.

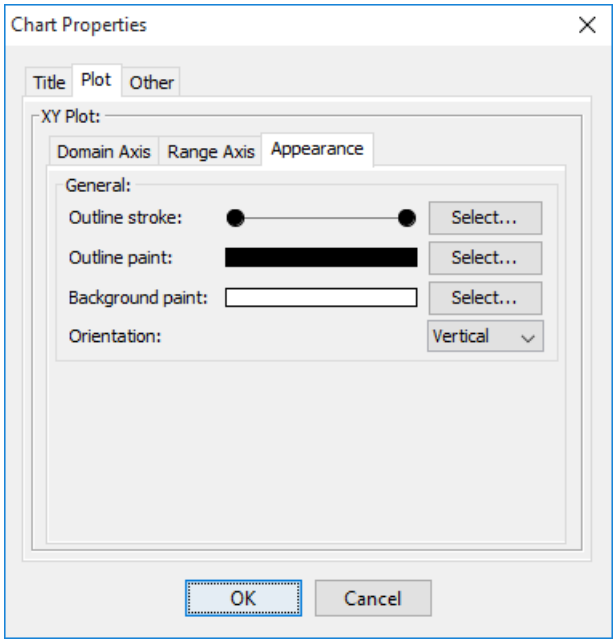
- Minimum range value. This option allows the user to configure the initial value for the chart (this value is not the first value of the plot to be represented). This value can be used to have or not a margin before the represented plot. Can be used to eliminate some values at the beginning of the represented plot. This option is only enabled if the "Auto-adjust range" check box is not selected.
- Maximum range value. This option allows the user to configure the final value for the chart (this value is not the last value of the plot to be represented). This value can be used to have or not a margin after the represented plot. Can be used to eliminate some values at the ending of the represented plot. This option is only enabled if the "Auto-adjust range" check box is not selected.

- TickUnit. This tab configures the parameters to the units of the ticks of the axis:



- Auto-selection of TickUnit. When this check box is selected, the value between two consecutive ticks for the axis on the chart will be calculated automatically over the range of values.
- TickUnit value. This option allows the user to configure the value for the interval between two consecutive ticks. This option is only enabled if the "Auto-selection of TickUnit" check box is not selected.

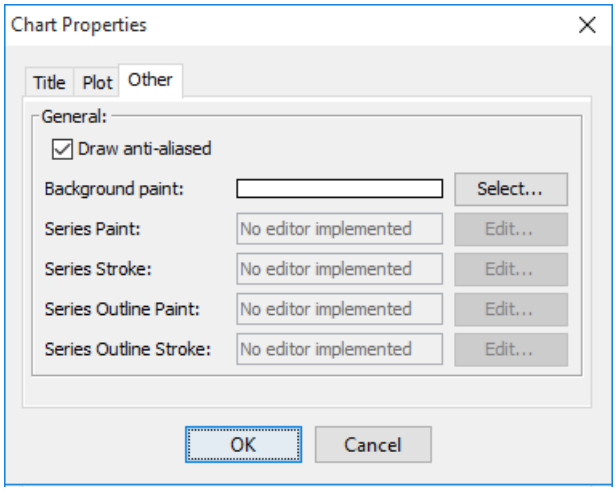
- Range Axis. With this tab the user configures the parameters for the vertical axis (Y):
- Appearance. With this tab the user configures the parameters for the chart:



- Outline stroke. This option allows the user to select the type of the outline of the chart. With "Select..." button a panel appears, where the user can select the prefer type of line.
- Outline paint. This option allows the user to configure the outline color of the chart. The text field shows the current color and "Select..." button displays a panel to select a new color. The panel that appears is the same as the other color choosers explained on this section.
- Background paint. This option allows the user to configure the background color of the chart. The text field shows the current color and "Select..." button displays a panel to select a new color. The panel that appears is the same as the other color choosers explained on this section.
- Orientation. This option allows the user to select the orientation of the chart. When "Vertical" option is selected, the domain appears on the horizontal axis and the range appears on the vertical axis. Otherwise, the axis for the domain and the range will be changed.

Other

This tab allows the user to configure the parameters for the appearance of the plot and series:



- Draw anti-aliased. When this option is selected,the display of the graph is improved, smoothing the contours of the components. When this option is selected, chart drawing has more definition for all the drawing components.
- Background paint. This option allows the user to configure the Background color of the chart panel, not the chart. The text field shows the current color and "Select..." button displays a panel to select a new color. The panel that appears is the same as the other color choosers explained on this section.
- Series Paint. This option is not implemented.
- Series Stroke. This option is not implemented.
- Series Outline Paint. This option is not implemented.
- Series Outline Stroke. This option is not implemented.

2.8.2 Copy

This option allows the user to copy the chart panel to paste on a desired file.

2.8.3 Save As

This option allows the user to save the chart panel as a file. When this option is selected a submenu appears to select the file format to the output file:

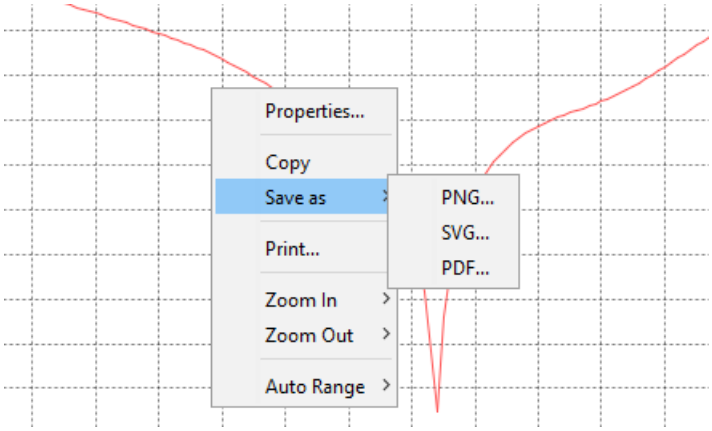


Figure 267: Selecting "PNG...", "SVG..." or "PDF..." a file chooser appears to select the path to the output file.

2.8.4 Print

This option allows the user to print the chart panel. When this option is selected a panel appears to configure printing parameters:

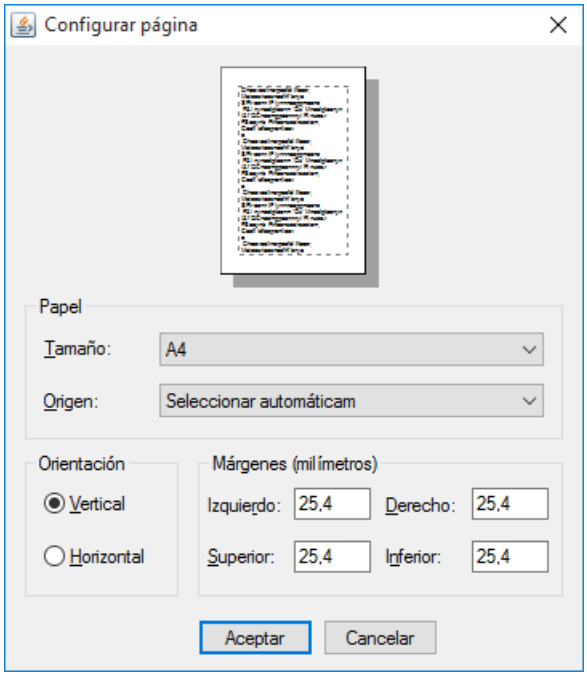
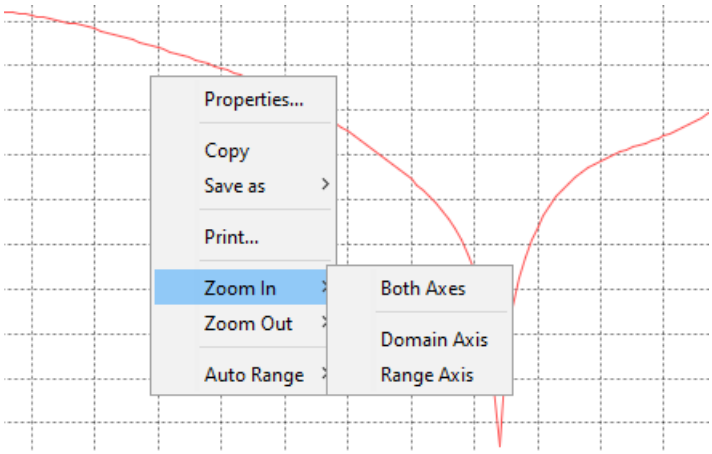


Figure 268: This panel is a default printing panel that allows configure parameters as:

- Size ('Tamaño' on the figure). Size of the print paper.
- Printer ('Origen' on the figure). Machine used to print the chart.
- Orientation ('Orientación' on the figure). Orientation of the print pages, vertical or horizontal.
- Margins ('Márgenes' on the figure). Margins at the four sides of the page to keep without print.

2.8.5 Zoom In

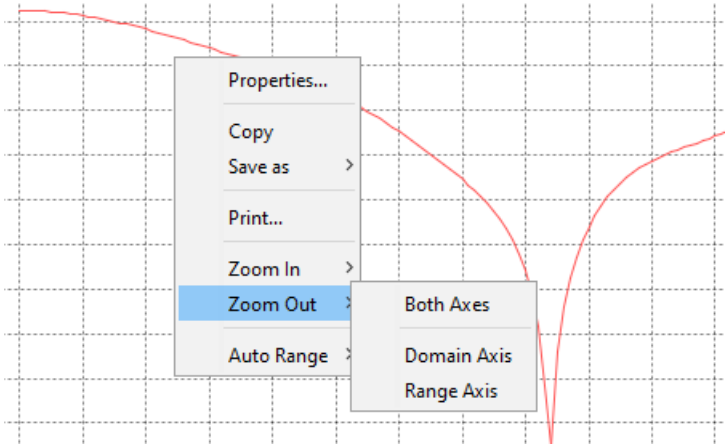
This option allow the user to reduce the visualization range of the chart to see less values. When this option is selected a sub-menu appears with three options to the zoom:



- Both Axes. Reduce the visualization range on the two coordinates of the chart.
- Domain Axis. Reduce the visualization range on the horizontal axis of the chart.
- Range Axis. Reduce the visualization range on the vertical axis of the chart.

2.8.6 Zoom Out

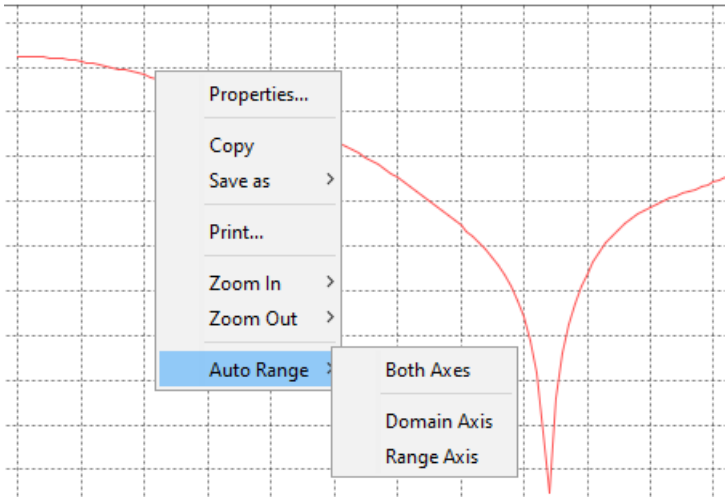
This option allow the user to enlarge the visualization ranges of the chart to see more values. When this option is selected a sub-menu appears with three options to the zoom:



- Both Axes. Enlarge the visualization range on the two coordinates of the chart.
- Domain Axis. Enlarge the visualization range on the horizontal axis of the chart.
- Range Axis. Enlarge the visualization range on the vertical axis of the chart.

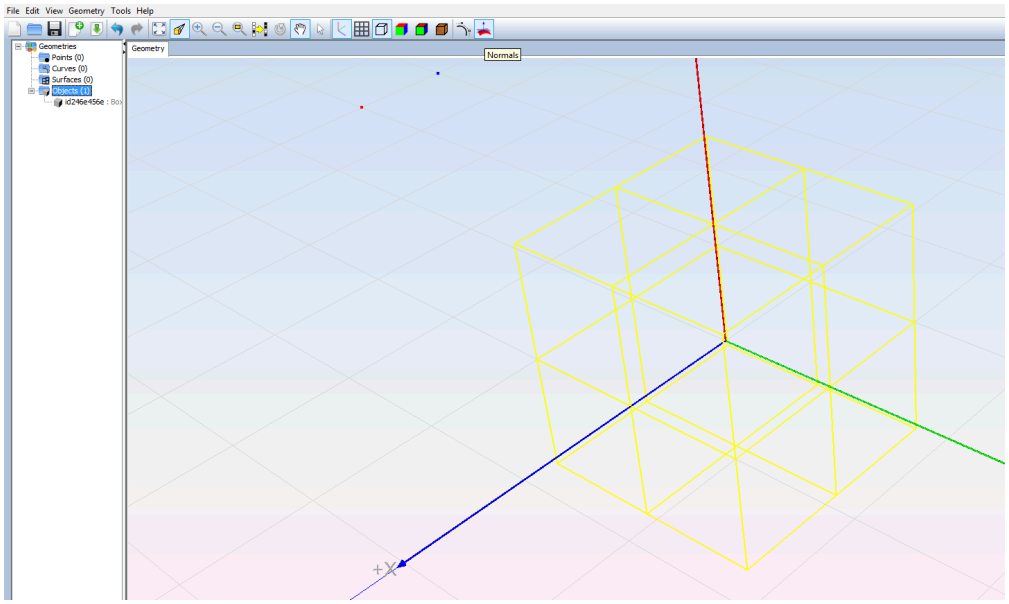
2.8.7 Auto Range

This option allow the user to set the range of the chart values automatically, calculated over the represented values. When this option is selected, a sub-menu appears with three options to set the Auto Range:

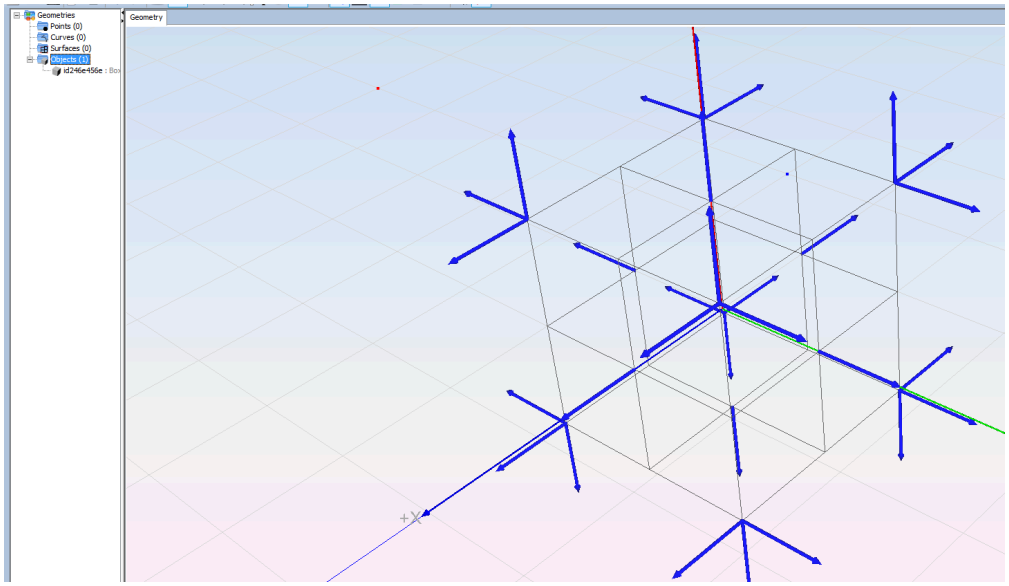


- Both Axes. Set the chart values automatically on the two coordinates.
- Domain Axis. Set the chart values automatically on the horizontal axis.
- Range Axis. Set the chart values automatically on the vertical axis.

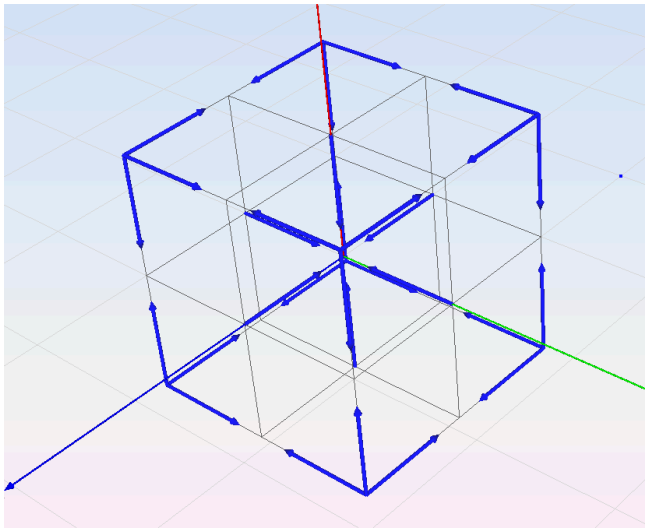
2.9 Annex 2: Inverting Normals



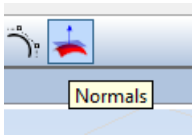
We created a cube of side 2. We select the cube and click on normal button in the upper bar.



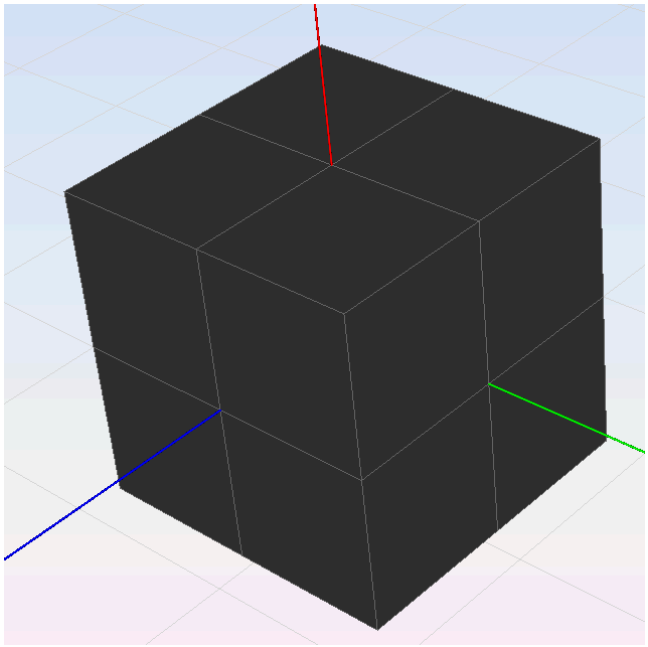
The normal vectors with their senses are visualized.



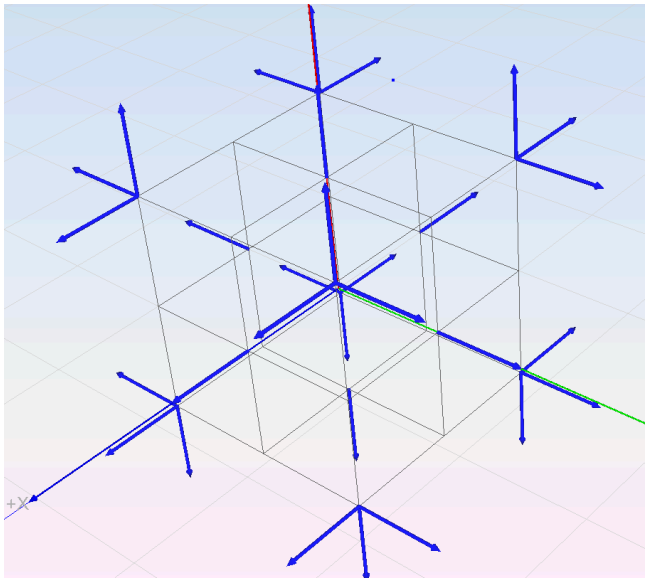
By clicking on the main window the sense of the normal vectors are changed.



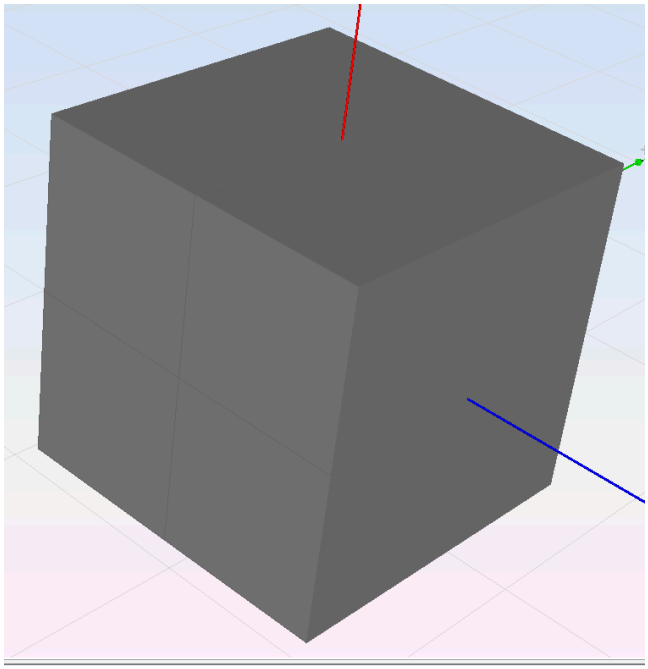
By clicking on the "Normals" button the new normal senses are saved.



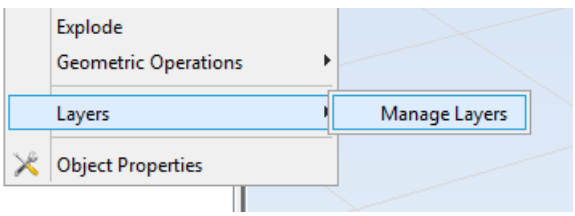
By clicking on the "shaded with lines" button the cube appears with a dark color because its normal is pointing ingoing.

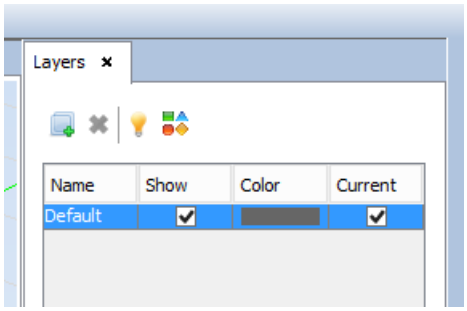


We click again in"Normal button" and then in the main window the change the sense of normal. Finally we click on "Normals button" to save the new normal sense.

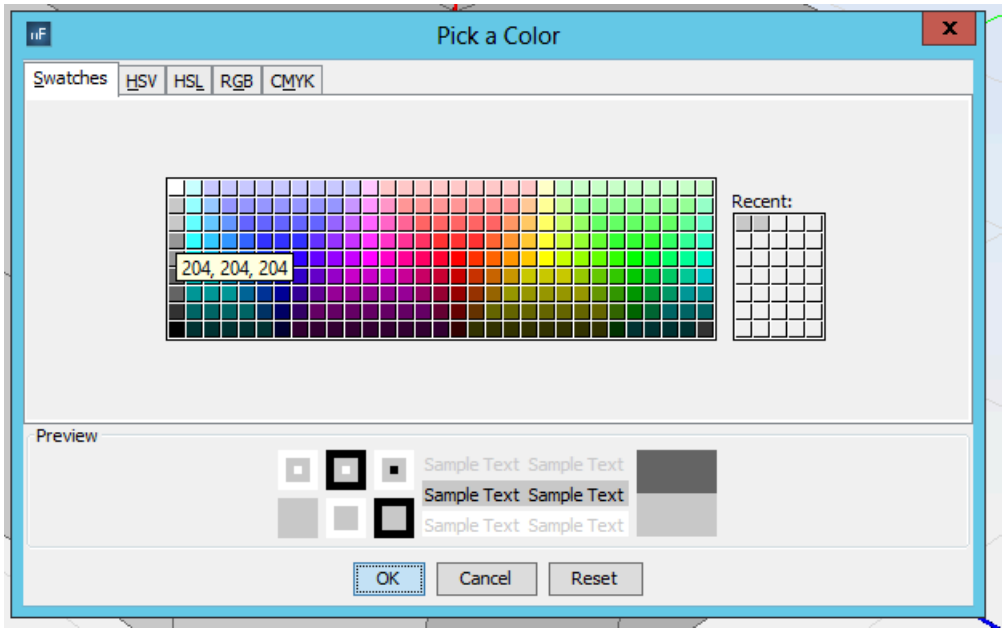


Now we have the cube with a brilliant color because its normal is pointing outgoing.

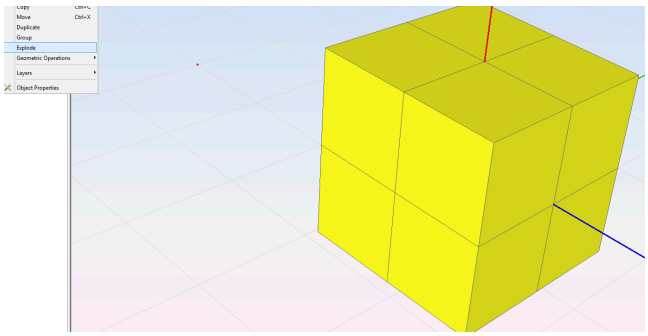




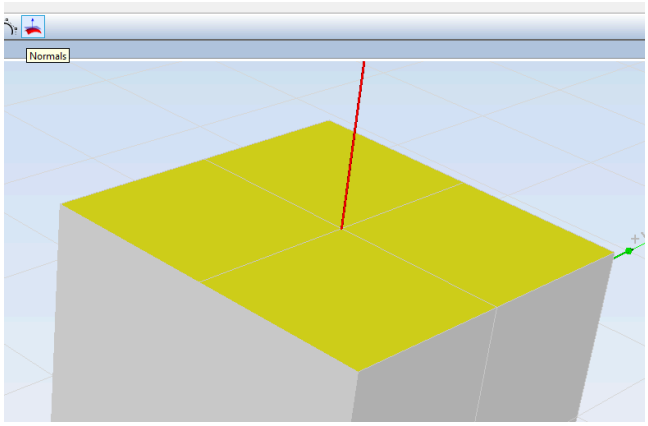
Some time is useful change the color to a more clear one. For that click on Layer, manage layer.



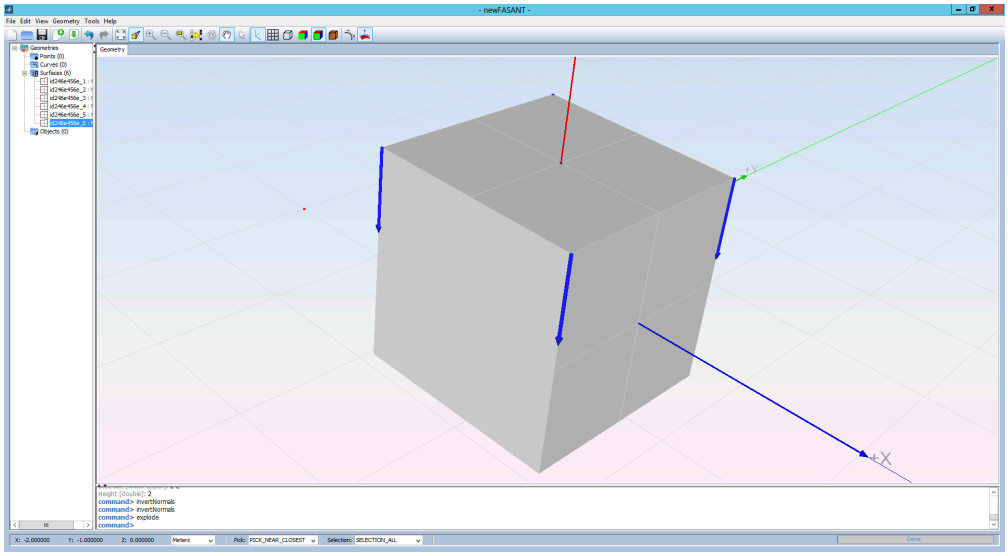
Change the color of the layer.



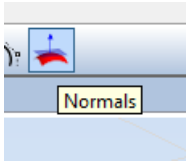
Now we will change the sense of the normal of only the surface in the top of the cube. We select the cube and explode it.



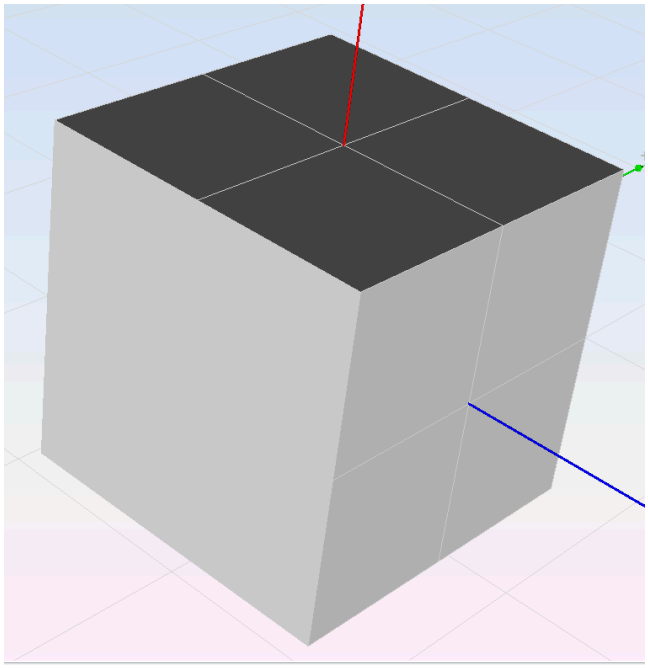
We select the surface in the top and click on the "Normals" button.



We click on the main window the change the sense of the normal of the top surface.



We save the new normal sense of the top surface by clicking on the “Normal” button.



Now the top surfaces are darker than the other surfaces because the normal in the top surface is pointing ingoing.

Post-Process

3

Results can be post-processed to obtain customized results.

This chapter covers the following:

- 3.1 Post-Process Menu (p. 276)
- 3.2 Post-Process Window (p. 277)
- 3.3 Trace Window (p. 279)
- 3.4 Trace Results Output (p. 283)
- 3.5 Examples (p. 287)

3.1 Post-Process Menu

Post-Process is an option from the “Show Results” menu, available for some modules.

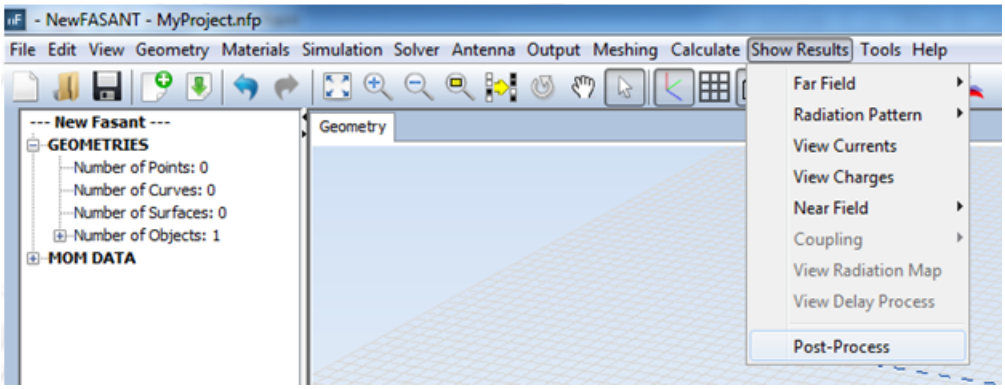


Figure 269: Show Results menu

The steps to be followed in order to post-process a result are the same in all the cases:

1. Select the type of the representation for the results in the first window that appears (post-process window).

2. Select, on the second window (trace window), the type of the results for the post-processing, set the required options and select the domain of the representation (only for table and plot representations).

In the next chapters more details are given regarding the functionalities of the two windows that post-process uses to get a customized result.

3.2 Post-Process Window

This window allows the user to select the type of representation to visualize the new trace.

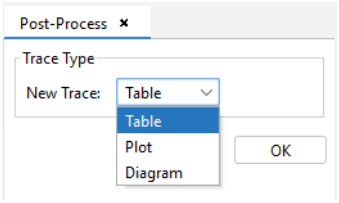


Figure 270: Post-Process Window

New Trace options:

- Table visualize the results as a table of values indicating, for each value of the domain, the value of the result to be associated.

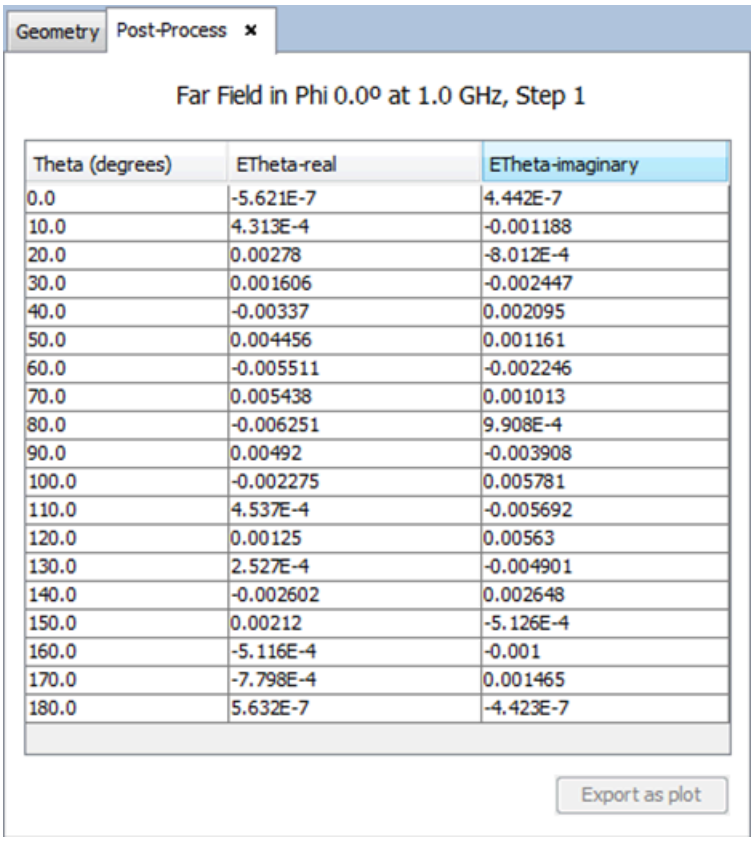


Figure 271: Table trace

- Plot visualize the results as a plot of values drawing a line that represent the result values for each value of the domain of representation.

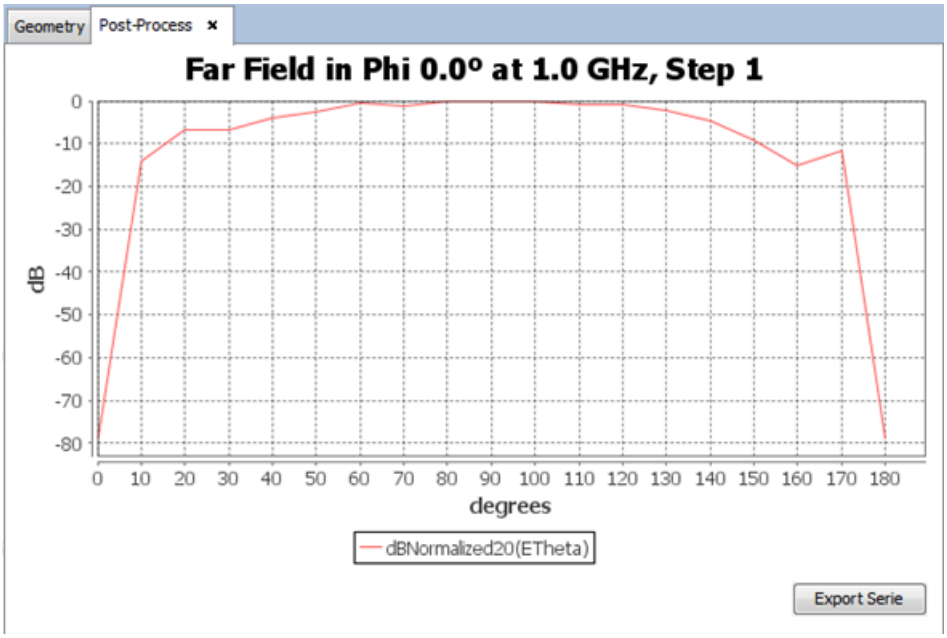


Figure 272: Plot trace

Most of plots show 2-D curves of a result field versus an input parameter. This type of graphics share additional features of zoom, raxis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

- Text File visualize the results on plain text and show the results file.

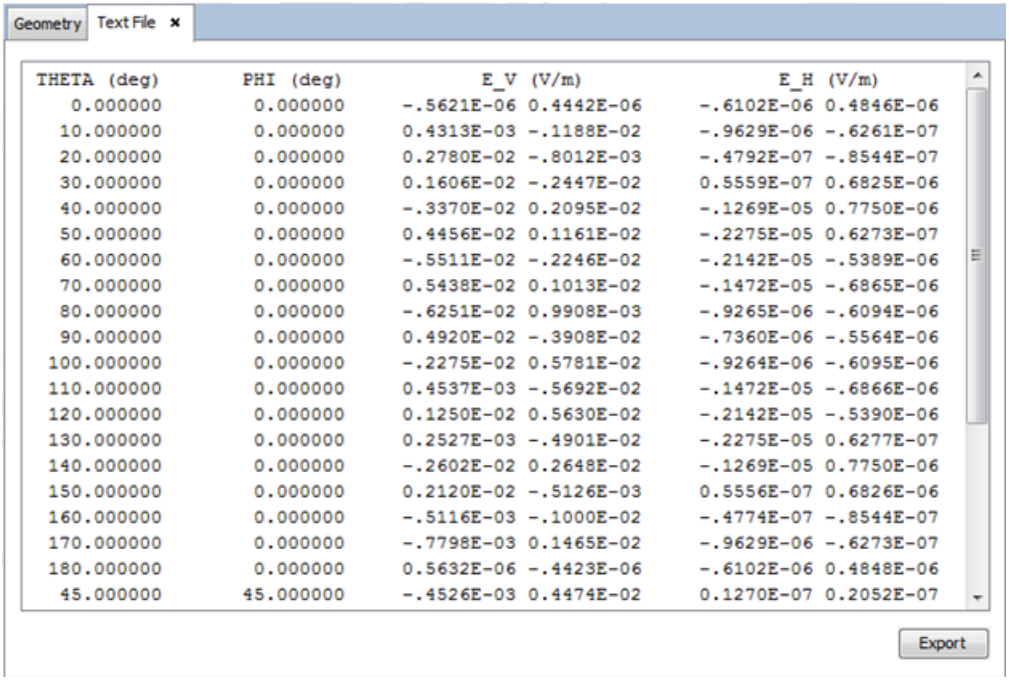


Figure 273: Diagram Trace

3.3 Trace Window

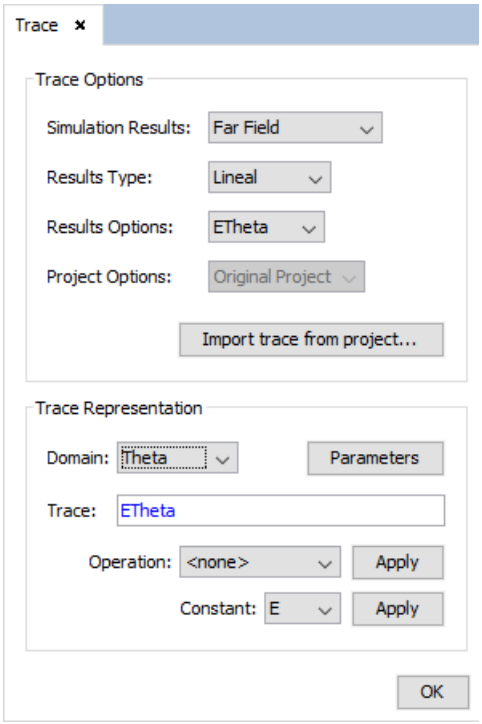


Figure 274: Trace Window

3.3.1 Trace Options Panel

This panel allows the user to configure the results that will be shown in the new trace.

- Simulation Results in this list, the user selects the type of the result to visualize. This list only includes the results according to the type of representation selected from the post-process window. For example, if the user selects the “table” option in the post-process window, the “Current Density” option for the simulation result will not appear because “Current Density” results only can only be visualized in diagram trace.

- Results Type in this list, the user selects a sub-type form the results selected on the previous list. This list only includes the sub-types of the results selected on the previous list, modifying it when the user changes the selected value on the “Simulation Results” list.

- Results Options this list is an auxiliary list that will be filled only if the results have other options (additional to the “Result Type” options). This list will be used, for example, for the “Near Field” results that have “Ex”, “Ey”,“Ez”,“ETotal”,“Hx”,“Hy”,“Hz” and“HTotal” as “Results Type” and “Scatter”, “Direct” and “Total” as “Result Options”.

- Project Options allows the user to select which project will take the results to load at that time to bring them into line results. The projects will be displayed in the list with the name of the project in question, accompanied by the nickname that will put the loaded result, for example, if the selected project is

"Project002.nfp (_1)" and the result selected in "ETheta" when loaded in the result line will appear as "ETheta_2" .

- “Import trace from project” button Allows the user to import the result that corresponds with a previous project having these same results. This button shows a window where the user needs to select the “.nfp” file(s) to import the results. Then, in the “Result Type” list, a new option appears including the name of the selected option with an added numeric value corresponding to each imported file.

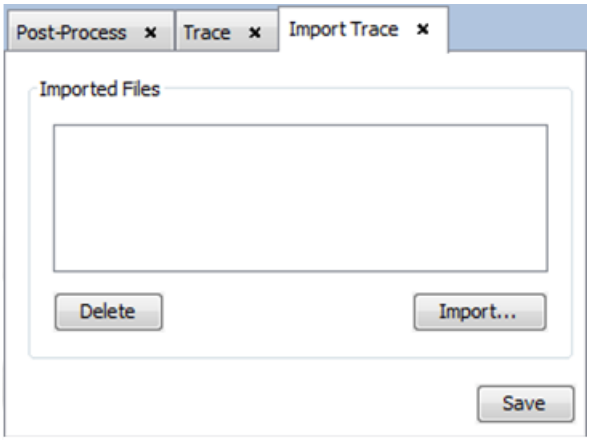


Figure 275: Import trace Window

- “Delete” remove the selected row for the file previously imported and close it from imported projects.
- “Import” import the results from a selected file and open the project to used its results.
- “Save” load the changes into "Trace Window" to actualise the enable imported projects.

3.3.2 Trace Representation Panel

This panel allows the user to configure the representation of the results and the application of formulas over the original results to customize the results to be visualized.

- Domain: this list allows the user to select a parameter to visualize the results (applying a function to such parameter). For each value of this parameter one result value will be visualized.
- “Parameters”: this button shows a window that allows the user to configure the options for the domain of the representation. This window is specific for the result type selected and includes parameters such as “step”, “frequency”, “cut” (for far field), “point”, “observation” (for near field) and “Angular Average” for results with domain in degrees and trace type as table, plot or diagram.
- Trace: this option shows the value that, as a function of the original results, will be represented as a result. This option allows the user to customize the representation results with the operation option and the selected original results.
- Operation: this list allows the user to select an operation to be applied to the trace results. The list contains all the operations supported. The operations are:
  - <none> apply the result type directly.
  - abs returns the absolute value of the value.

- acos returns the arc cosine value in degrees.
- acosr returns the arc cosine value in radians.
- ampl10 returns the amplitude in dB of a squared magnitude value.
- ampl20:returns the amplitude in dB of a lineal magnitude value.
- asin returns the arc sine value in degrees.
- asinr returns the arc sine value in radians.
- atan returns the arc tangent value in degrees.
- atanr returns the arc tangent value in radians.
- ceil rounds value to the higher integer.
- complex returns a new complex number with the first parameter as real part and second parameter as imaginary part.
- cos returns the cosine value of the value introduced in degrees.
- cosr returns the cosine value of the value introduced in radians.
- dB10 returns the amplitude in dB of a squared magnitude value.
- dBNormalized10 returns the normalized amplitude in dB of a squared magnitude value.
- dB20 returns the amplitude in dB of a lineal magnitude value.
- dBNormalized20 returns the normalized amplitude in dB of a lineal magnitude value.
- deg returns the value in degrees from a value in radians.
- exp returns the power of ‘e’ raised to the specified the value.
- floor truncates to the lower integer value.
- imag returns the imaginary part of a complex value
- ln returns the natural logarithm of the value
- log returns the logarithm in base of the first parameter form the second parameter.
- log10 returns the logarithm in base 10 of the value.
- max returns the maximum value of the parameters.
- min returns the minimum value of the parameters.
- mod returns the module of a complex value.
- nat10 returns the natural value of a squared magnitude. Returns a complex value if the input is complex and the module of the natural complex if the input is real.
- nat20 returns the natural value of a lineal magnitude. Returns a complex value if the input is complex and the module of the natural complex if the input is real.
- phase returns the phase value from a complex value in natural units.
- pow returns the power of the first argument raised to second argument.
- rad returns the value in radians from a value in degrees.
- random returns a value between 0.0 and 1.0.
- rcs returns the Radar Cross Section value from the value.
- real returns the real part of a complex value.
- round returns the nearest integer value of the input value.
- sin returns the sine of the value introduced in degrees.

  - sinr returns the sine of the value introduced in radians.
  - sqrt returns the square root of the value.
  - tan returns the tangent of the value introduced in degrees.
  - tanr returns the tangent of the value introduced in radians.
- Operation: "Apply" button this button allows the user to apply the selected operation to the represented trace. There are two possibilities to apply the operations:
  - If the user selects some text in the trace text before clicking on this button, the operation will be applied over the selected text.
  - If the user doesn't select some text, the operation will be applied over the Result Type option and will be written at the cursor position of the trace text.
- Constant: this list allows the user to select a constant to be added to the trace formula. The list contains all the constants defined. The constants are:
  - E: adds the value of 'e' number.
  - PI: adds the value of 'PI'.
  - J: adds the value of the (0,1) complex number.
  - SubJ: adds the value of the (0,-1) complex number.
- Constant "Apply" button: this button allows the user to apply the selected constant to the trace formula. The constant will be applied at the cursor position of the trace text.

3.4 Trace Results Output

This section explains the output results returned by the selected options on "Simulation Results", "Results Type" and "Results Options" combo boxes of "Trace" window.

Table 1: Far Field and Radiation Pattern Results

Simulation Results	Results Type	Results Options	Output
FarField / RadiationPattern	Lineal	ETheta	Complex natural of ETheta Field.
FarField / RadiationPattern	Lineal	EPhi	Complex natural of EPhi Field.
FarField / RadiationPattern	Lineal	ETotal	Complex natural of ETotal Field.
FarField / RadiationPattern	Lineal	Ex	Complex natural of Ex Field.
FarField / RadiationPattern	Lineal	Ey	Complex natural of Ey Field.
FarField / RadiationPattern	Lineal	ExEyTotal	Complex natural of ExEyTotal Field.
FarField / RadiationPattern	Circular	RHCP	Complex natural of RHCP Field.
FarField / RadiationPattern	Circular	LHCP	Complex natural of LHCP Field.
FarField / RadiationPattern	Circular	CircularTotal	Complex natural of CircularTotal Field.
FarField / RadiationPattern	Circular	AxialRatio	Axial Ratio value in dB.
FarField / RadiationPattern	+/-45	+45	Complex natural of +45 Field.
FarField / RadiationPattern	+/-45	-45	Complex natural of -45 Field.
FarField / RadiationPattern	3er Ludwing	3X_Co-Polar	Complex natural of 3X_Co-Polar Field.

Simulation Results	Results Type	Results Options	Output
FarField / RadiationPattern	3er Ludwing	3X_Cross-Polar	Complex natural of 3X_Cross-Polar Field.
FarField / RadiationPattern	3er Ludwing	3Y_Co-Polar	Complex natural of 3Y_Co-Polar Field.
FarField / RadiationPattern	3er Ludwing	3Y_Cross-Polar	Complex natural of 3Y_Cross-Polar Field.
FarField / RadiationPattern	Gain	ETheta	Complex natural of ETheta Power.
FarField / RadiationPattern	Gain	EPhi	Complex natural of EPhi Power.
FarField / RadiationPattern	Gain	ETotal	Total Power value in dBi.

Table 2: Far Field Results (PO Module)

Simulation Results	Results Type	Results Options	Output
FarField	VV	<<Effects>>	Complex natural of VV Field for selected effect.
FarField	HV	<<Effects>>	Complex natural of HV Field for selected effect.
FarField	VH	<<Effects>>	Complex natural of VH Field for selected effect.
FarField	HH	<<Effects>>	Complex natural of HH Field for selected effect.
FarField	VTotal	<<Effects>>	Complex natural of VTotal Field for selected effect.
FarField	HTotal	<<Effects>>	Complex natural of HTotal Field for selected effect.

<<Effects>> Can be "Total", "Simple Reflection", "Double Reflection", "Triple Reflection" or "Diffraction"

Table 3: Near Field Results

Simulation Results	Results Type	Results Options	Output
NearField	Ex	<<Type>>	Complex natural of Ex Field of selected type.
NearField	Ey	<<Type>>	Complex natural of Ey Field of selected type.
NearField	Ez	<<Type>>	Complex natural of Ez Field of selected type.
NearField	ETotal	<<Type>>	Complex natural of ETotal Field of selected type.
NearField	Hx	<<Type>>	Complex natural of Hx Field of selected type.
NearField	Hy	<<Type>>	Complex natural of Hy Field of selected type.
NearField	Hz	<<Type>>	Complex natural of Hz Field of selected type.
NearField	HTotal	<<Type>>	Complex natural of HTotal Field of selected type.

<<Type>> Can be "Scatter", "Direct" or "Total"

Table 4: Current Density Results

Simulation Results	Results Type	Results Options	Output
Current Density	X	-	Natural value of X current value.
Current Density	Y	-	Natural value of Y current value.
Current Density	Z	-	Natural value of Z current value.
Current Density	Total	-	Natural value of Total current value.

Table 5: Charge Density Results

Simulation Results	Results Type	Results Options	Output
Charge Density	Density	-	Natural value of charge value.

Table 6: RCS (MONCROS Module)

Simulation Results	Results Type	Results Options	Output
RCS	ETheta	-	Complex natural of ETheta Field.
RCS	EPhi	-	Complex natural of EPhi Field.
RCS	ETotal	-	Complex natural of ETotal Field.

3.5 Examples

In this section some basic examples of the use of the post-process functionality are shown. For the examples a case of the MOM Module is considered. Obtain the radiation of four planes of 250mm per side centred at points (0,0,0), (0,0,0.25), (0,0,5.25) and (0,0,5.5), on mm units, at 5.0 GHz, using a plane wave source with default parameters and the default solver parameters. The are two 'Observation Points' defined two planes of 100x100 mm centred at (0 0 -0.5) and (0,0,6), on mm units and with 11x11 points.

To see more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

3.5.1 Example 1: Table Trace

Step 1: Select "Table" option on "Post-Process" window.

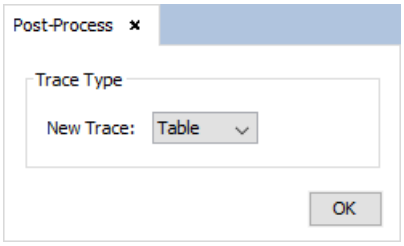


Figure 276: Step 1

Step 2: Select the simulation results and the type and sub-type from it.

Step 3: Select the domain and apply the desired operations over the original results.

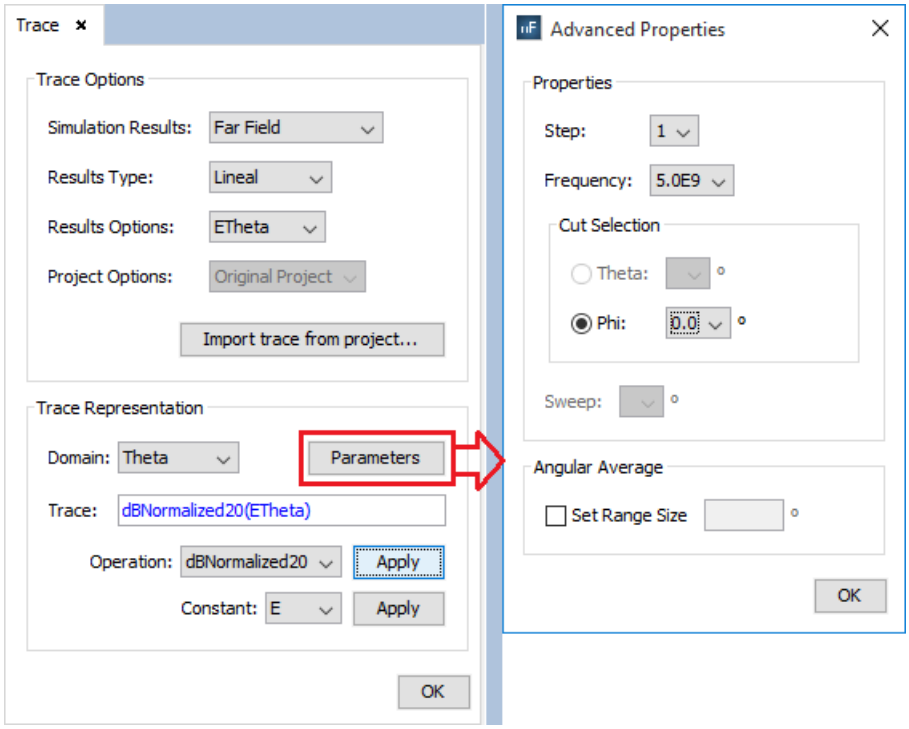
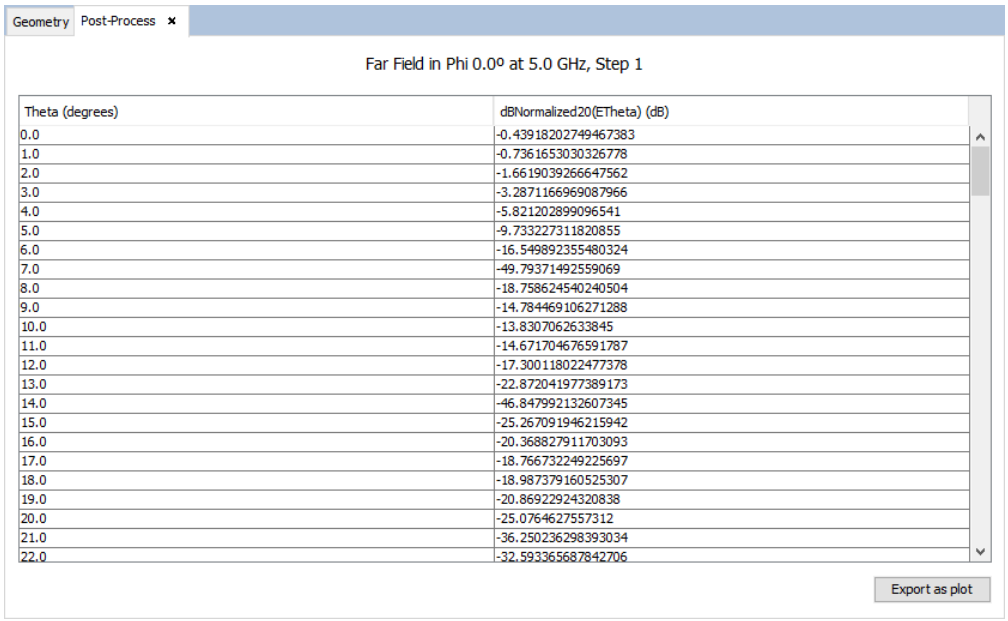


Figure 277: Step 2 and Step 3

Step 4: Click on "OK" and visualize the result.



Theta (degrees)	dBNormalized20(ETheta) (dB)
0.0	-0.43918202749467383
1.0	-0.7361653030326778
2.0	-1.6619039266647562
3.0	-3.2871166969087966
4.0	-5.821202899096541
5.0	-9.733227311820855
6.0	-16.549892355480324
7.0	-49.79371492559069
8.0	-18.758624540240504
9.0	-14.784469106271288
10.0	-13.8307062633845
11.0	-14.671704676591787
12.0	-17.300118022477378
13.0	-22.872041977389173
14.0	-46.847992132607345
15.0	-25.267091946215942
16.0	-20.368827911703093
17.0	-18.766732249225697
18.0	-18.987379160525307
19.0	-20.86922924320838
20.0	-25.0764627557312
21.0	-36.250236298393034
22.0	-32.593365687842706

Figure 278: Step 4

3.5.2 Example 2: Plot Trace

Step 1: Select "Plot" from the "Post-Process" window.

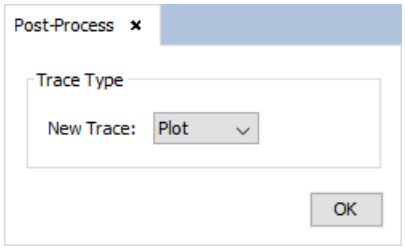


Figure 279: Step 1

Step 2: Select the simulation results and the type and sub-type from it.

Step 3: Select the domain and apply the desired operations over the original results.

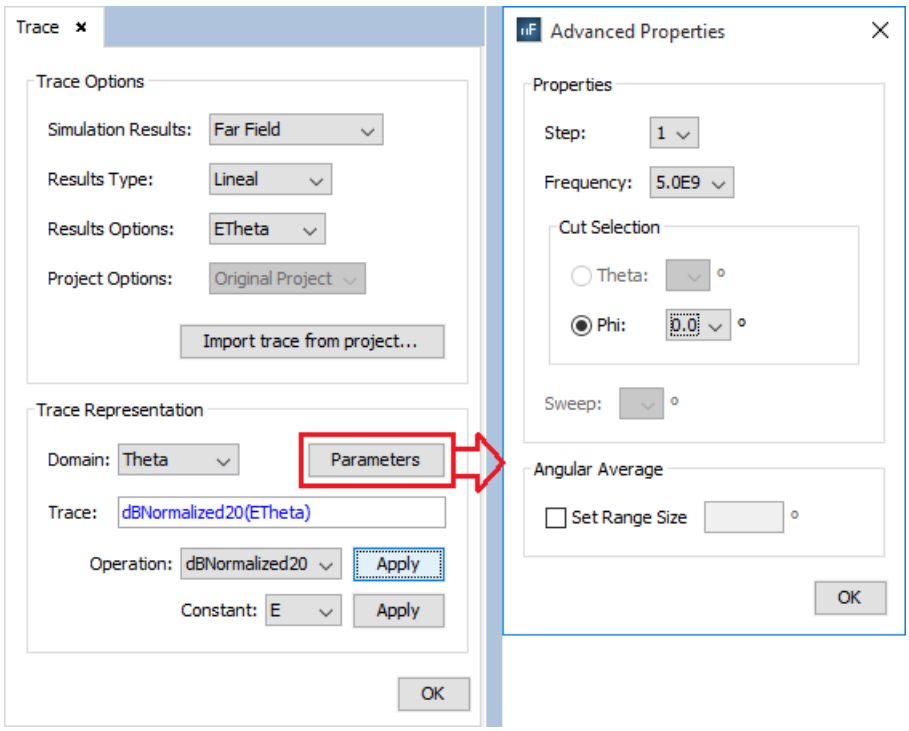


Figure 280: Step 2 and Step 3

Step 4: Click on "OK" button and visualize the result.

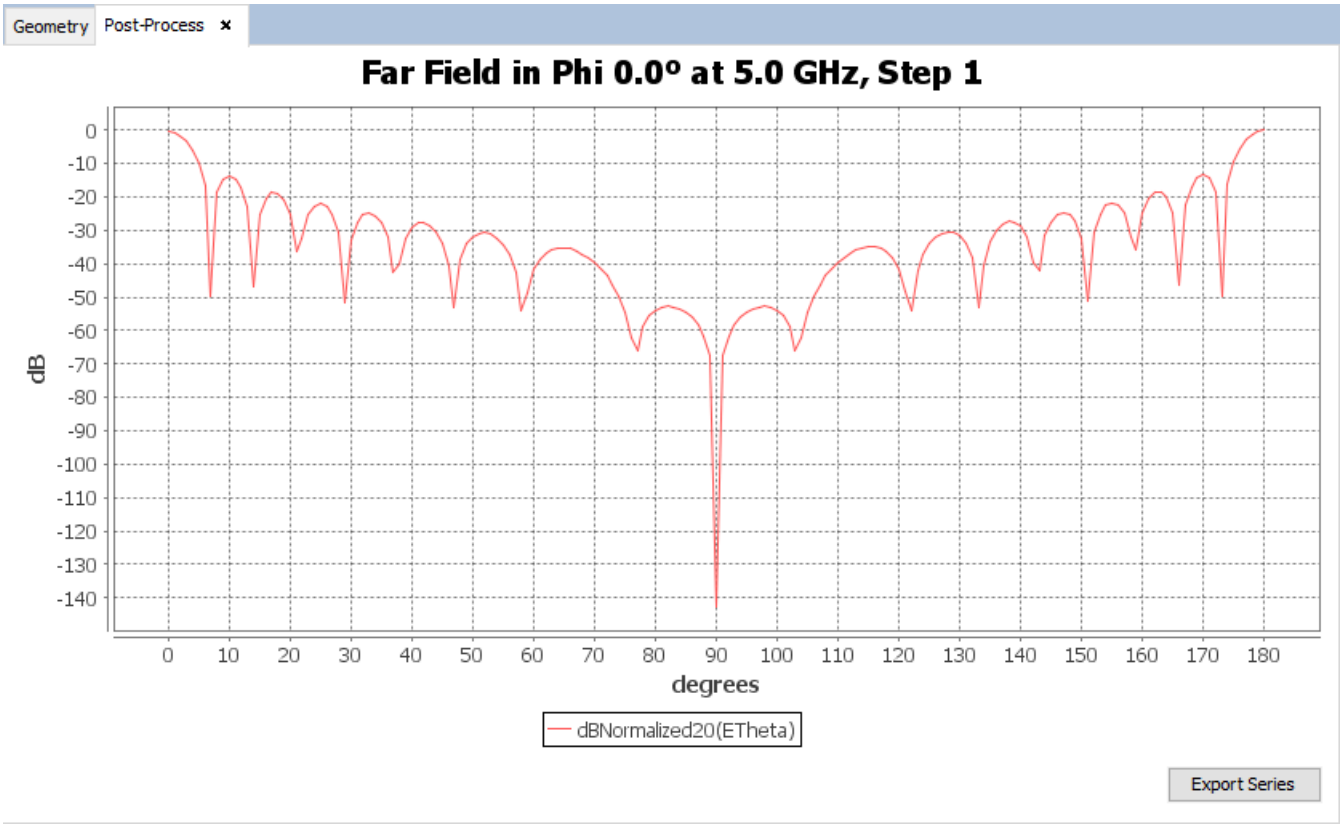


Figure 281: Step 4

3.5.3 Example 3: Diagram 2D Trace

Step 1: Select the "Diagram" option on the "Post-Process" window.

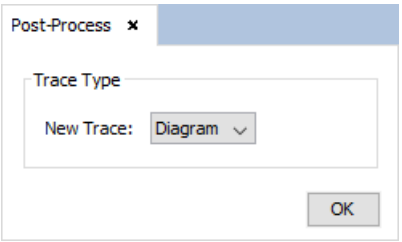


Figure 282: Step 1

Step 2: Select the simulation results and the type and sub-type from it.

Step 3: Select the domain and apply the desired operations over the original results.

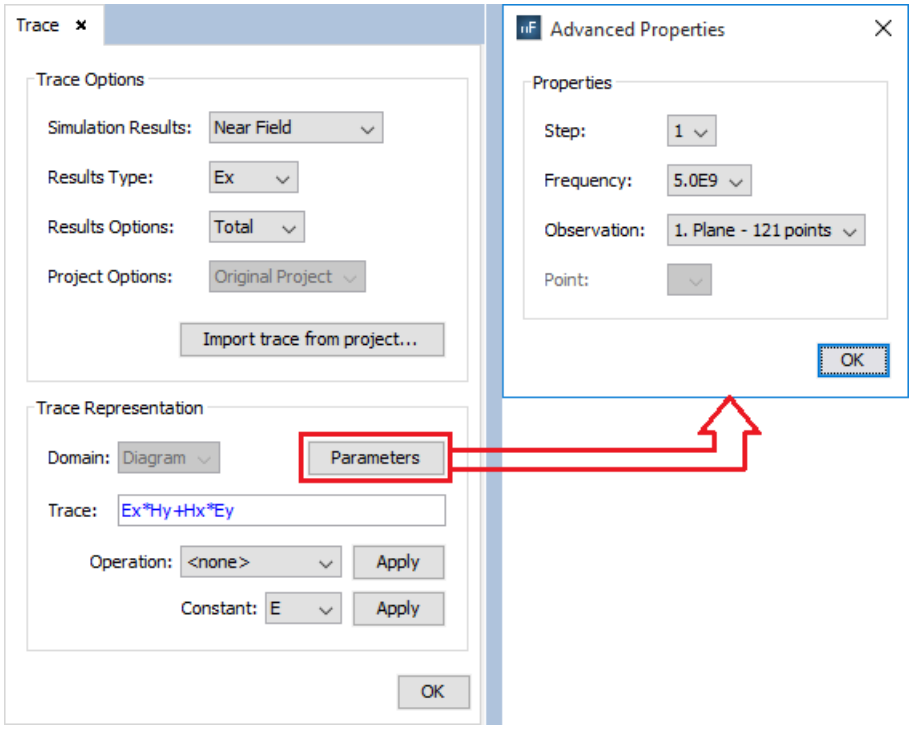


Figure 283: Step 2 and Step 3

Step 4: Click on "OK" button and visualize the result.

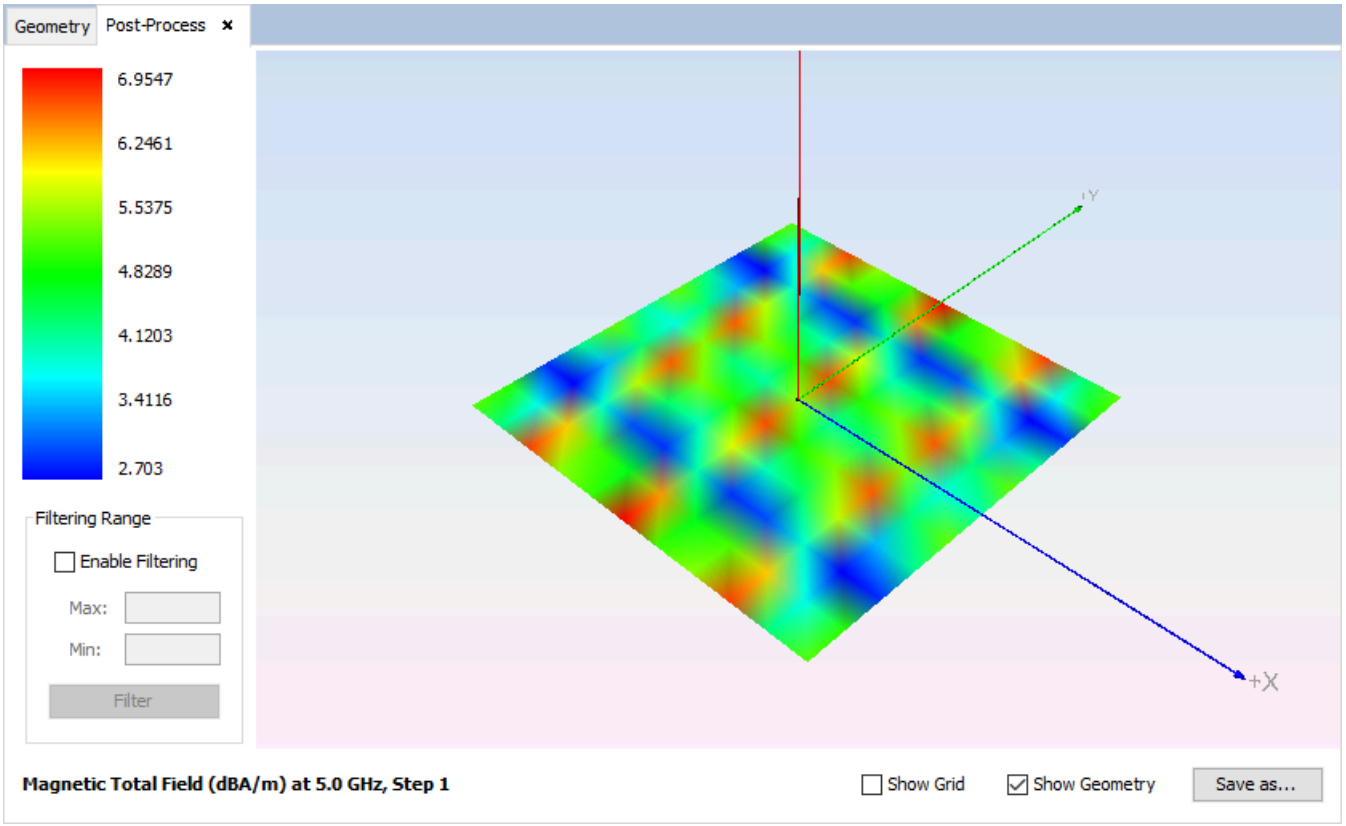


Figure 284: Step 4

3.5.4 Example 4: Diagram 3D Trace

Step 1: Select the "Diagram" option on "Post-Process" window.

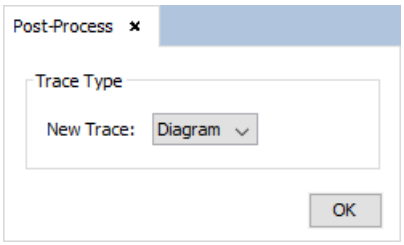


Figure 285: Step 1

Step 2: Select the simulation results and the type and sub-type from it.

Step 3: Select the domain and apply the desired operations over the original results.

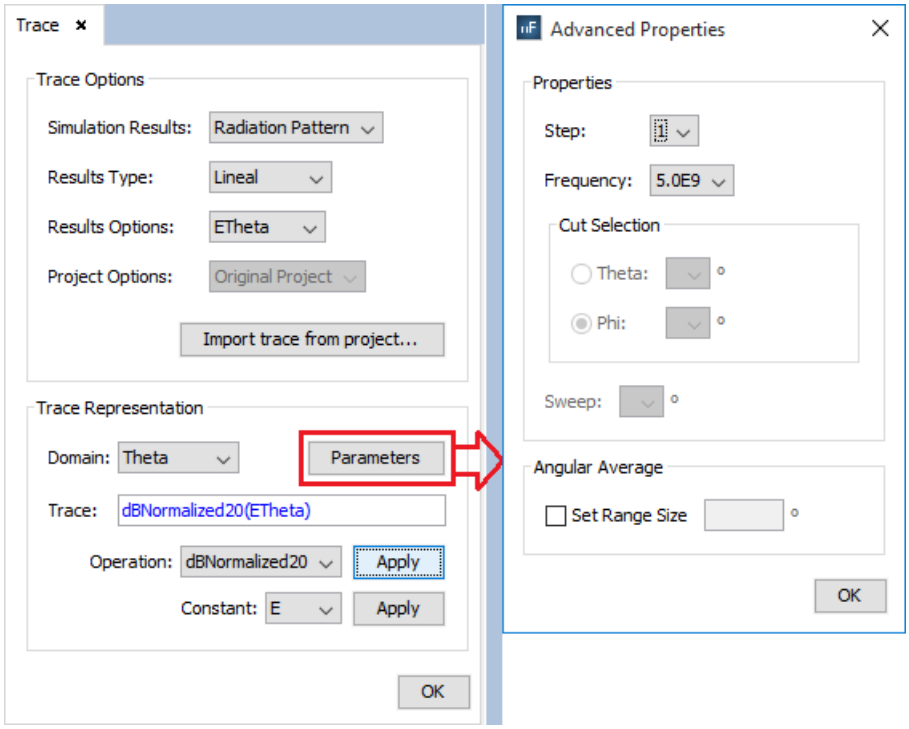


Figure 286: Step 2 and Step 3

Step 4: Click on "OK" and visualize the result.

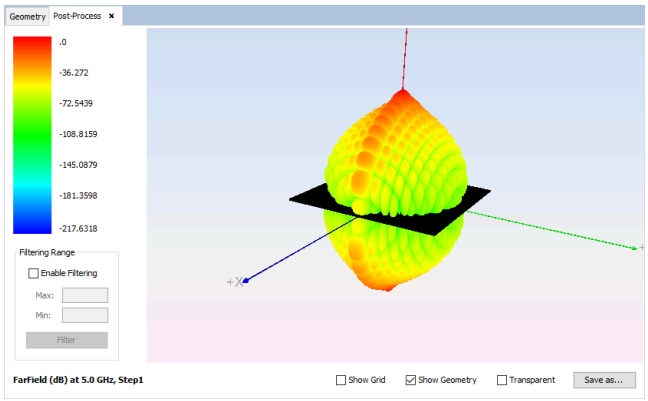


Figure 287: Step 4

Antennas

4

A large collection of antennas are supported as well as importing radiation patterns and specifying the antenna feeding.

This chapter covers the following:

- 4.1 Plane Wave (p. 295)
- 4.2 Dipole (p. 297)
- 4.3 Pattern File (p. 305)
- 4.4 Equivalent Sources (p. 311)
- 4.5 Coaxial Feed (p. 318)
- 4.6 Waveguides (p. 319)
- 4.7 Primitive Antenna (p. 321)
- 4.8 Radome (p. 358)
- 4.9 Reflectors (p. 366)
- 4.10 Reflectarrays (p. 377)
- 4.11 Import Pattern File (p. 380)
- 4.12 Edit Antenna (p. 381)
- 4.13 Delete Antenna (p. 382)
- 4.14 Antenna Size (p. 383)
- 4.15 Antenna Feeding (p. 384)

4.1 Plane Wave

One possible kind of excitation source is the plane wave. If enabled, a plane wave with the parameters described by the user will be simulated. When a plane wave is being defined, a panel similar to the one displayed in the following figure is presented. The user can modify the values for the polarization wave and the incidence angle, represented using spherical coordinates.

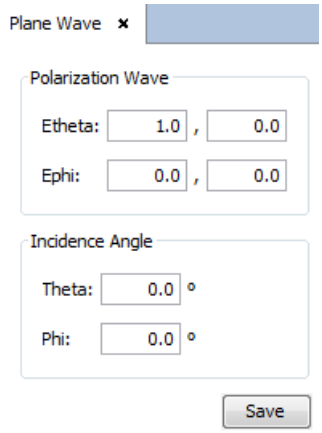


Figure 288: Plane wave parameters

A plane wave has the following properties:

- Polarization Wave this value specifies the polarization for the incident field of the plane wave.
- Incidence Angle this value specifies the incidence direction for the plane wave.
- Monostatic this check box enables the Monostatic mode, causing the values set in Incidence Angle to be ignored in order to use the range of angles specified in Observation Directions.

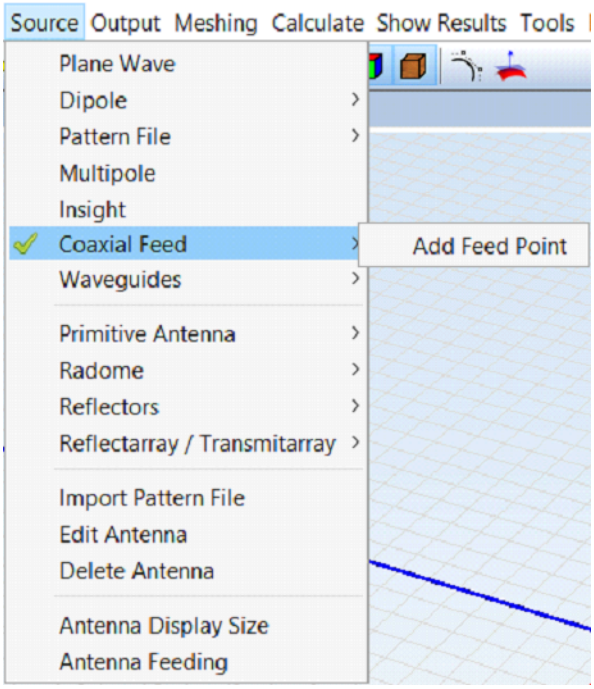


Figure 289: Adding a Coaxial feeding point

To add the Observation Directions, the user must select Obervation Directions in the Output dropdown.

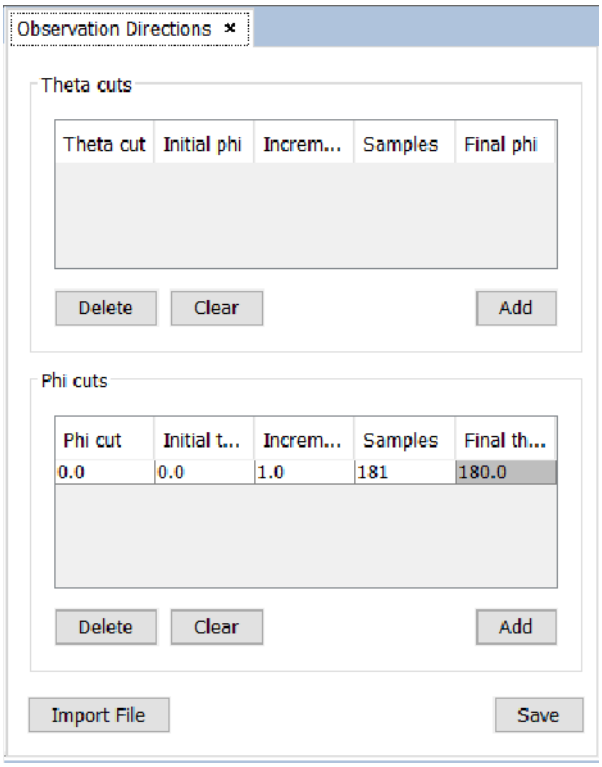


Figure 290: Observation directions parameters

4.2 Dipole

This menu allows the user to create Dipole antennas.

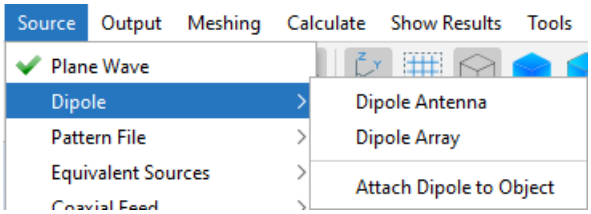


Figure 291: Dipoles Menu

The following menu items are available:

- **Dipole Antenna**: using this option the user is able to create a single dipole.
- **Dipole Array**: using this option the user is able to create an array of dipoles.
- **Attach Dipole to Object**: using this option lets the user to attach (associate) a pre-existent dipole to an object.

4.2.1 Dipole Antenna

Using the "Dipole Antenna" option, the user can define the parameters for a dipole antenna that will be added to the project and processed during simulation.

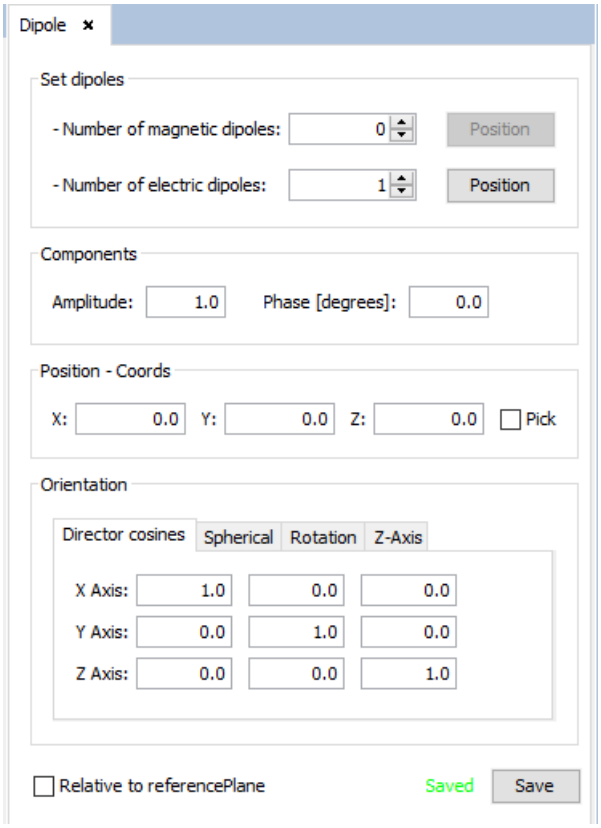


Figure 292: Dipole Antenna panel

The following options are available:

- Set dipoles this setting specifies how many magnetic and electric dipoles have to be generated. This feature is covered later in this section.
  - **Number of magnetic dipoles**: sets the number of magnetic dipoles that the generated dipole will have.
  - **Number of electric dipoles**: sets the number of electric dipoles that the generated dipole will have.
- **Components**: the amplitude and phase for the dipole moment of the generated antenna.
- **Position**: the Cartesian center for the generated antenna. For a more precise mode, the pick-point mode can be enabled. Select Pick, then click anywhere in the 3D Panel. The position fields will be filled with the coordinates of the clicked point.
- **Orientation**: dipole antenna rotation. There are multiple ways of setting the orientation.
  - **Director Cosines**: manually set the director cosines for the dipole.
  - **Spherical**: set the orientation using spherical system (theta, phi, alpha).
  - **Rotation**: specify X, Y, and Z angles in degrees.
  - **Z-Axis**: this option can be used for manually defining the local Z-axis of the dipole.
- **Relative to reference plane**: when this option is set, the given coordinates and rotations are considered relative to the coordinate system defined by the current reference plane. If this option is unchecked, the given position and rotation are absolute.

When the number of magnetic dipoles is greater than 0, the position and orientation of magnetic dipoles can be selected by pressing the Position button.

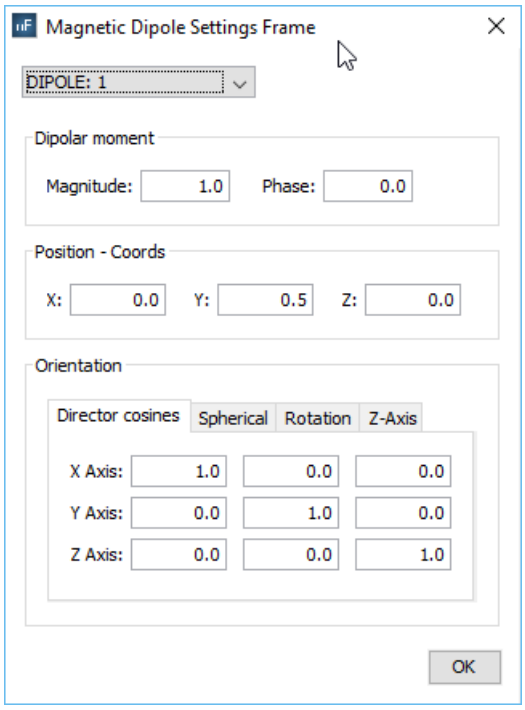


Figure 293: Magnetic Dipoles

The selector at the top of the window lists every magnetic dipole defined in the antenna. The following settings can be defined:

- **Dipolar moment**: values for the magnetic dipole moment, magnitude, and phase, can be set here.
- **Position**: Cartesian coordinates for the magnetic dipole, relative to the location of the antenna.
- **Orientation**: similar to the rotation controls under Dipole tab, this setting lets the user set the rotation for this magnetic dipole using either Director cosines, Spherical, Rotation or Z-Axis values.

Similarly, when the number of electric dipoles is higher than 0, the position and rotation for every electric dipole can be set by pressing the Position button.

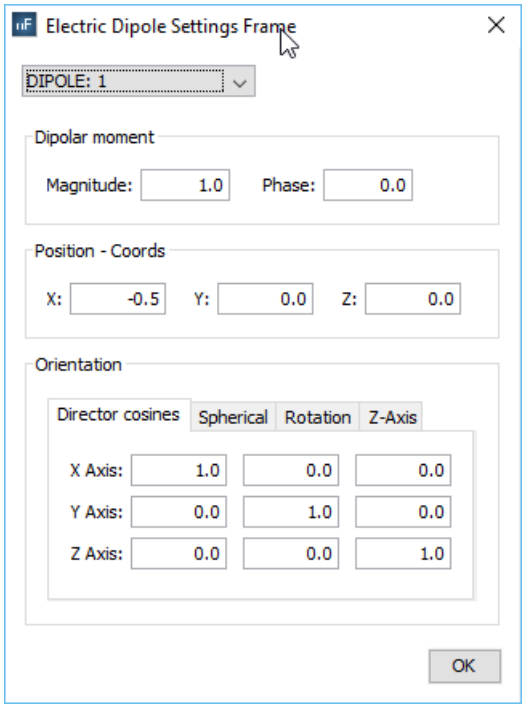


Figure 294: Electric Dipoles

The behavior of this dialog is similar to the one for setting magnetic dipoles.

After every possible setting has been set, press Save to create the Dipole. When generated, the dipole is displayed as a set of red cones. The bigger one is the antenna itself. Every another small dipole represents each magnetic and electric dipole associated with the antenna.

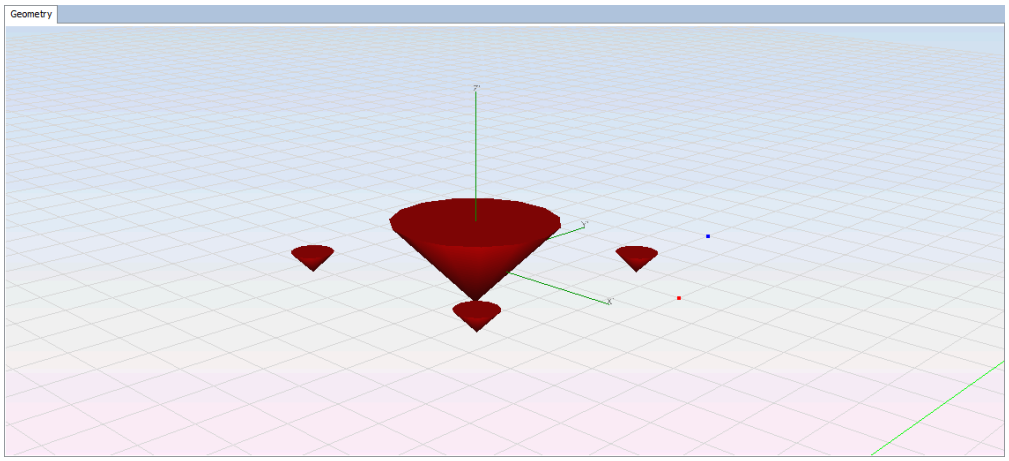


Figure 295: Dipole Antenna visualization with 2 electrical dipoles and 2 magnetic dipoles

4.2.2 Dipole Array

The "Dipole Array" operation is a faster way for generating multiple dipole antennas when they are located in an array structure, where a set of closely located dipoles should be simulated. To create a Dipole Array, select a Dipole Array source to display the Dipole Array panel, as shown in the figure:

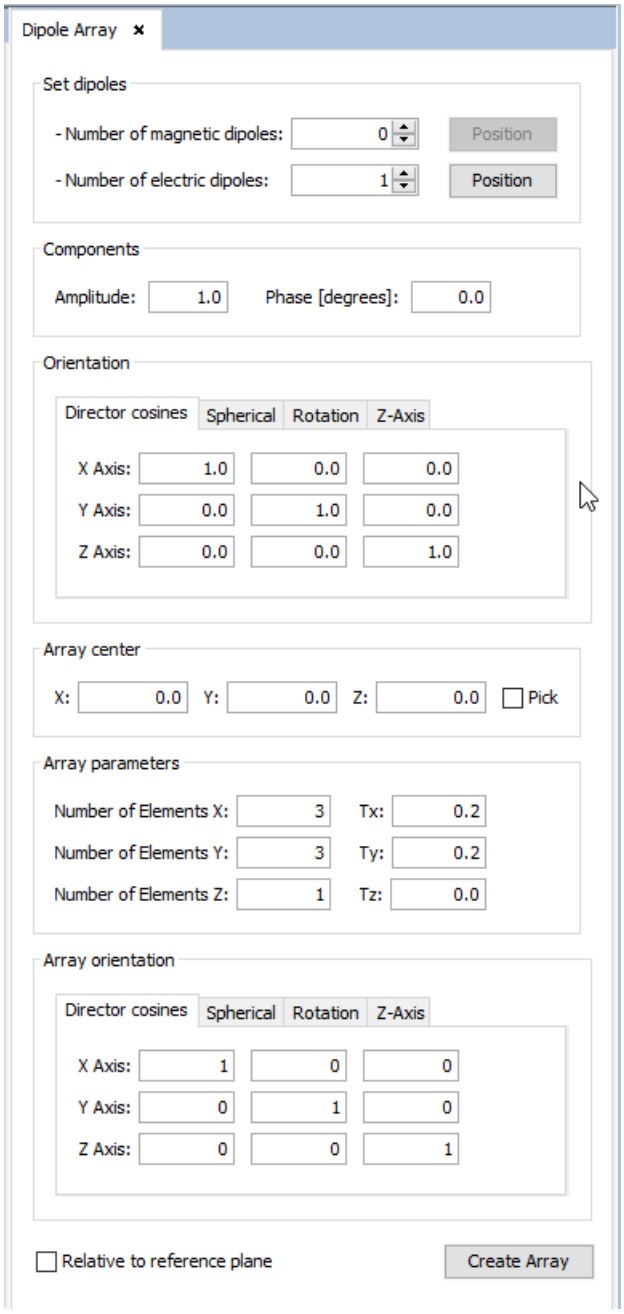


Figure 296: Dipole Array Panel

The following settings are available for defining the parameters of the overall dipole structure.

- Set dipoles this setting specifies how many magnetic and electric dipoles have to be generated. This feature is covered later in this section.

  - **Number of magnetic dipoles**: set the number of magnetic dipoles that each one of the generated dipole will have.
  - **Number of electric dipoles**: set the number of electric dipoles that each one of the generated dipole will have.
- **Components**: the amplitude and phase for the dipole moment for each generated antenna.
- **Array orientation**: dipole antenna rotation. There are multiple ways for setting the orientation.
  - **Director cosines**: manually set the director cosines for the dipole.
  - **Spherical** set the orientation using spherical system (theta, phi, alpha).
  - **Rotation**: specify X, Y and Z angles as degrees.
  - **Z-Axis**: this option can be used for manually defining the local Z-axis of the dipole.
- **Array center**: the Cartesian center for the generated dipole array structure. For a more precise mode, pickpoint mode can be enabled. Select Pick, then click anywhere in the 3D Panel. The position fields will be filled with the coordinates of the clicked point.
- **Array parameters**: this setting lets the user specify the number of dipoles to be generated and the separation between each dipole.
- **Array orientation**: similar to the dipole orientation, this control lets the user specify the array rotation.

When the number of magnetic dipoles is greater than 0, the position and orientation of magnetic dipoles can be selected by pressing the Position button.

When the number of magnetic dipoles is greater than 0, the position and orientation of magnetic dipoles can be selected by pressing the Position button.

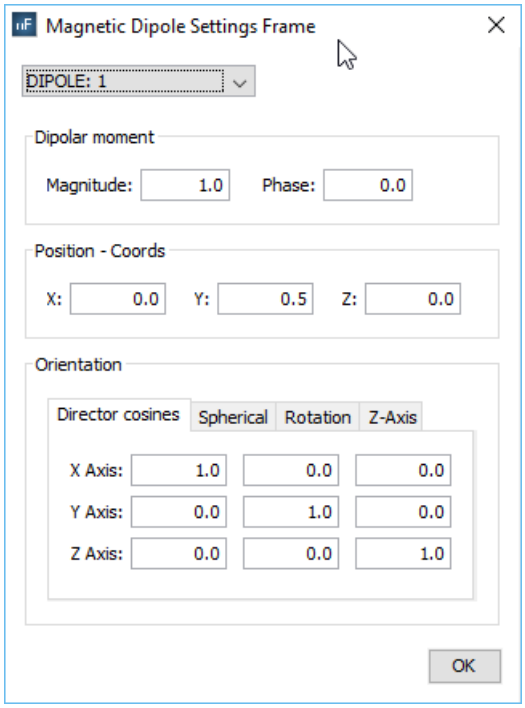


Figure 297: Magnetic Dipoles

The selector at the top of the window lists every possible magnetic dipole defined in the antenna. The following settings can be defined:

- **Dipolar moment**: values for the magnetic dipole moment, magnitude and phase, can be set here.
- **Position**: Cartesian coordinates for the magnetic dipole, relative to the location of the antenna.
- **Orientation**: similar to the rotation controls under Dipole tab, this setting lets the user set the rotation for this magnetic dipole using either Director cosines, Spherical, Rotation or Z-Axis values.

Similarly, when the number of electric dipoles is higher than 0, the position and rotation for every electric dipole can be set by pressing the Position button.

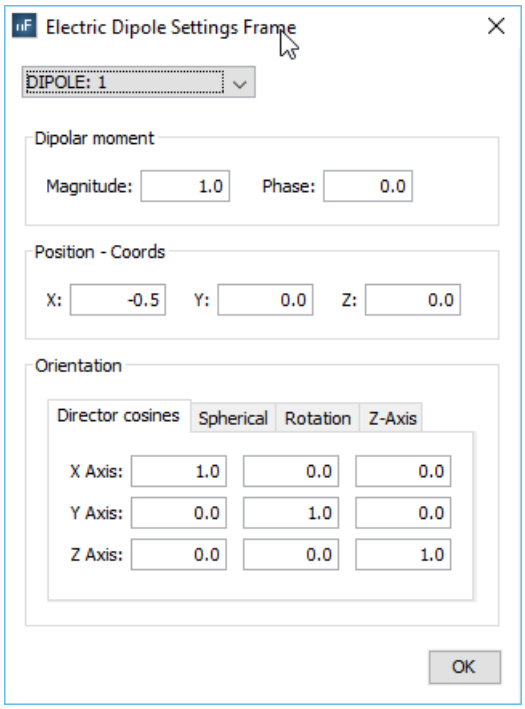


Figure 298: Electric Dipoles

The behaviour of this dialog is similar to the one for setting magnetic dipoles.

When all the parameters have been set, press Create Array to create the dipoles. Note that the dipole generation will create separate dipoles. This means that after generating the dipole array, you will be able to modify the values for each dipole separately.

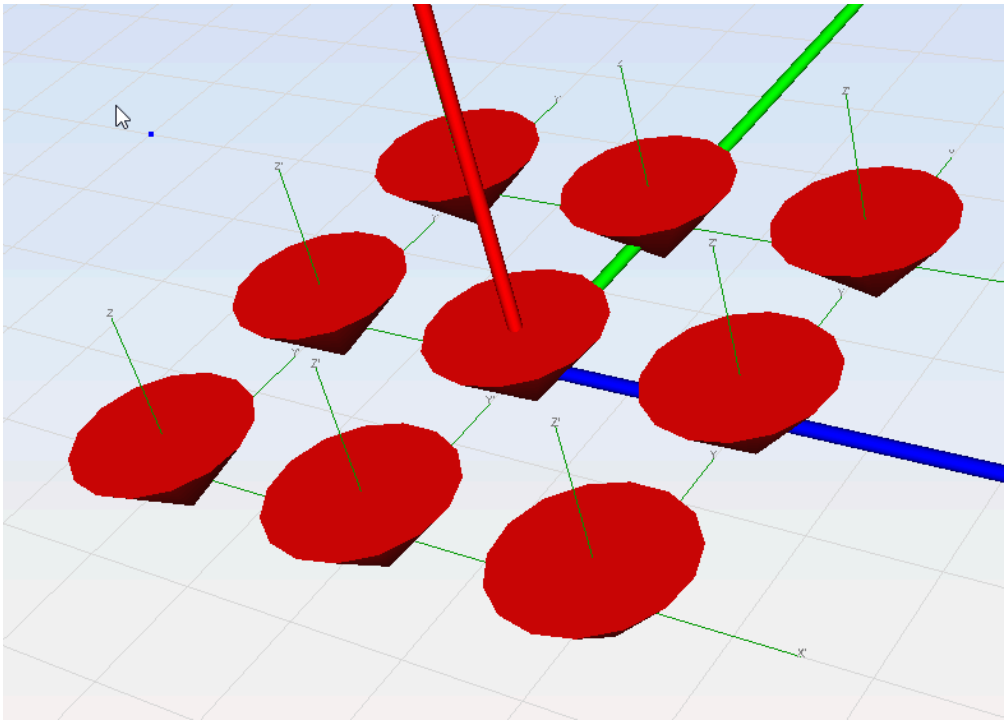


Figure 299: Generated dipole array

4.2.3 Attach Dipole to Object

The "Attach Dipole to Object" lets the user to anchor one or more dipoles to the current position of an object, refreshing the dipole position whenever the object is moved.

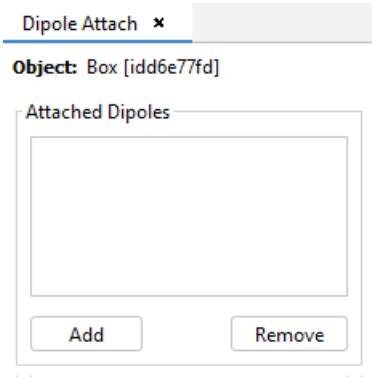


Figure 300: Dipole Attach Panel

In order to attach the dipoles to any object, the user must create them first. After doing so, selecting the desired object in the Geometry Panel or Tree and clicking in Source → Dipole → Attach Dipole to Object will take us to the proper panel. Here the user can select the desired dipole and clicking the Add button will attach that dipole to the specified object. The Remove button let the user detach an attached dipole without deleting the dipole.

4.3 Pattern File

This menu allows the user to create Pattern File antennas.

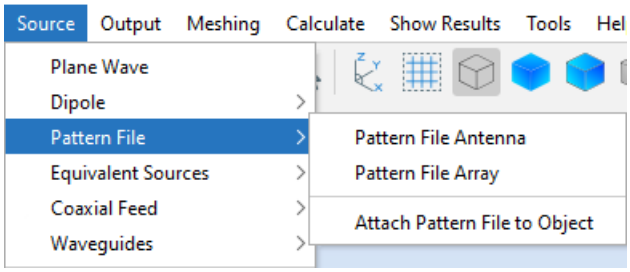


Figure 301: Dipoles Menu

The following menu items are available:

- **Pattern File Antenna**: using this option the user is able to create a single pattern file antenna.
- **Pattern File Array**: using this option the user is able to create an array of pattern file antennas.
- **Attach Pattern File to Object**: using this option lets the user to attach (associate) a pre-existent Pattern File to an object.

However, before using any of these options, the user needs to import at least one radiation pattern file. This is covered in the appropriate chapter later in this guide.

4.3.1 Pattern File Antenna

The Pattern File Antenna option is used to add pattern file antennas to the project.

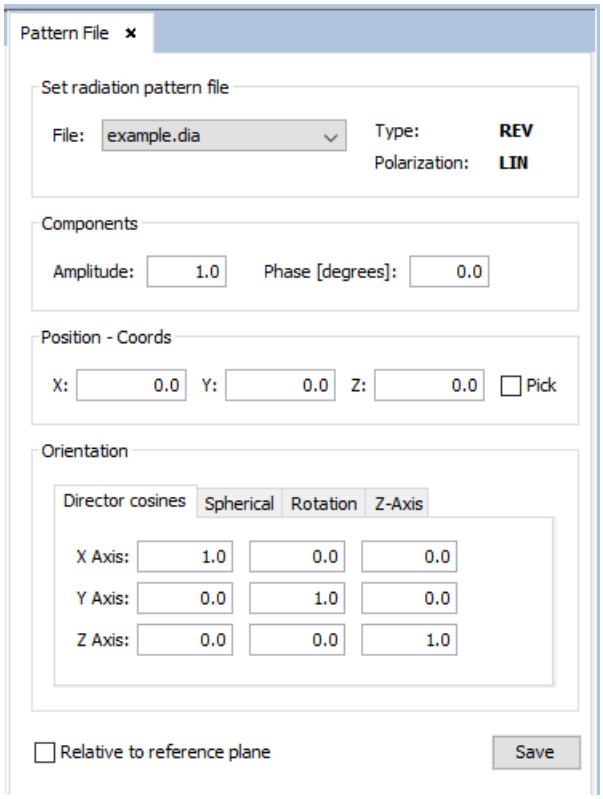


Figure 302: Pattern File Antenna panel

To create a radiation pattern antenna using a pattern file, select a previously imported Pattern File using the File drop-down menu (see Import Pattern File). The Type and Polarization labels will display information about the currently selected pattern file antenna. For a particular pattern file antenna, the following options are available:

- Components the amplitude and phase for the generated antenna.
- Position the Cartesian center for the generated antenna. For a more precise mode, the pick-point mode can be enabled. Select Pick, then click anywhere in the 3D Panel. The position fields will be filled with the coordinates of the clicked point.
- Orientation antenna rotation. There are multiple ways for setting the orientation.
  - Director Cosines manually set the director cosines for the dipole.
  - Spherical set the orientation using spherical system (theta, phi, alpha).
  - Rotation specify X, Y, and Z angles as degrees.
  - Z-Axis this option can be used for manually defining the local Z-axis of the dipole.
- Relative to reference plane when this option is set, given coordinates and rotations are considered relative to the current relative plane. If this option is unchecked, it is assumed an unchanged relative plane.

Press Save to add the Pattern File to the project. It will be represented as a brown cone in the geometry view.

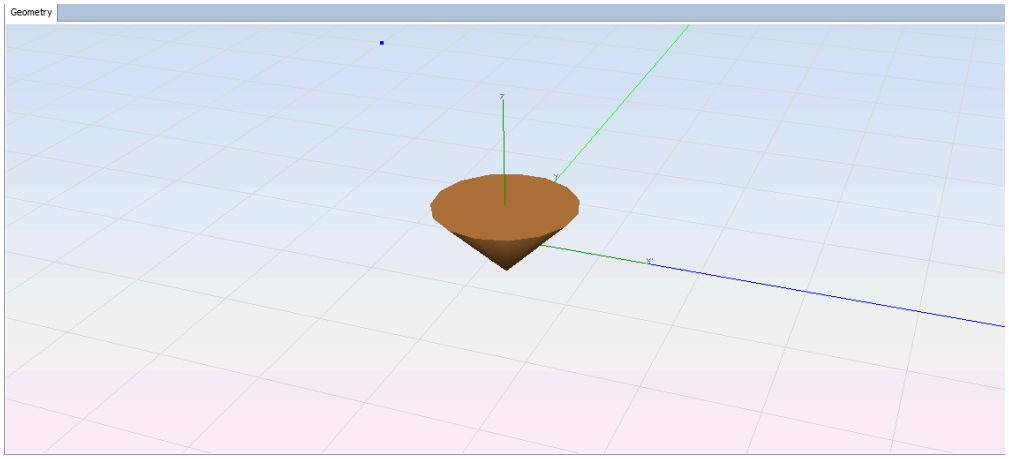


Figure 303: Pattern File Antenna added to the project

4.3.2 Pattern File Array

Using the "Pattern File Array" option, it is possible to create arrays of radiation pattern antennas defined based on information described in a radiation pattern file. This is useful when an array of several antennas has to be created and added to the project. When the Pattern File Array option is selected, the following panel appears.

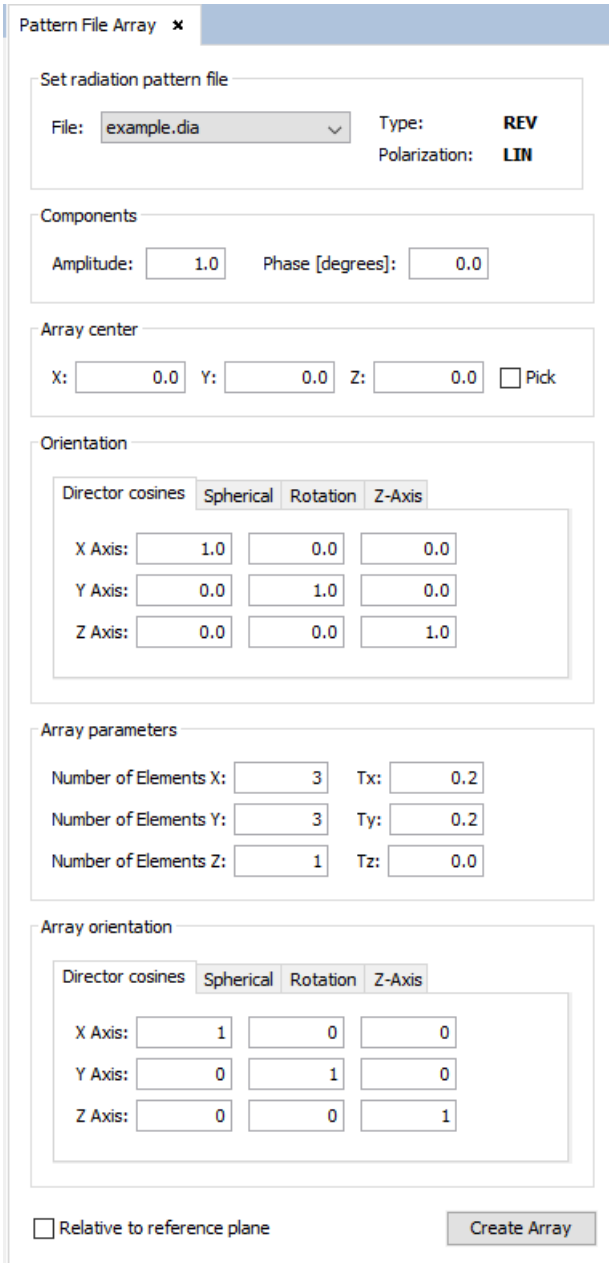


Figure 304: Pattern File Array Tab

The following options are available:

- Components the amplitude and phase for each generated antenna.
- Orientation antenna rotation. There are multiple ways for setting the orientation.
  - Director Cosines manually set the director cosines.
  - Spherical set the orientation using spherical system (theta, phi, alpha).
  - Rotation specify X, Y, and Z angles as degrees.
  - Z-Axis this option can be used for manually defining the local Z-axis.

- Array center the Cartesian center for the generated radiation pattern array structure. For a more precise mode, the pick-point mode can be enabled. Select Pick, then click anywhere in the 3D Panel. The position fields will be filled with the coordinates of the clicked point.
- Array parameters this setting lets the user specify the number of pattern files to be generated along each axis and the separation between each antenna.
- Array orientation similar to the pattern file orientation, this control lets the user specify the array rotation.

When all the parameters have been set, click the "Create Array" button to create the radiation pattern antennas. Note that the pattern file generation will create separate antennas. This means that after generating the pattern file array, you will be able to modify the values for each pattern file separately.

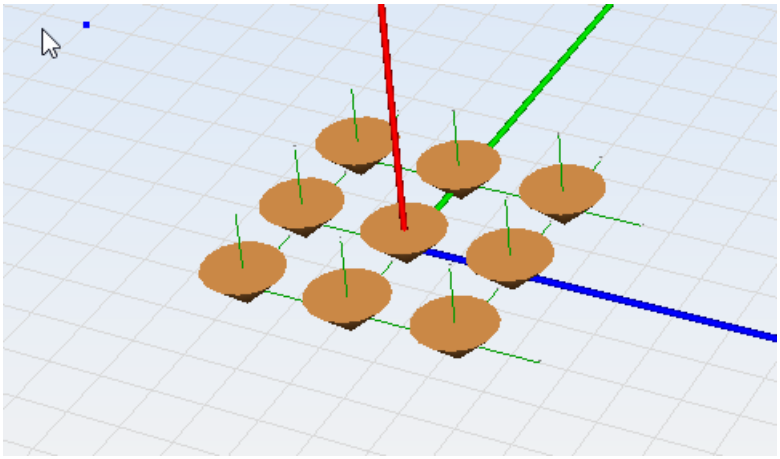


Figure 305: Radiation Pattern Array structure.

4.3.3 Attach Pattern File to Object

The Attach Pattern File to Object lets the user to anchor one or more Pattern Files to the current position of an object, refreshing the pattern position whenever the object is moved.

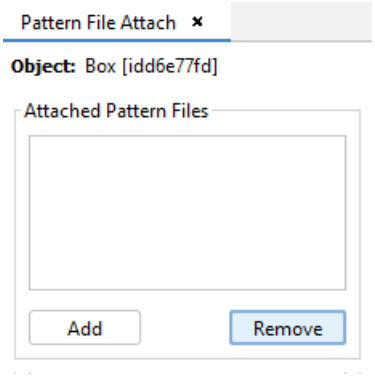


Figure 306: Pattern File Attach Panel

In order to attach any pattern file, they must be created and set as a valid source first. In order to do so, the user can import them via the Source menu, in Import Pattern File option. After doing that, the

file must be set as a valid source, thing that can be done in the Pattern File Antenna or Pattern File Array, in Source → Pattern File, specifying the `.dia` file (Pattern File) in the Set Radiation Pattern Panel, and clicking on the Save button.

When these requisites are fulfilled we can select the object which we want to attach Pattern Files, and clicking on the Source → Pattern File → Attach Pattern File to Object will take us to the Pattern File Attach Panel. Here by clicking on the desired Pattern Files on the Geometry Panel or Tree and clicking the Add button will successfully attach the pattern file to our object.

In the same manner we can detach any previously attached pattern file by selecting it and clicking the Remove button.

4.4 Equivalent Sources

4.4.1 Multipole Antenna

Multipole antennas are antennas that have multiple dipoles. When the antennas are close to other structures, the simulation using multipole antennas provides a higher accuracy than when using regular radiation pattern antennas, while avoiding the extra computational cost of a physical antenna. To create a multipole antenna, select "Multipole Antenna" from the Antenna menu.

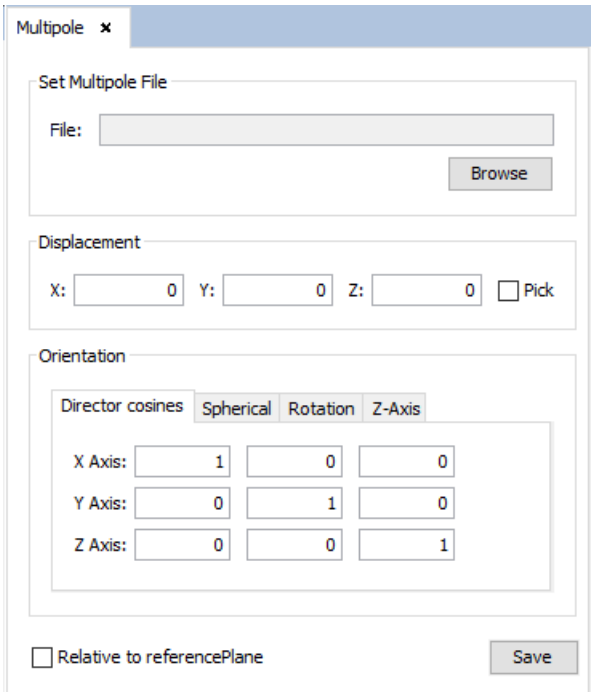


Figure 307: Multipole Antenna panel

To add a multipole antenna, a Multipole File has to be imported. These files usually have a .suj extension. Click Browse to locate on your hard drive the desired Multipole File to import.

The user can also specify a transformation to apply to the loaded multipole. The transformation consists on a displacement (given by a three coordinate vector) and a rotation (given by the X, Y and Z axes of the local coordinate system of the rotation). If the "Relative to reference plane" check box is selected, the entered displacement and rotation will be relative to the reference plane coordinate system. Pressing the **Save** button will add the multipole antenna.

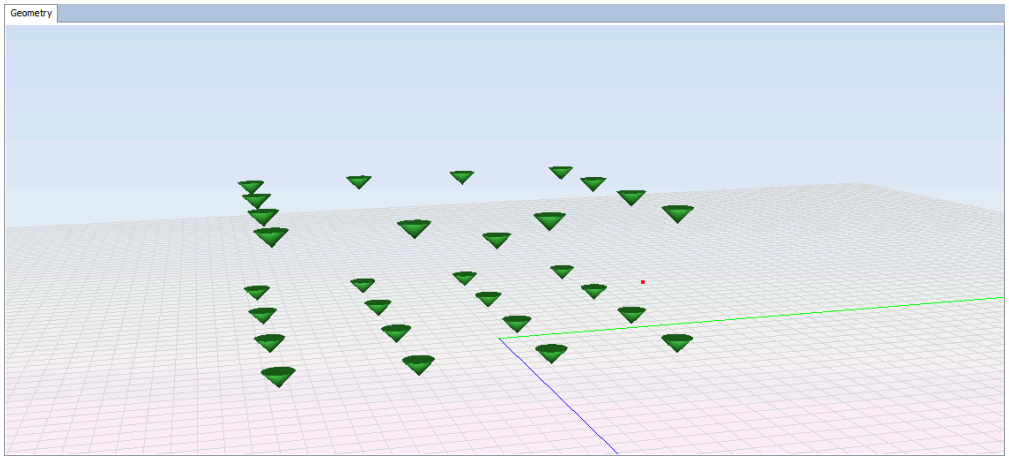


Figure 308: Multipole Antenna Visualization

4.4.2 Insight

When the "Insight" option is selected, the following panel is shown:

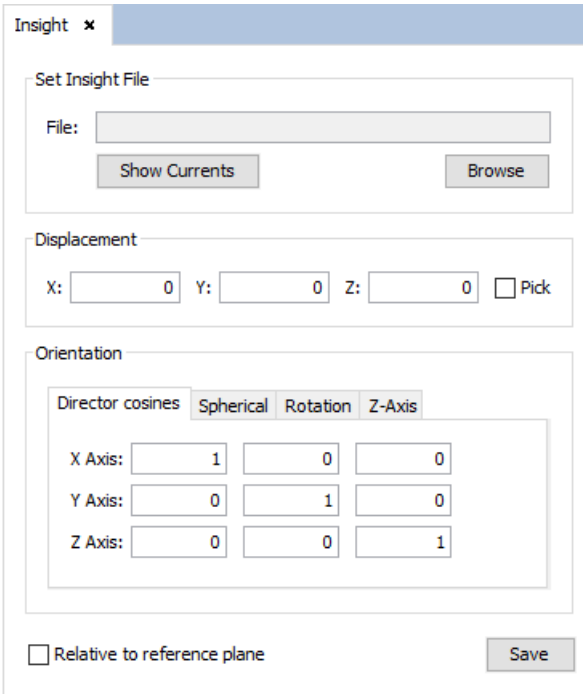


Figure 309: Insight panel

This panel allows the user to add a new Insight antenna. An insight antenna is a type of antenna that is defined by an Insight file (a file with a `.ins` extension). This file defines a geometry made from points, where each point is assigned a current density.

The user can import a INS file by pressing the "Browse" button and selecting the file in their hard drive. Before adding the Insight antenna to the simulation, the user needs to specify the following parameters:

- **Displacement**: This parameter is a vector that specifies the translation applied to the points of the insight file when added to the simulation.
- **Orientation**: This parameter allows the rotation of the points of the Insight file. There are several ways the user can specify this rotation by giving each of the axis directions of the transformation, by specifying spherical rotation angles, by specifying rotation angles on each axis or by specifying the Z axis and a rotation angle.

If the "Relative to reference plane" check box is checked, the displacement and rotation will be relative to the current reference plane coordinate system.

The user can also choose to view a 3D diagram of the current densities of the imported Insight file by pressing the "Show Currents" button:

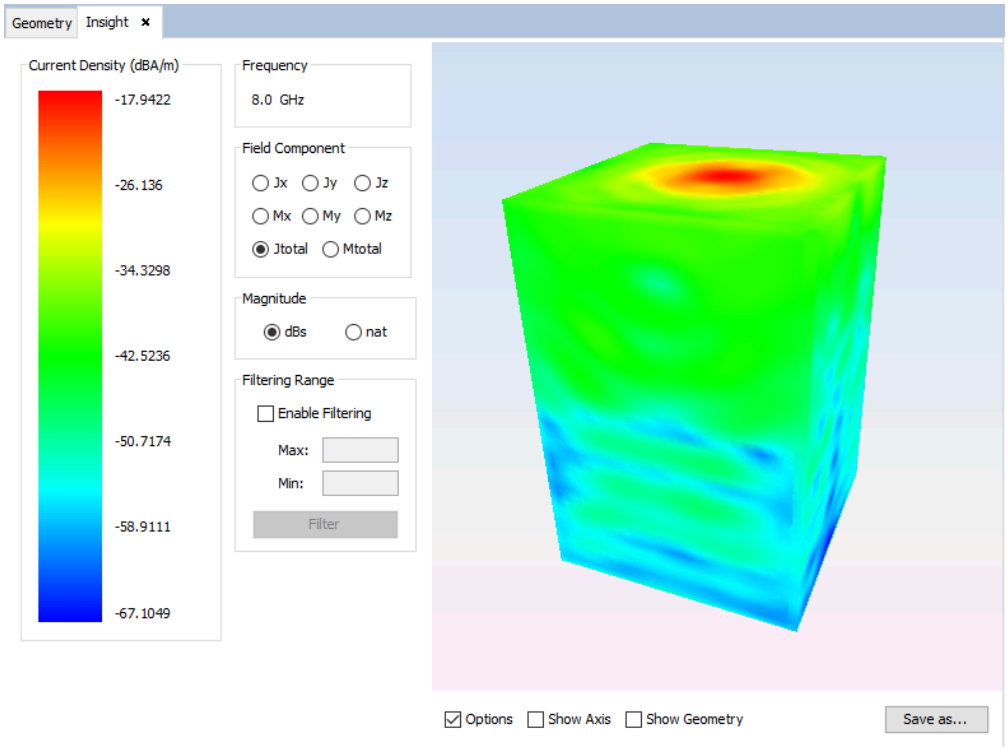


Figure 310: Insight currents panel

In this panel, the user can select the component of the current density that is displayed in the diagram, as well as choose the magnitude and filter the displayed values to be between a given range. The check boxes below the diagram allow the user to hide/show the left panel, show/hide the axes and show/hide the wireframe of the Insight antenna geometry. The user can also save the diagram as a PNG image file by pressing the "Save as..." button.

When the user has finished setting up the parameters of the Insight antenna, they need to press the "Save" button in order to actually add the antenna to the simulation.

4.4.3 Define Near Field from Data

Apply an array of electric and magnetic dipoles in the model (in the form of a planar aperture) that is equivalent to measured or calculated field values.

Source Files

The user can import the E-Field and H-Field files by pressing the **Browse** button and selecting the text files in their hard drive. These files contain the electric and magnetic field for every point in the U and V direction, four columns for the E-Field file with E_U (Amplitude V/m), E_U (Phase deg), E_V (Amplitude V/m), E_V (Phase deg); and four columns for the H-Field file with H_U (Amplitude V/m), H_U (Phase deg), H_V (Amplitude V/m), H_V (Phase deg) (see Figure 312).

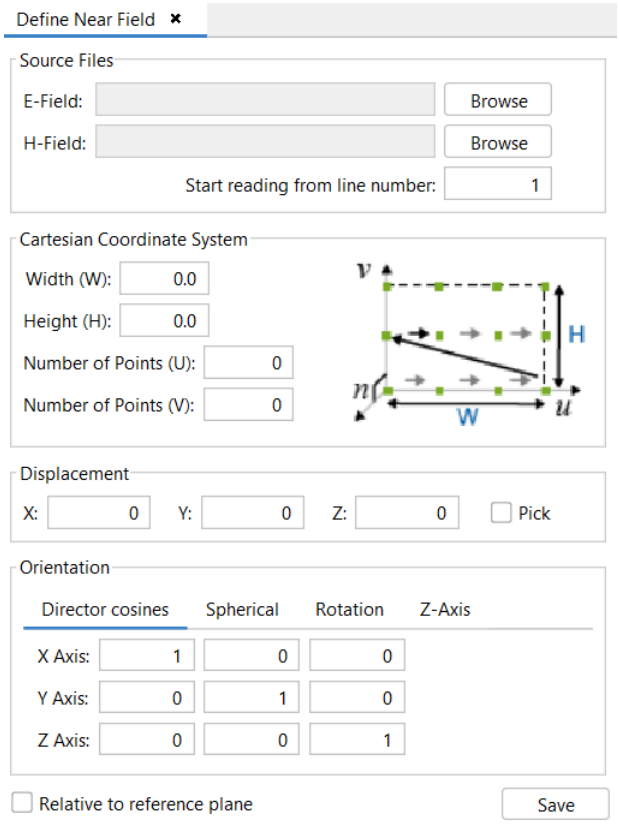


Figure 311: Define Near Field

- **Start reading from line number**: Skip this number of lines before reading the field values.

```
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.139647   -180.0000
0.000000      0.0000      0.139774   -180.0000
0.000000      0.0000      0.139955   -180.0000
0.000000      0.0000      0.140160   -180.0000
0.000000      0.0000      0.140364   -180.0000
0.000000      0.0000      0.140545   -180.0000
0.000000      0.0000      0.140687   -180.0000
0.000000      0.0000      0.140777   -180.0000
0.000000      0.0000      0.140808   -180.0000
0.000000      0.0000      0.140777   -180.0000
0.000000      0.0000      0.140687   -180.0000
0.000000      0.0000      0.140545   -180.0000
0.000000      0.0000      0.140364   -180.0000
0.000000      0.0000      0.140160   -180.0000
0.000000      0.0000      0.139955   -180.0000
0.000000      0.0000      0.139774   -180.0000
0.000000      0.0000      0.139647   -180.0000
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.000000      0.0000
0.000000      0.0000      0.000000      0.0000
```

Figure 312: E-Field file example

- **Cartesian Coordinate System**: Allows the user to define the planar aperture, the E-Field and H-Field files must contain a field value for each point defined in this section.
  - **Width (W)**: Array size in U direction.
  - **Height (H)**: Array size in V direction.
  - **Number of points (U)**: The number of points in U direction.
  - **Number of points (V)**: The number of points in V direction.
- **Displacement**: This parameter is a vector that specifies the translation applied to the points when added to the simulation.
- **Orientation**: This parameter allows the rotation applied to the points. There are several ways the user can specify this rotation: by giving each of the axis directions of the transformation, by specifying spherical rotation angles, by specifying rotation angles on each axis or by specifying the Z axis and a rotation angle.

If the **Relative to reference plane** check box is selected, the displacement and rotation will be relative to the current reference plane coordinate system.

When the user has finished setting up the parameters of the Near Field Source, they need to press the **Save** button in order to add the antenna to the simulation.

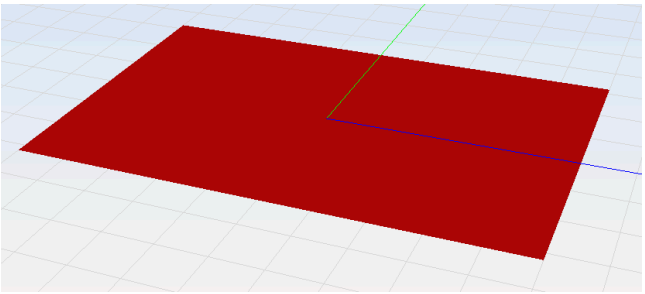


Figure 313: Near Field Source

4.4.4 Import Feko Near Field (.EFE, .HFE)

Allows the user to import an equivalent near field antenna using Feko near field files. Only **Cartesian** and **Cartesian Boundary** types are supported.

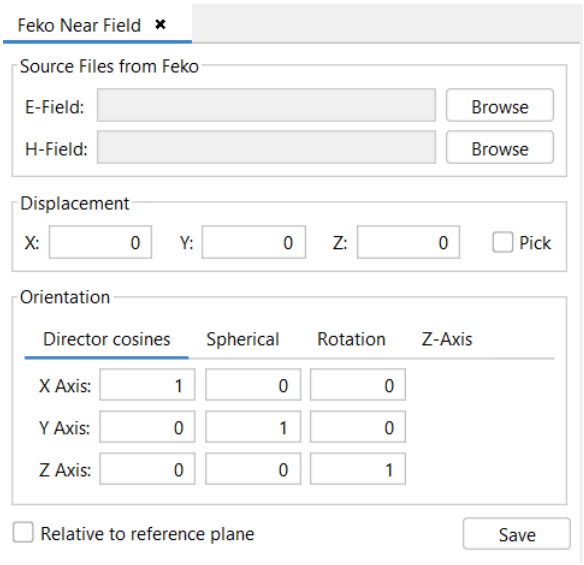


Figure 314: Feko Near Field

Source Files
: The user can import the E-Field (`.efe`) and H-Field (`.hfe`) files from Feko by pressing the **Browse** button and selecting the files in their hard drive.

Displacement
: This parameter is a vector that specifies the translation applied to the points when added to the simulation.

Orientation
: This parameter allows the rotation applied to the points. There are several ways the user can specify this rotation: by giving each of the axis directions of the transformation, by specifying spherical rotation angles, by specifying rotation angles on each axis or by specifying the Z axis and a rotation angle.

If the **Relative to reference plane** check box is selected, the displacement and rotation will be relative to the current reference plane coordinate system.

When the user has finished setting up the parameters of the Feko Near Field Source, they need to press the **Save** button in order to add the antenna to the simulation.

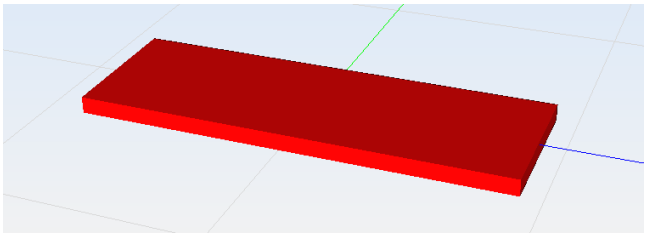


Figure 315: Feko Near Field Source

4.5 Coaxial Feed

4.5.1 Add Feed Point

This option is used to add a coaxial feed point between two surfaces of an object. To create a Feed Point, select one object using the geometry view, then go to Coaxial Feed > Feed Point menu in order to display the Coaxial Feed Panel:

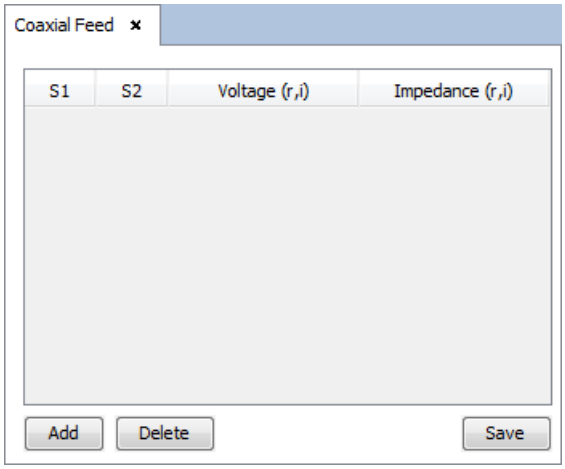


Figure 316: Add Feed Point panel

To add a feed point, select one or two surfaces of the object and click on 'Add' button. A feed point between the surfaces (or an auto feed if only one surface is selected) will be added from the table with a voltage of (1.0, 0.0) and impedance of (0.0, 0.0) that can be modified clicking on the corresponding cell. After setting the values, press Save to add the feed point.

When a coaxial feed has been added, the involved surfaces are represented using red dashed lines.

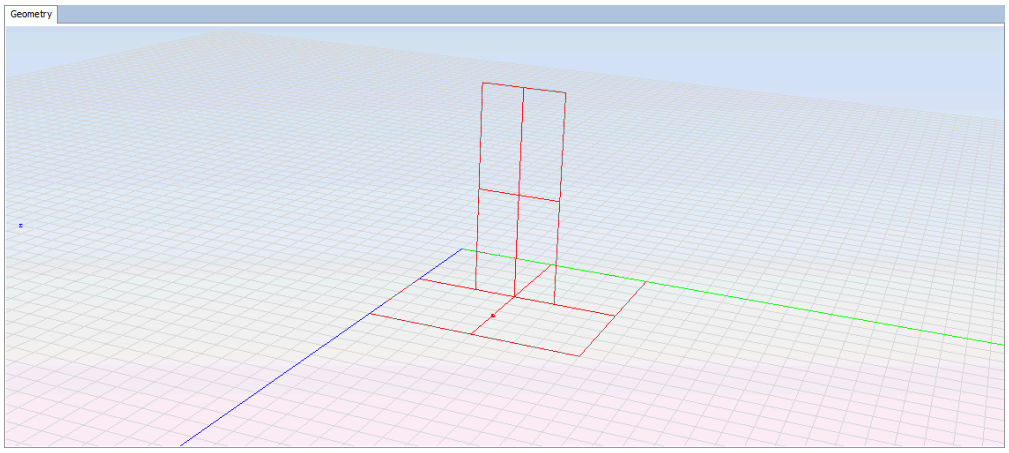


Figure 317: Coaxial Feed Visualization

4.6 Waveguides

4.6.1 Add Waveguide Port

This section explains how to add a waveguide port.

Note This option requires an already created waveguide geometry. The geometry must be only one object (use the group command if it is not) and the waveguide cap needs to be removed.

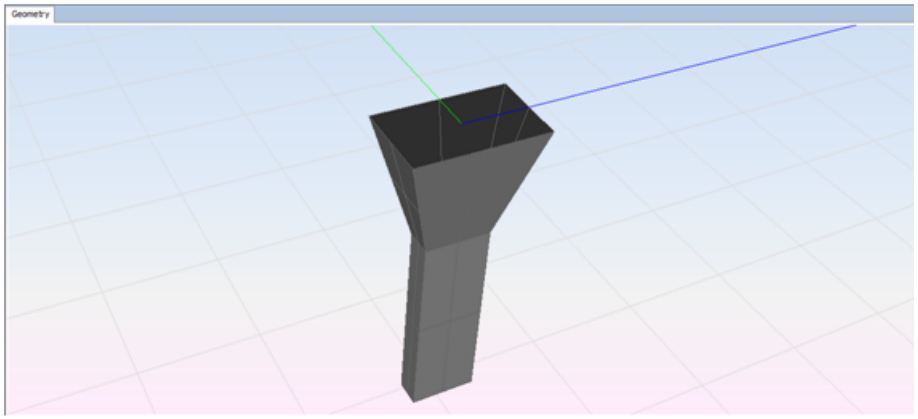


Figure 318: Waveguide geometry

To add a waveguide port, select the object and click on 'Antenna->Waveguides->Add Waveguide port' menu to open the panel.

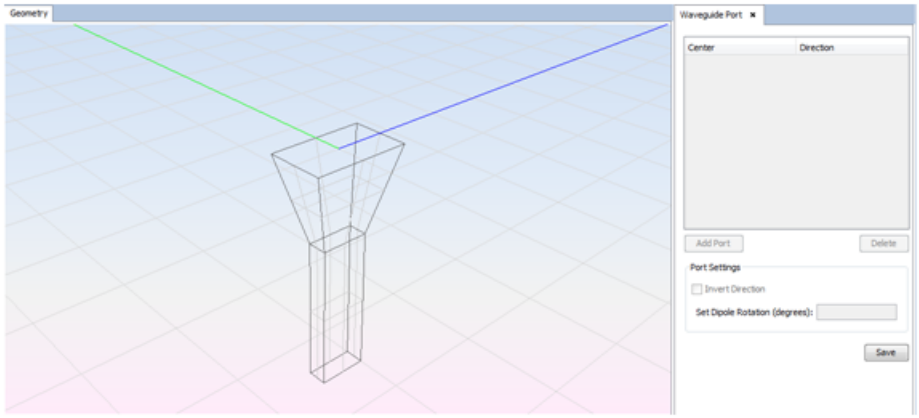


Figure 319: Waveguide port panel with waveguide geometry

Select the port edges with 'Ctrl' pressed and click on 'Add Port'.

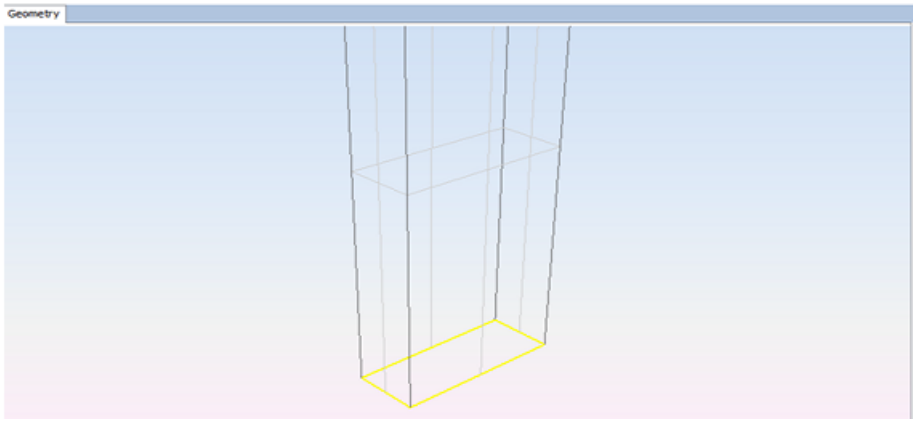


Figure 320: Port edges selected

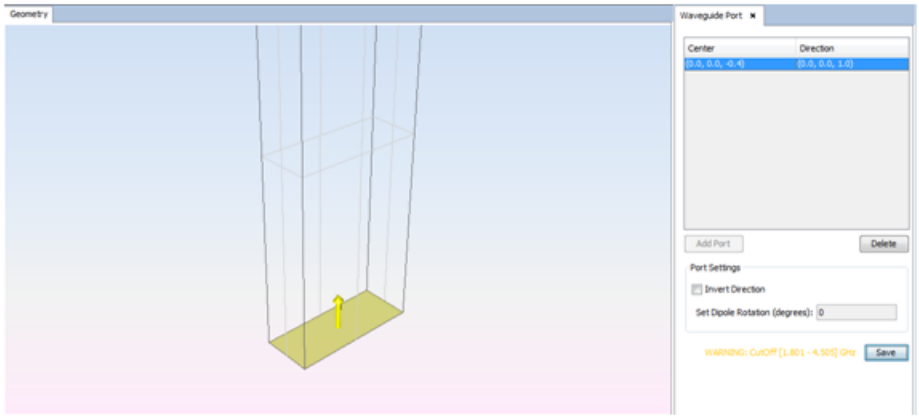


Figure 321: Waveguide port added

Note CutOff frequency warning appears if the current simulation frequency is not adequate for this waveguide. However, you can save the port and modify the frequency later.

Click on the 'Save' button to apply the changes to the waveguide object.

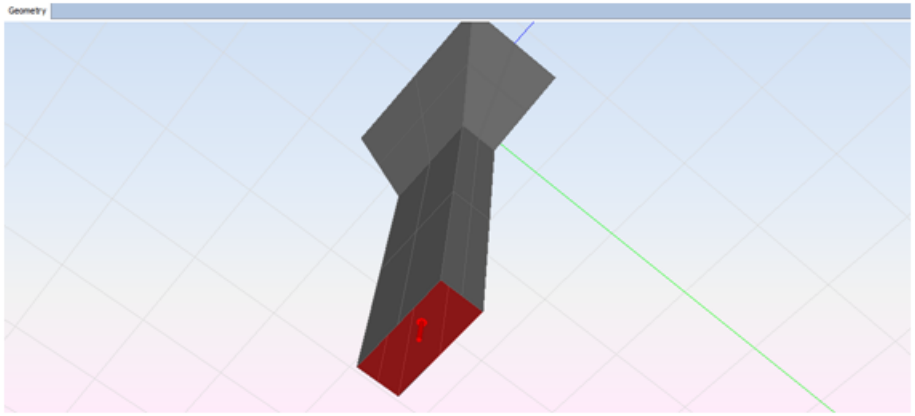


Figure 322: Waveguide port created

4.7 Primitive Antenna

4.7.1 Horns

Pyramidal Horn

This section explains how to create a pyramidal horn.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

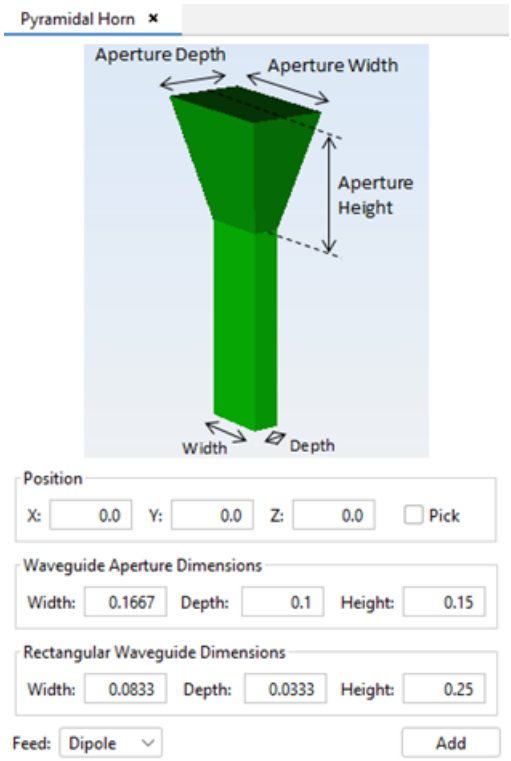


Figure 323: Pyramidal Horn panel

- **Position**: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the 'Pick' check box and clicking on a point in the geometry panel.
- **Waveguide Aperture Dimensions**: set the values for the pyramidal geometry on the top of the antenna. 'Width' and 'Depth' values are only for the top sides of the antenna. The bottom sides of the pyramidal geometry will be added from the 'rectangular waveguide dimensions' parameters.
- **Rectangular Waveguide Dimensions**: set the values for the rectangular geometry on the bottom of the antenna. 'Width' and 'Depth' values will be added to the bottom sides of the pyramidal geometry on the top of the geometry.

- **Feed**: Set the type of feeding which will receive the antenna. Select None if no feeding point is required, Coaxial to establish a coaxial feeding point in the bottom of the horn (which will be represented by a red-dashed line), or Dipole in order to set a single dipole antenna.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

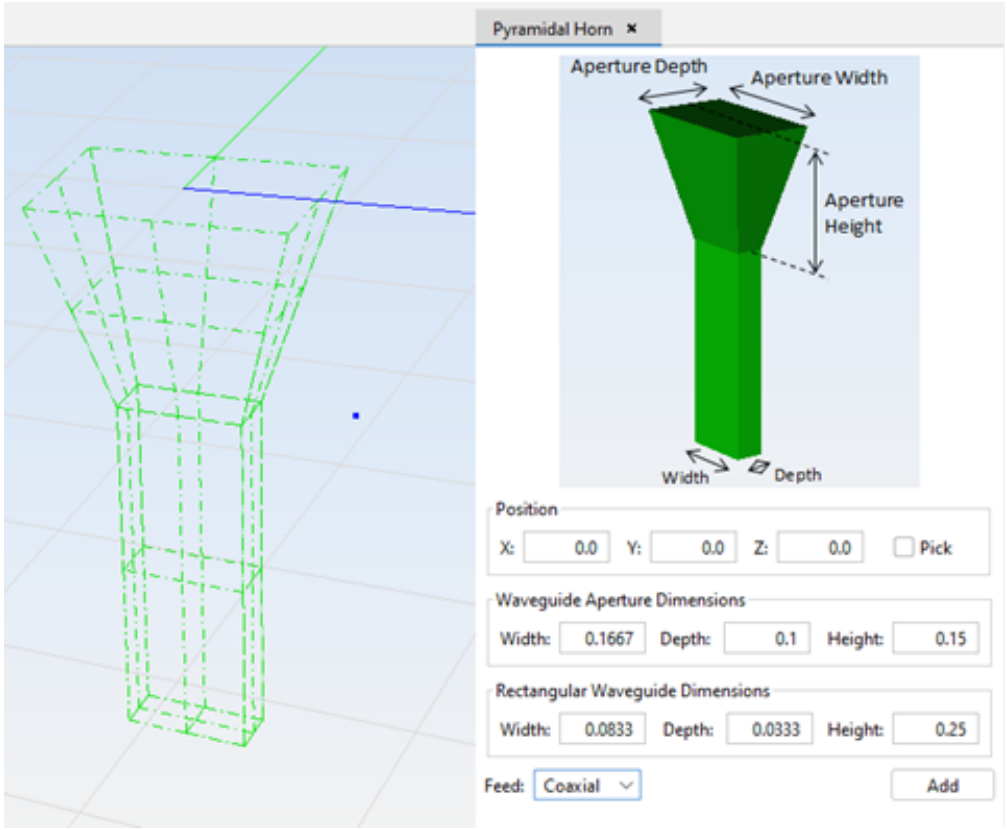


Figure 324: Pyramidal Horn previsualization

Clicking on 'Add' button, the current previsualization turns on the final visualization for the antenna with a red dashed line for the coaxial feed.

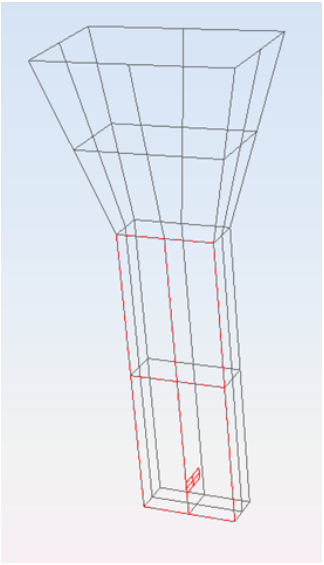


Figure 325: Pyramidal Horn visualization

Conical Horn

This section explains how to create a conical horn.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

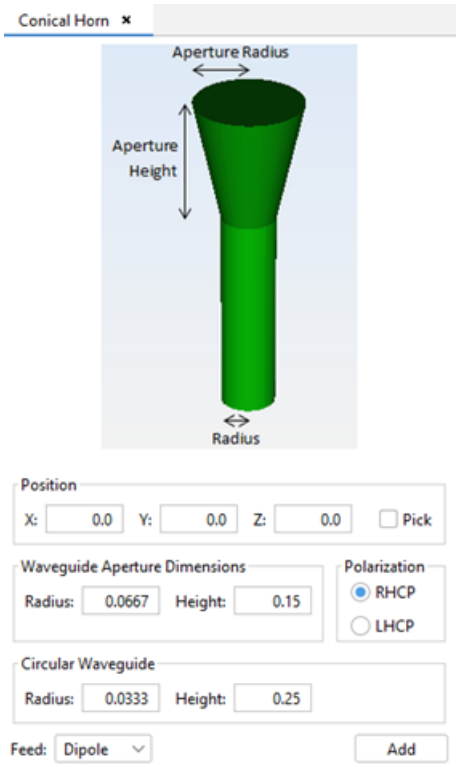


Figure 326: Conical Horn panel

- **Position**: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the 'Pick' check box and clicking any point in the geometry panel.
- **Waveguide Aperture Dimensions**: defines the parameters for the conical geometry on the top of the antenna. The height defines the length of the aperture section, while the radius defines the radius of the circular hole at the end of the aperture section.
- **Circular Waveguide Dimensions**: sets the values for the cylindrical geometry on the bottom of the antenna. The height defines the length of the cylindrical section, while the radius defines the radius of this cylindrical section.
- **Feed**: Set the type of feeding which will receive the antenna. Select None if no feeding point is required, Coaxial to establish a coaxial feeding point in the bottom of the horn (which will be represented by a red-dashed line), or Dipole in order to set a single dipole antenna.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

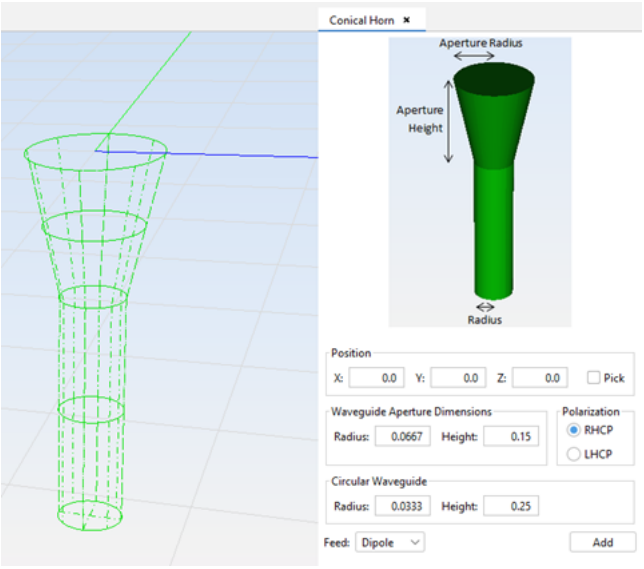


Figure 327: Conical Horn previsualization

Clicking on 'Add' button, the current previsualization turns on the final visualization for the antenna with a red cone for the dipole.

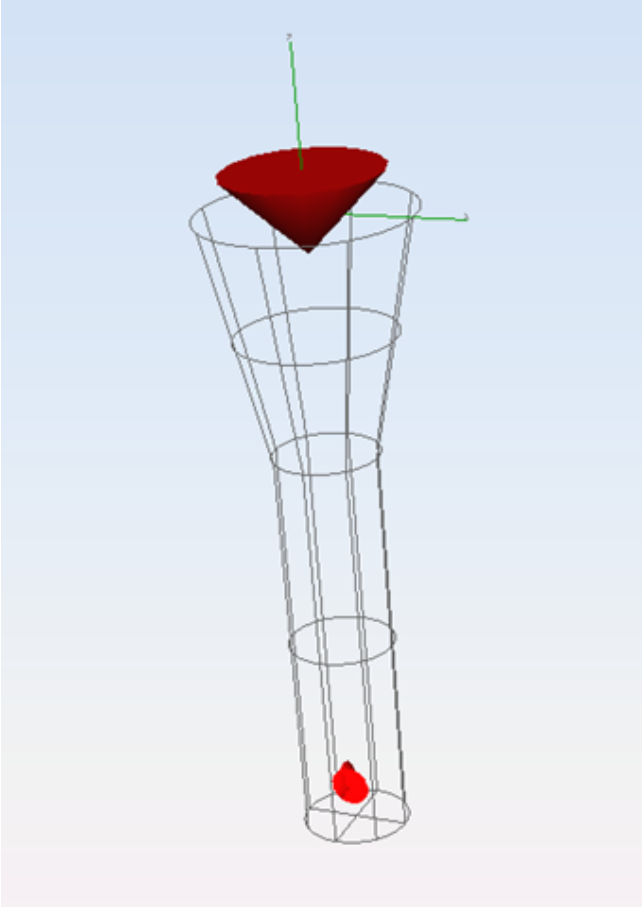


Figure 328: Conical Horn visualization

4.7.2 Coaxial Feed

Biconical

This section explains how to create a biconical antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

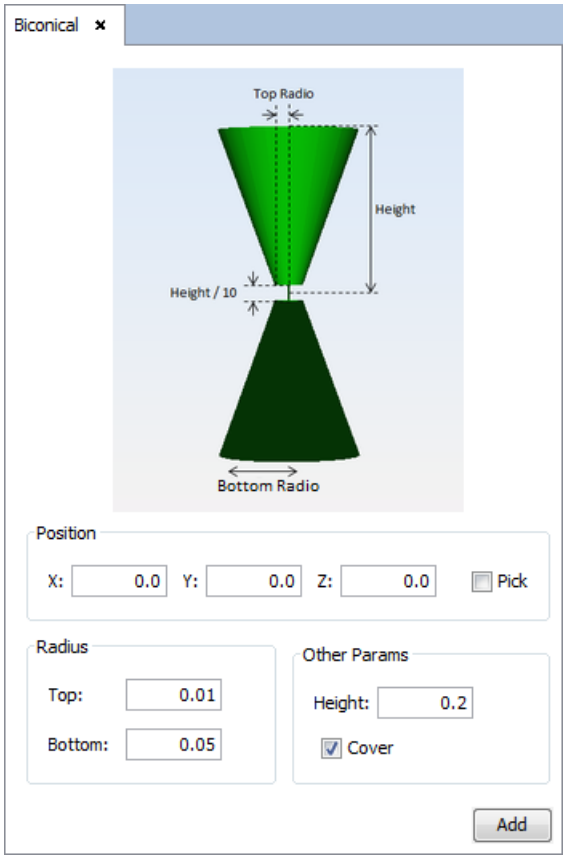


Figure 329: Biconical panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the 'Pick' check box and clicking any point in the geometry panel.
- Radius: set the values for the radius of the conical geometries of the antenna.
- Other Params: set the value for the height of the conical geometries of the antenna. Selecting the'Cover' option,the caps in both ends of the antenna geometry will be created.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

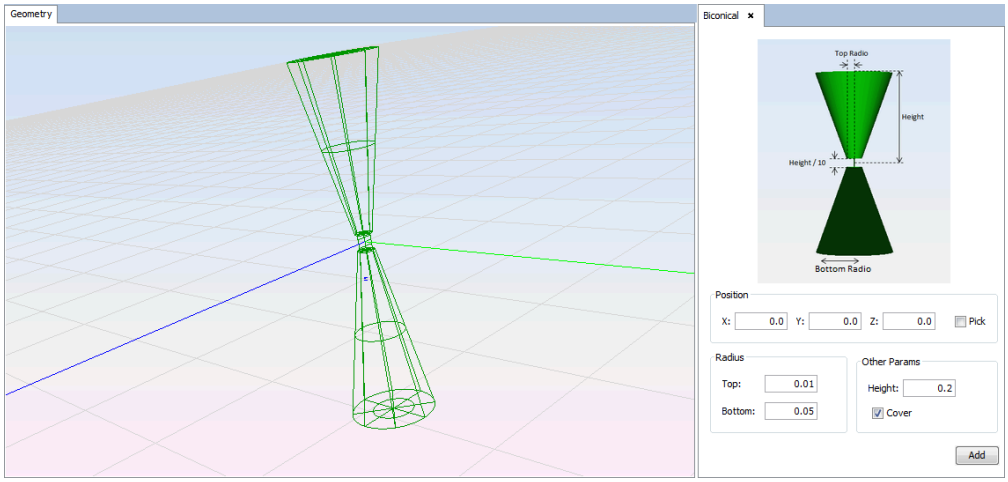


Figure 330: Biconical previsualization

Clicking on 'Add' button, the current previsualization turns on the final visualization for the antenna with a red dashed line for the coaxial feed.

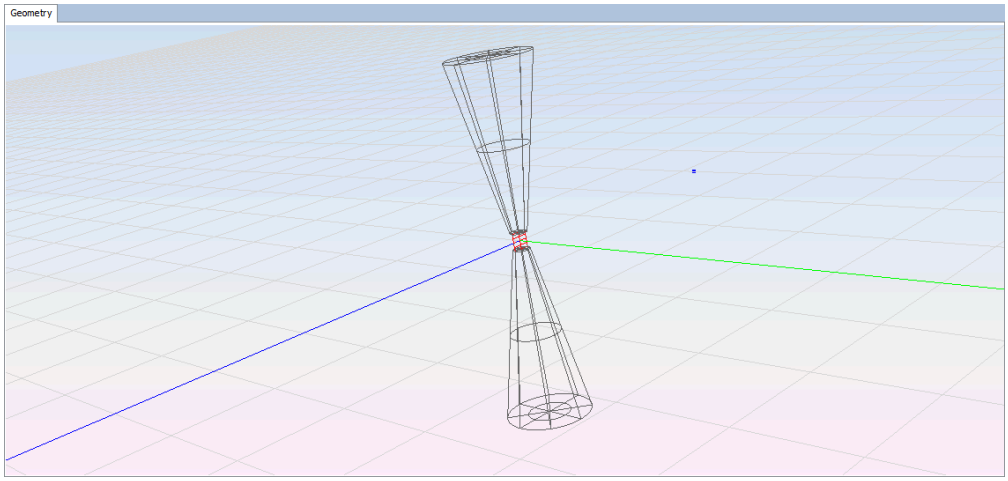


Figure 331: Biconical visualization

Blade

Simple Blade

This section explains how to create a simple blade.

When the option is selected, the following panel and a previsualization of the antenna with the parameters on the panel are displayed.

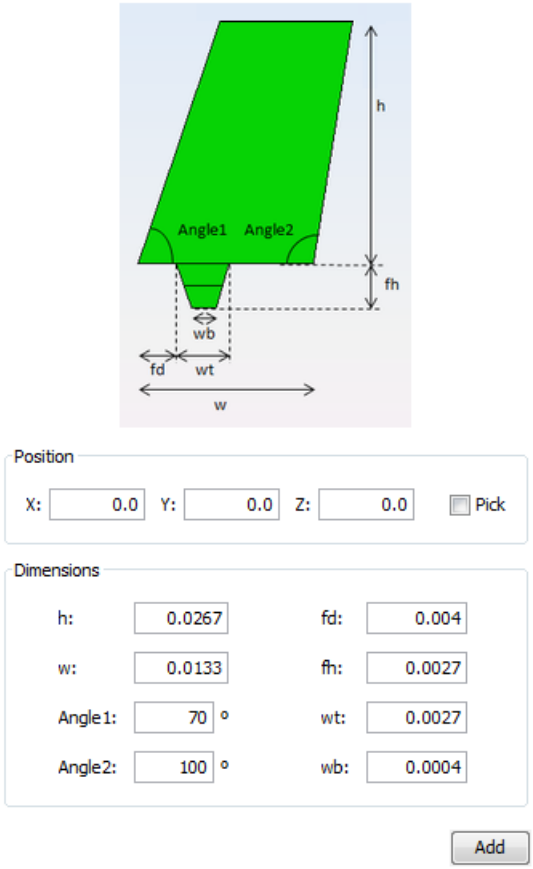


Figure 332: Simple Blade Panel

- Position: sets the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the "Pick" check box and clicking any point in the geometry panel.
- Dimensions: sets the height (h) and the width (w) of the antenna, the lower angles of the blade (Angle1 and Angle2), and the dimensions of the coaxial feed the height (fh), the base width (wb), the top width (wt) and the depth (fd).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

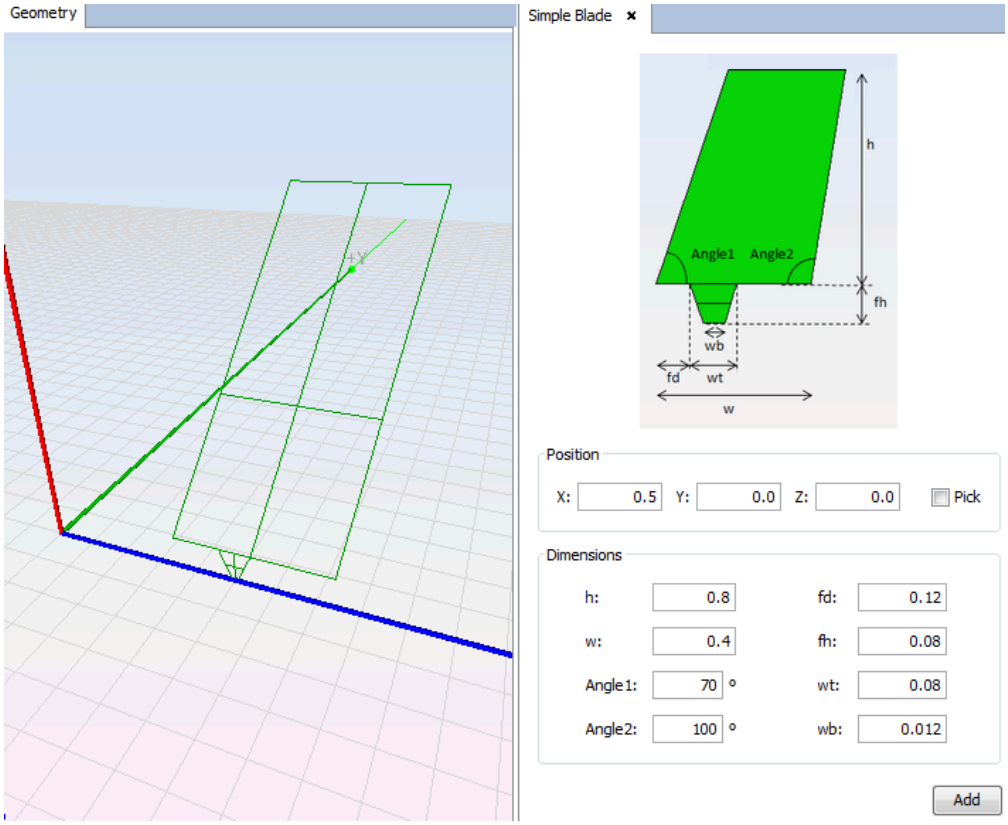


Figure 333: Simple Blade previsualization

Clicking on 'Add' button, the current previsualization turns on the final visualization for the antenna with a red dashed line for the coaxial feed.

Note that this antenna should be placed on a metallic ground plane to provide a good performance as shown in the below figure. The ground plane surface must be attached below the antenna, with no gap between them.

Figure 334: Simple Blade with ground plane visualization

Dipole

This section explains how to add a coaxial feed dipole.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

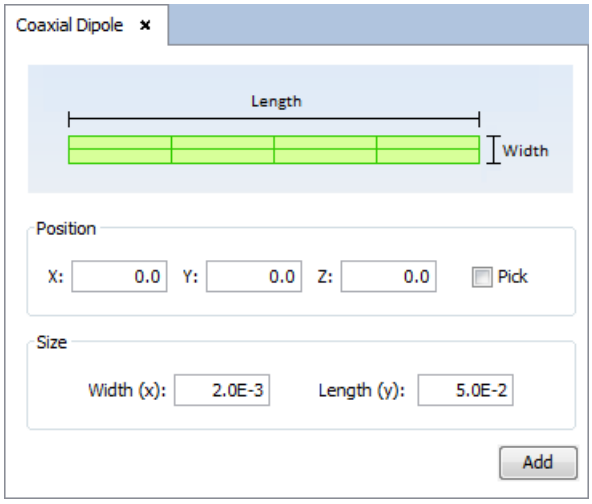


Figure 335: Coaxial Dipole panel

- Position set the Cartesian coordinates for the position of the antenna/dipole. For the antenna, the user can select the position with the mouse, selecting the "Pick" check box and clicking any point in the geometry panel. For the dipoles, these values are relative to the same values defined for the antenna.

- Size set the values for size on 'X' and 'Y' axis.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

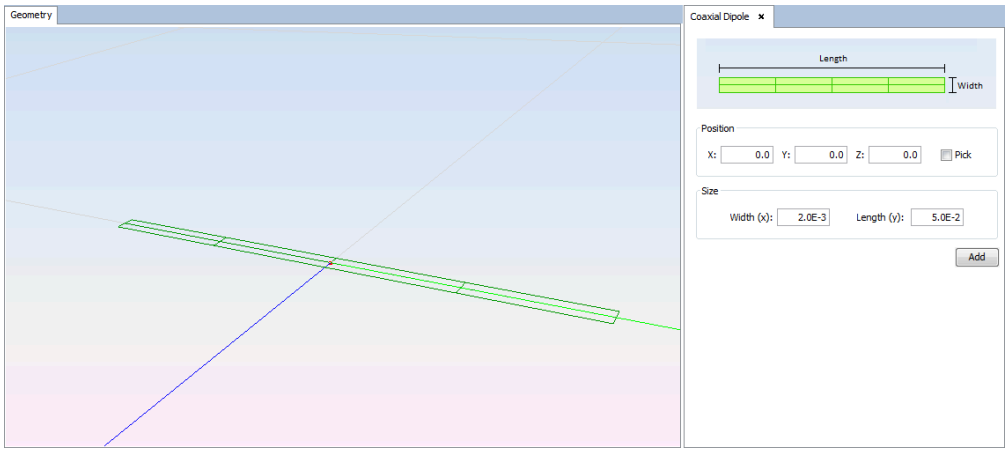


Figure 336: Coaxial Dipole previsualization

Clicking on the 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

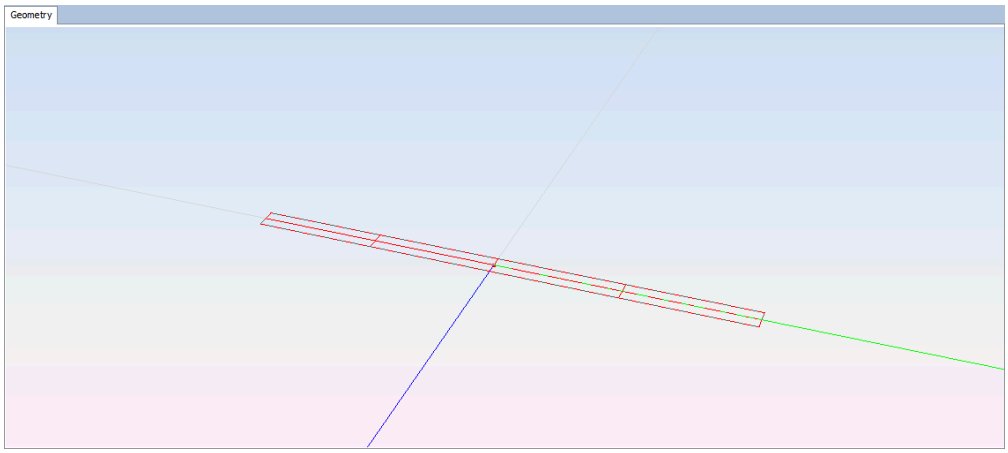


Figure 337: Coaxial Dipole visualization

Helix

Multi-Filar Helix

This section explains how to create a multi-filar helix.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

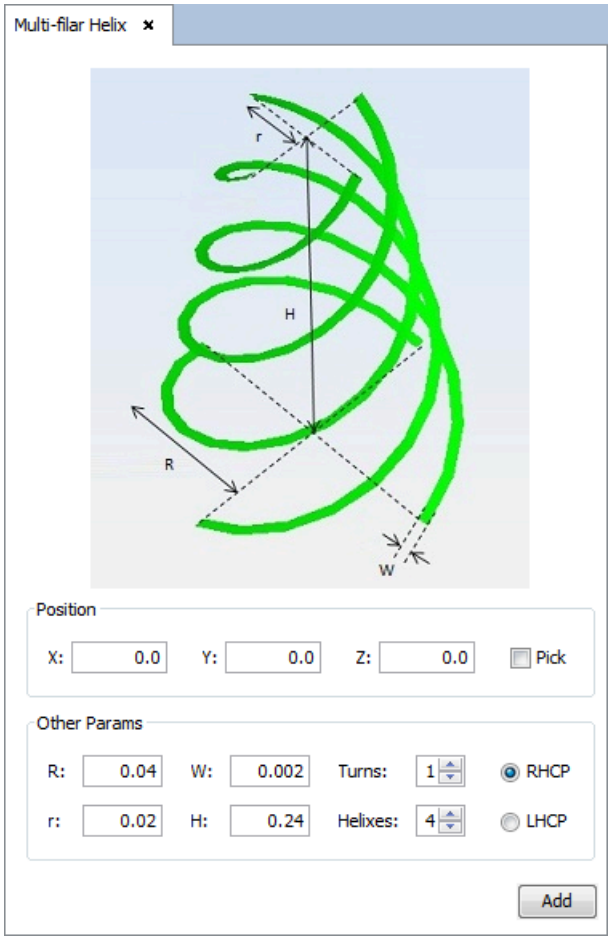


Figure 338: Multi-Filar Helix panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the 'Pick' check box and clicking any point in the geometry panel.
- Other Params: set the value for the bottom radius (R), top radius (r) and height for the helix. The width of the strip (w), the number of turns (Turns) and the number of strips (Helixes) will be defined too. RHCP or LCHP selected option will define the polarization and the turn direction.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

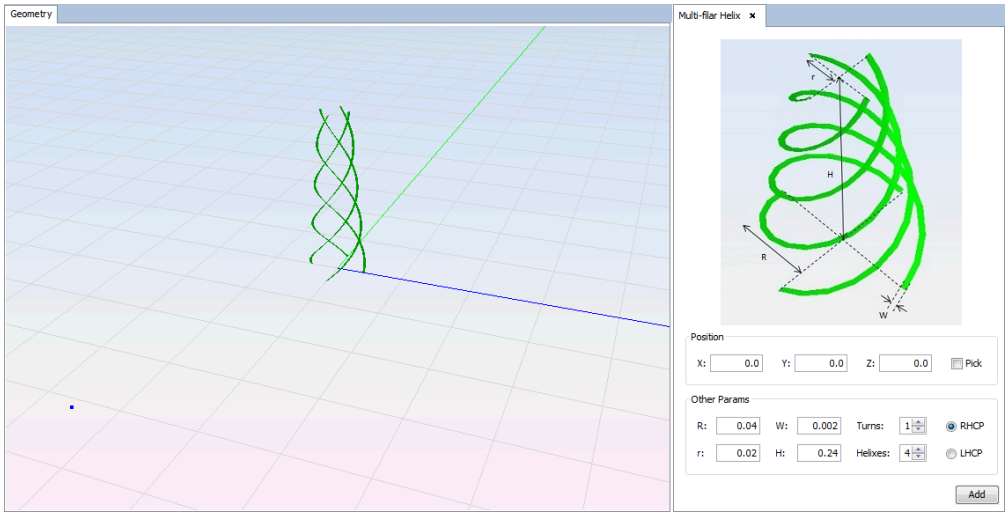


Figure 339: Multi-Filar Helix previsualization

Clicking on the 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

Note that this antenna should be placed on a metallic ground plane to provide a good performance as shown in the below figure. The ground plane surface must be attached below the antenna, with no gap between them.

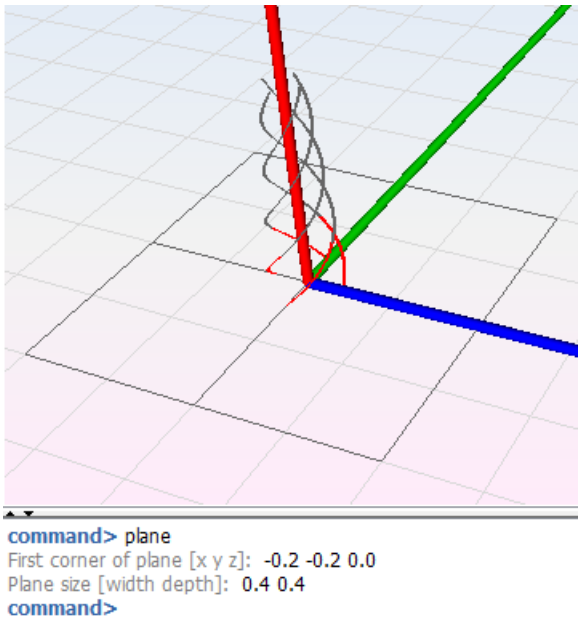


Figure 340: Multi-Filar Helix with ground plane visualization

LPDA

This section explains how to create a LPDA antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

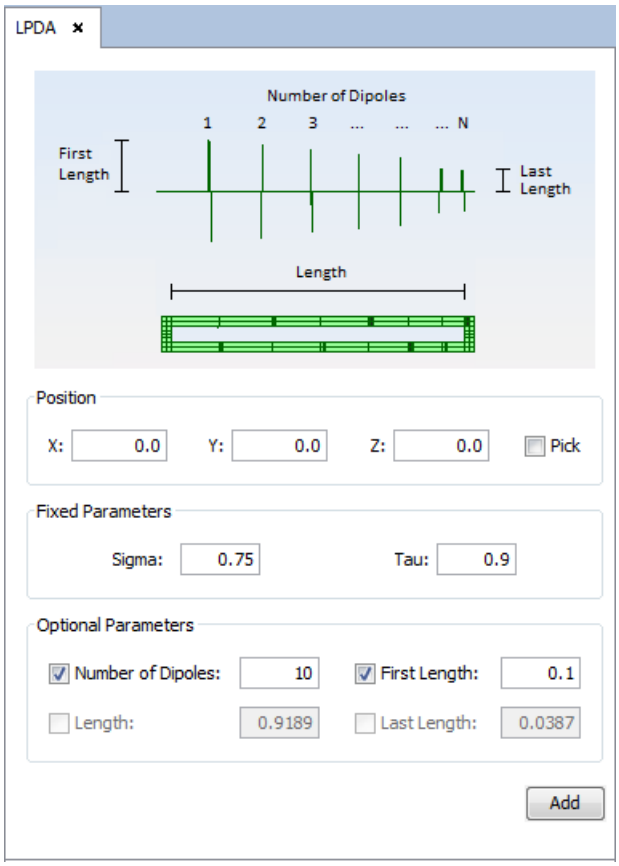


Figure 341: LPDA panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse, selecting the 'Pick' check box and clicking any point in the geometry panel.
- Fixed Parameters: set the sigma parameter (Sigma) and the tau parameter (Tau).
- Optional Parameters: set the number of dipoles (Number of Dipoles), the length of them (Length), the length of the first dipole (First Length) or length of the last dipole (Last Length).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

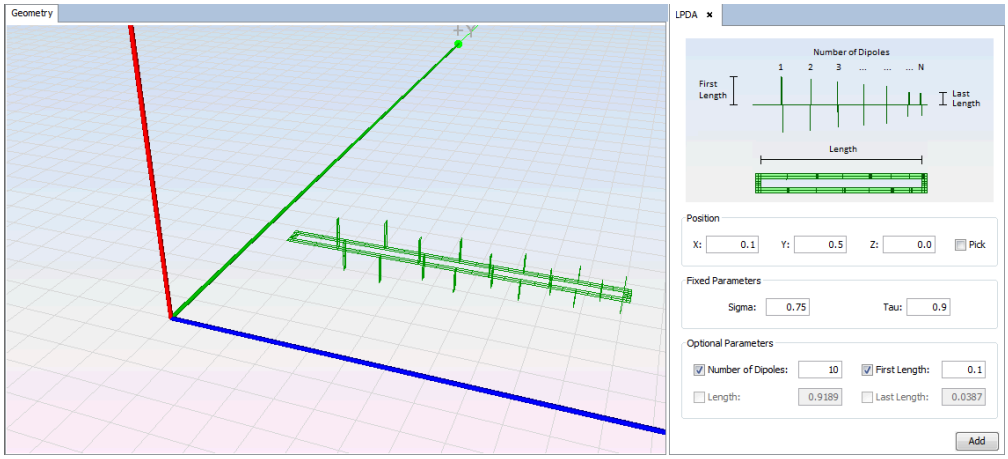


Figure 342: LPDA previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed is shown.

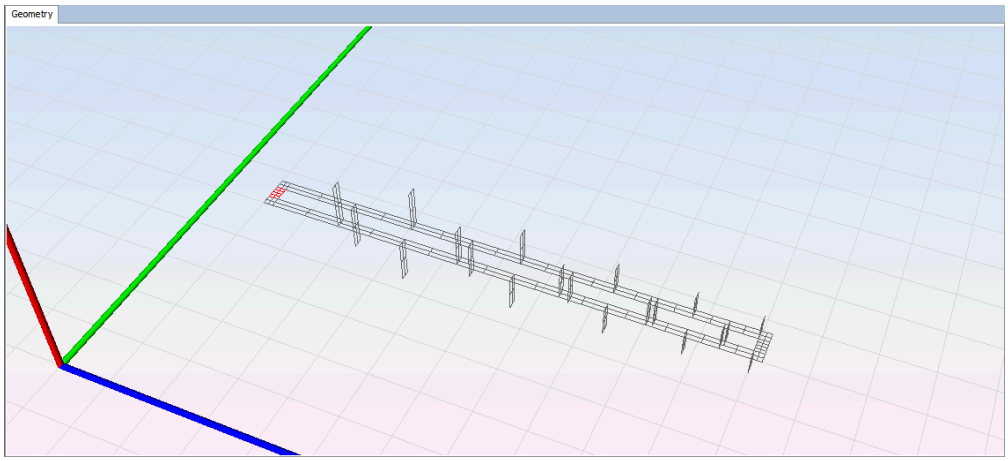


Figure 343: LPDA visualization

Spiral

Circular Spiral

This section explains how to create a circular spiral.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

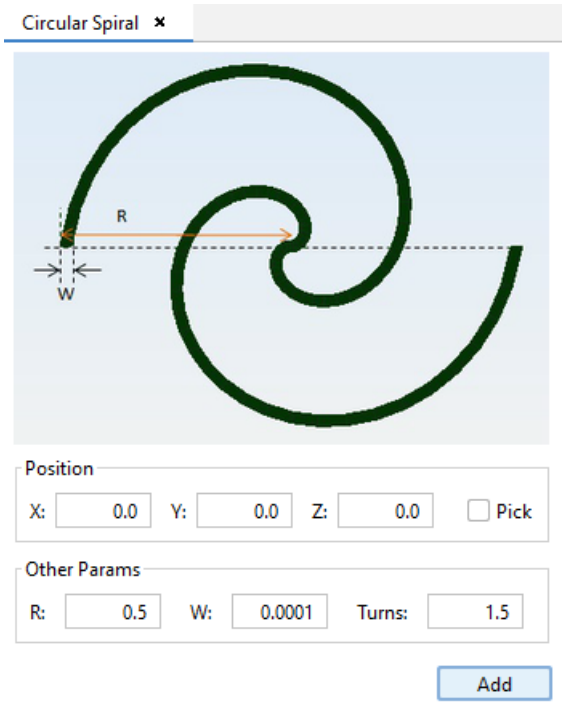


Figure 344: Circular Spiral panel

- **Position**: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse, selecting the 'Pick' check box and clicking any point in the geometry panel.
- **Other Params**: set the value for the radius (R), the width of the strip (w) and the number of turns (Turns). Turns can be a decimal value.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

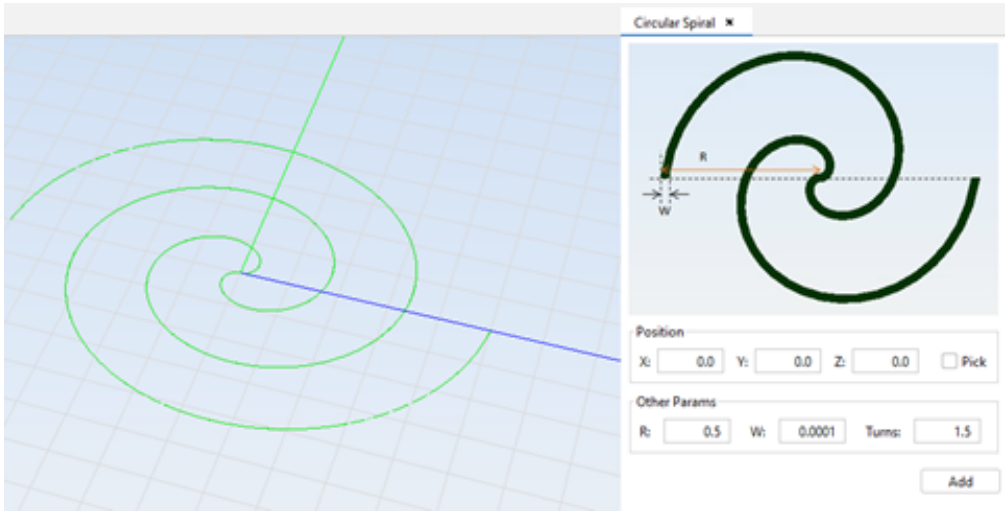


Figure 345: Circular Spiral previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

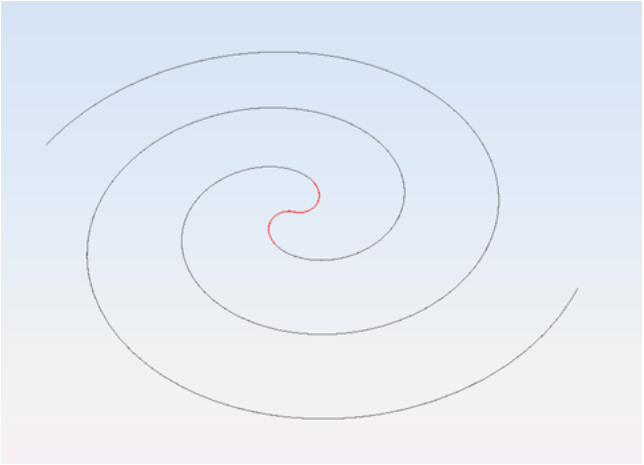


Figure 346: Circular Spiral visualization

Wire

Arbitrary Wire

This section explains how to create a arbitrary wire antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

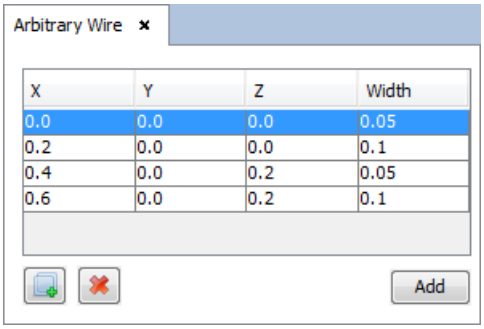


Figure 347: Arbitrary Wire panel

- Adds a new row for a new point with default values (0.0, 0.0, 0.0, 0.02).
- Delete the selected rows.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

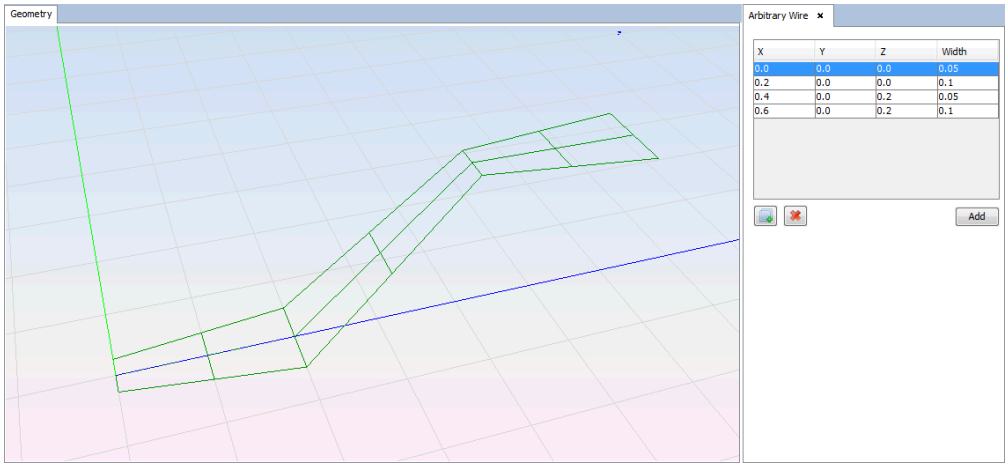


Figure 348: Arbitrary Wire previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

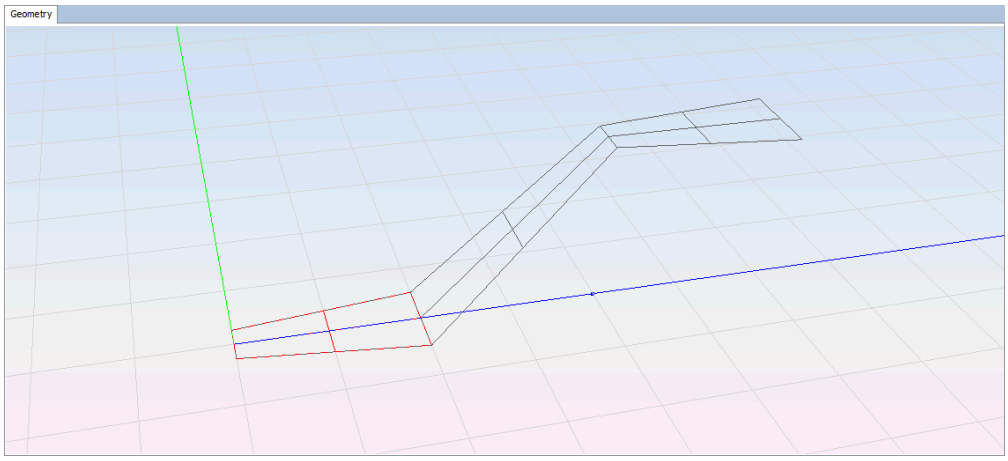


Figure 349: Arbitrary Wire visualization

4.7.3 Slot Antennas

LTSA

This section explains how to create a LTSA slot antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

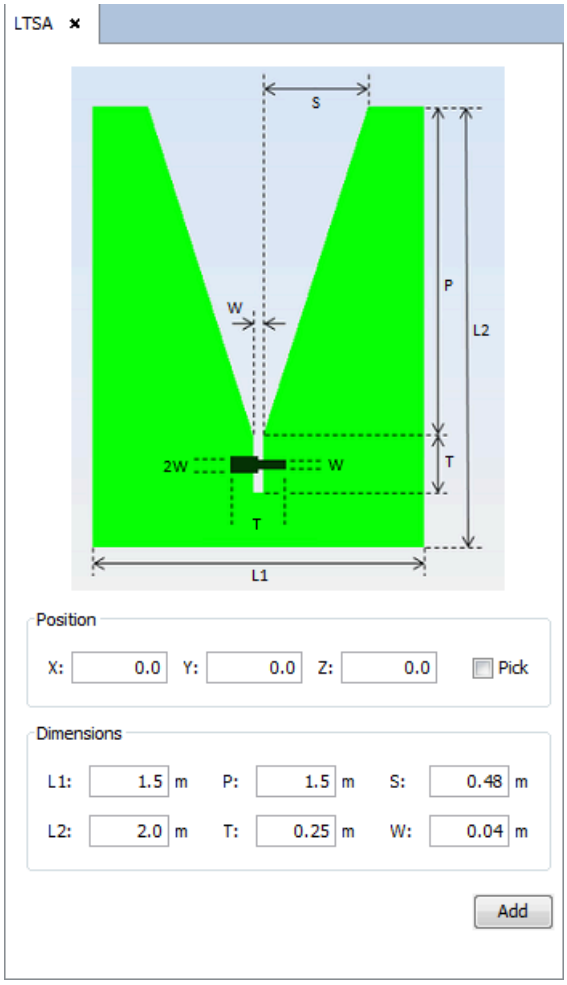


Figure 350: LTSA panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, the user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Dimensions: set the values of the length of the sides (L1 and L2), the separation from the slot (S), the length of the blades (P), the length of the coaxial (T) and the width of the slot (W).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

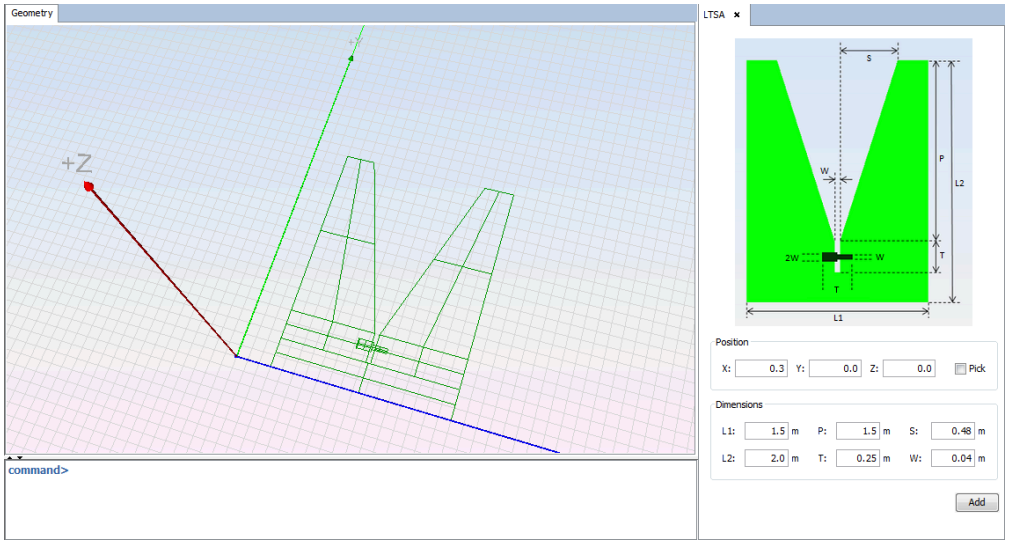


Figure 351: LTSA previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

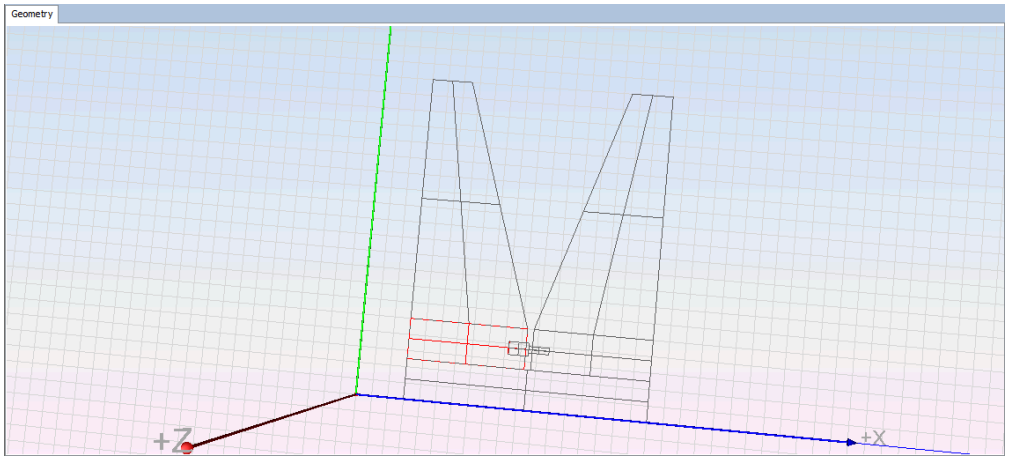


Figure 352: LTSA visualization

Vivaldi

This section explains how to create a Vivaldi slot antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

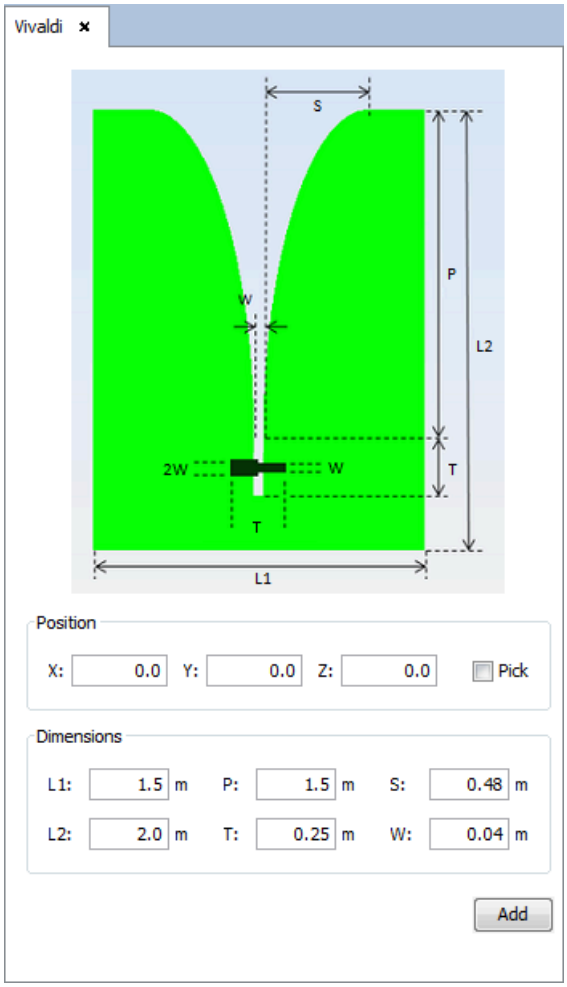


Figure 353: Vivaldi panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Dimensions: set the values of the length of the sides (L1 and L2), the separation from the slot (S), the length of the blades (P), the length of the coaxial (T) and the width of the slot (W).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

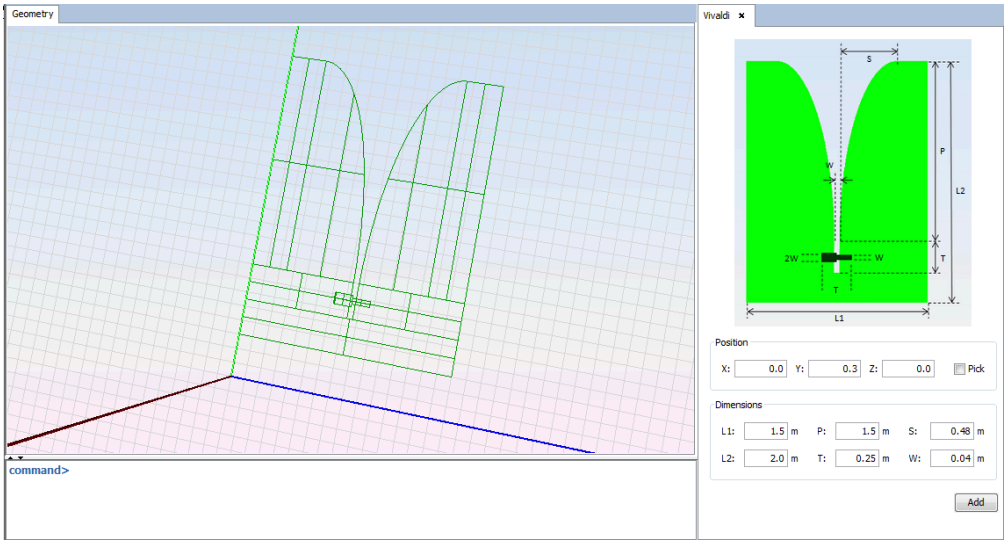


Figure 354: Vivaldi previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

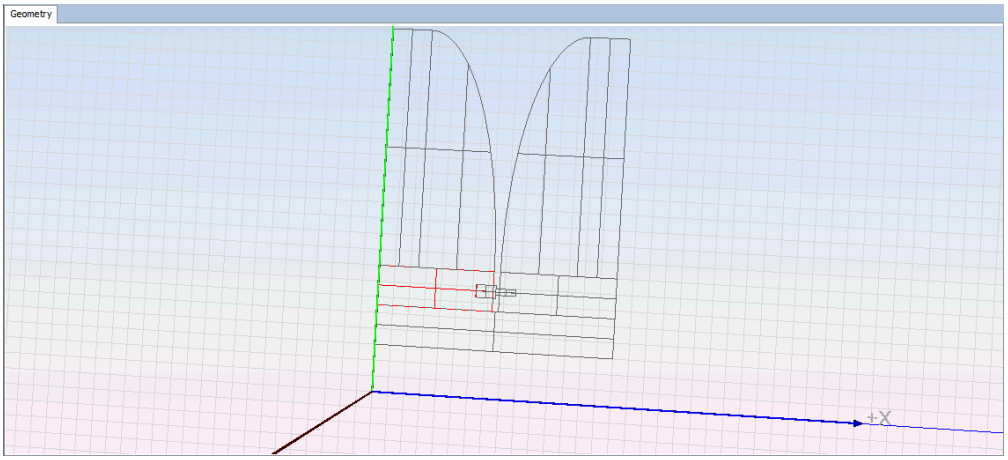


Figure 355: Vivaldi visualization

Travelling-Wave Array

This section explains how to create a travelling-wave slot array antenna (non-resonant) fed by a rectangular waveguide.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

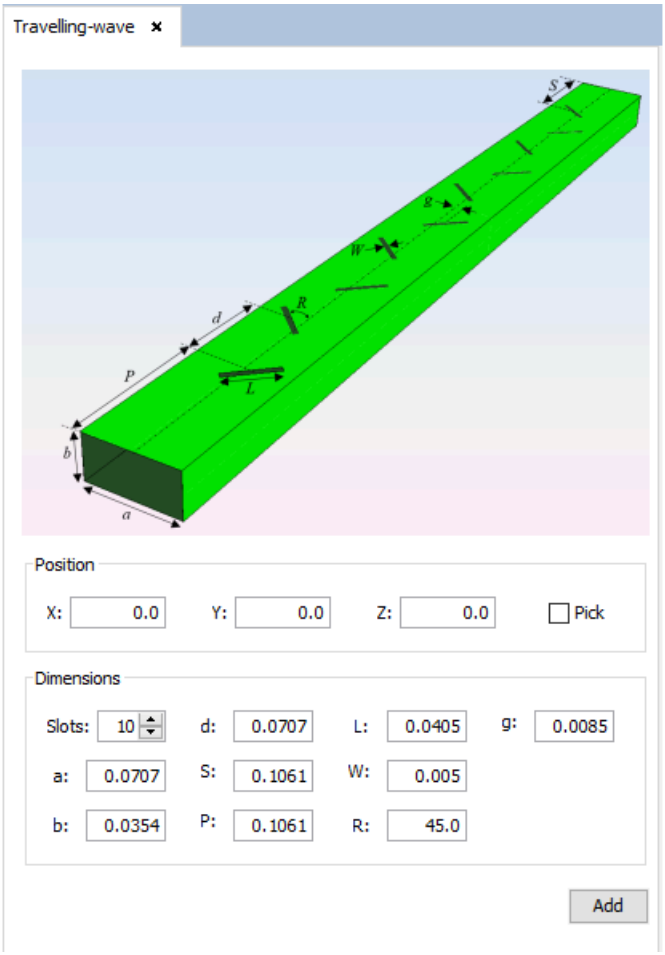


Figure 356: Travelling-wave Array panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Dimensions: set the values of the number of slots (Slots), the width and height of the feeding waveguide (a and b), the distance between adjacent slots (d), the length and width of the slots (L and W), the rotation angle of the slots (R in degrees), the distance from the start of the waveguide to the first slot center (P), the distance from the end of the waveguide to the last slot center (S) and the separation from the slot center to the broadwall center of the waveguide (g).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

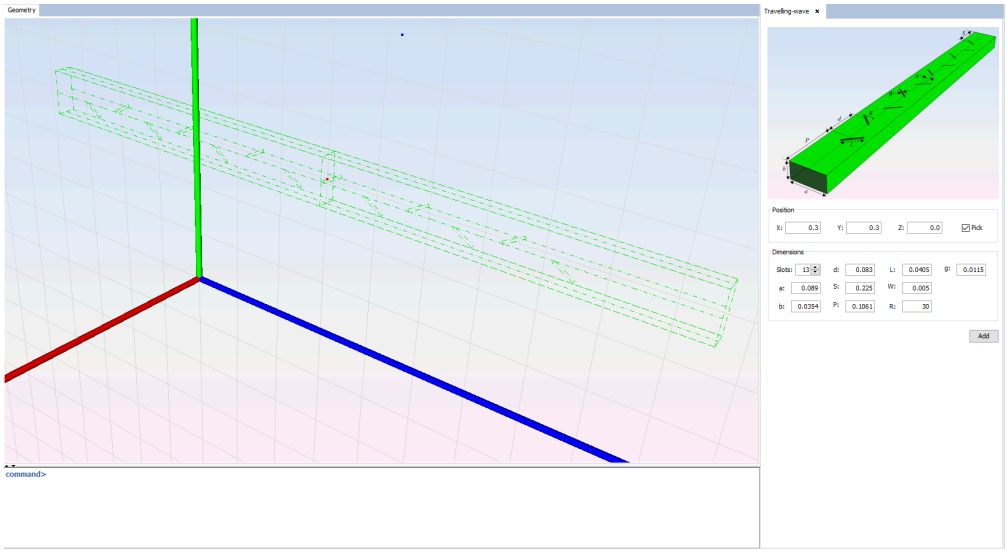


Figure 357: Travelling-wave Array previsualization

Clicking on 'Add' button, the final visualization for the antenna with red filled waveguide ports (input and output) will be shown.

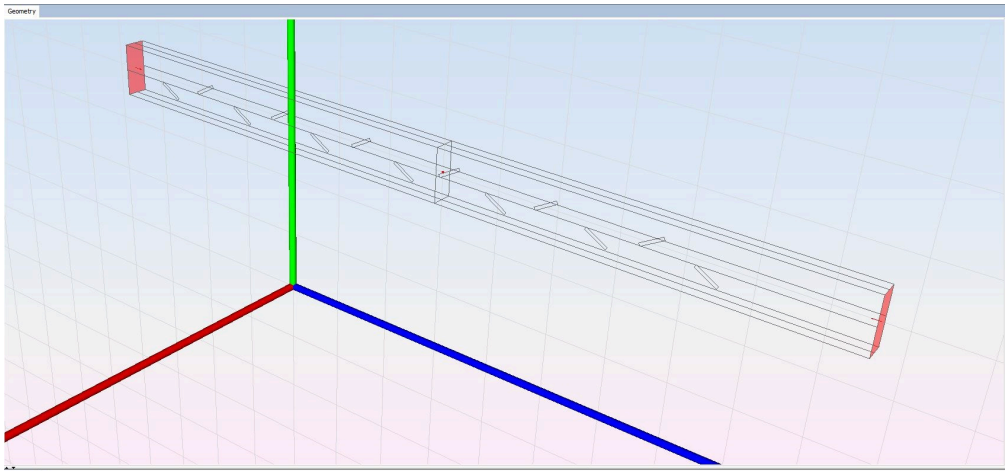


Figure 358: Travelling-wave Array visualization

Resonant 1D Array

This section explains how to create a resonant slot linear (1D) array antenna fed by a rectangular waveguide.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

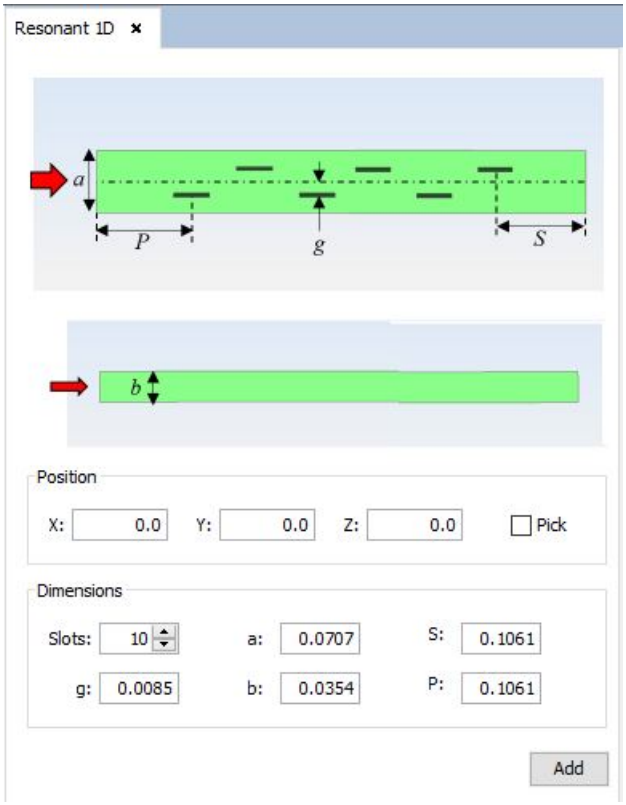


Figure 359: Resonant 1D Array panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Dimensions: set the values of the number of slots (Slots), the width and height of the feeding waveguide (a and b), the distance from the start of the waveguide to the first slot center (P), the distance from the end of the waveguide (short) to the last slot center (S) and the separation from the slot center to the broadwall center of the waveguide (g).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

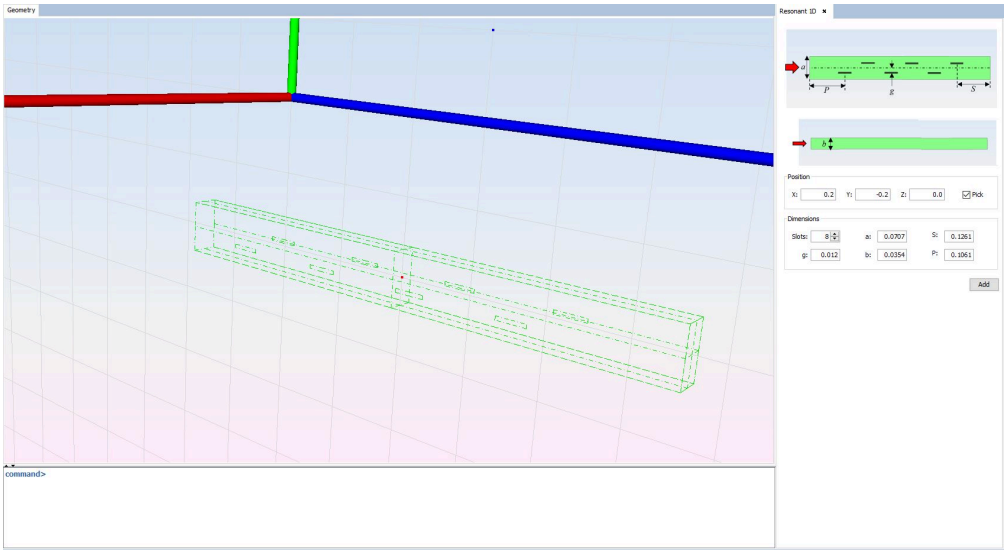


Figure 360: Resonant 1D Array previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red filled waveguide port will be shown.

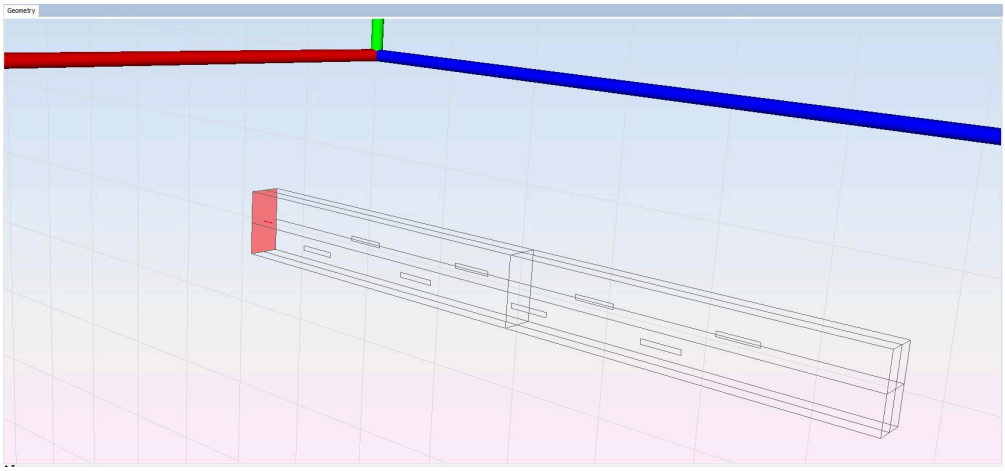


Figure 361: Resonant 1D Array visualization

Resonant 2D Array

This section explains how to create a resonant slot planar (2D) array antenna, where a feeding rectangular waveguide excites an array of resonant linear (1D) array antennas.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

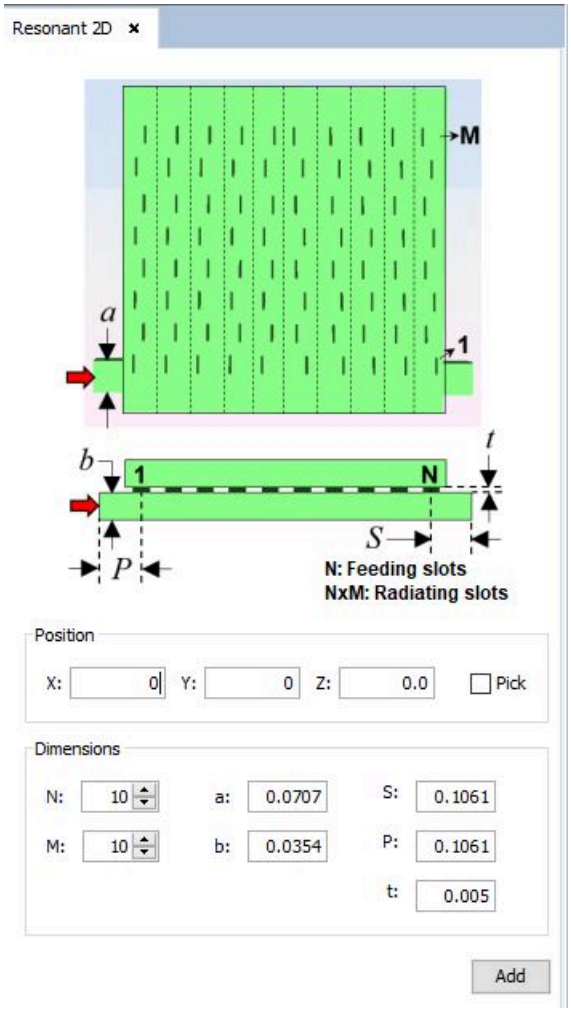


Figure 362: Resonant 2D Array panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Dimensions: Set the values of the number of radiating slots (NxM, being N the number of feeding slots and M the number of radiating slots for each resonant 1D array), the width and height of the feeding waveguide (a and b), the distance from the start of the waveguide to the first slot center (P), the distance from the end of the waveguide (short) to the last slot center (S) and the height of the feeding slots pipes (t).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

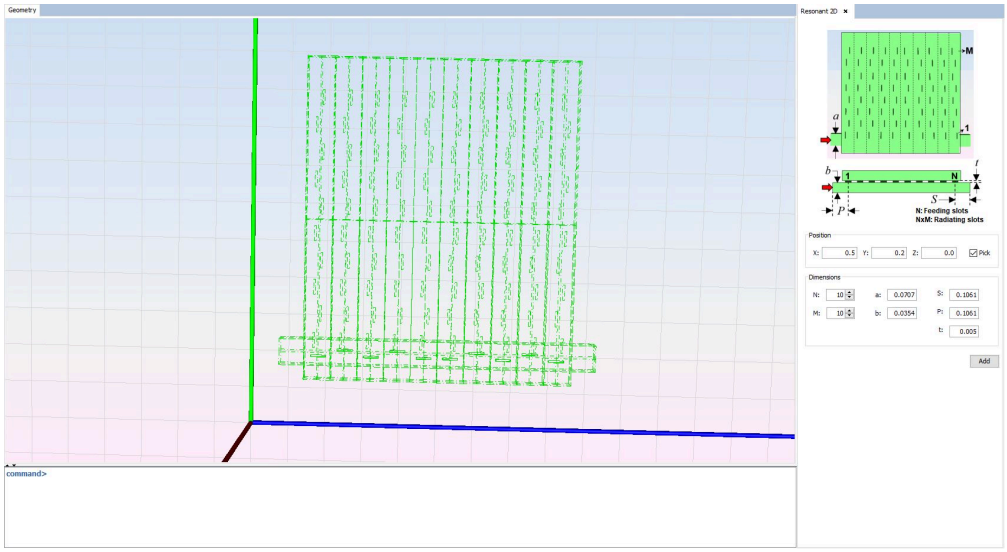


Figure 363: Resonant 2D Array previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red filled waveguide port will be shown.

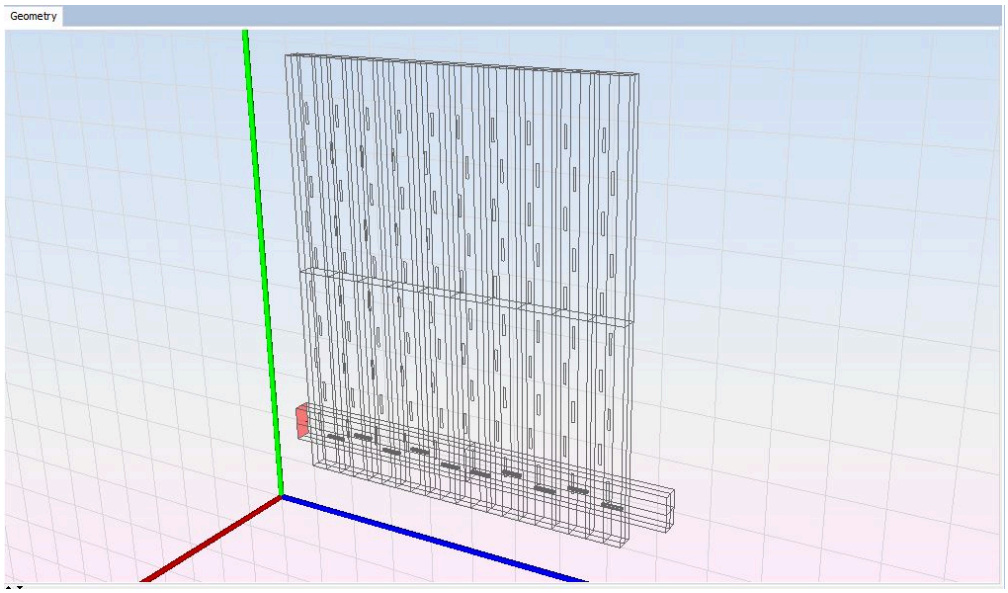


Figure 364: Resonant 2D Array visualization

4.7.4 Patch Antennas

Single Patch

This section explains how to create a single patch antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

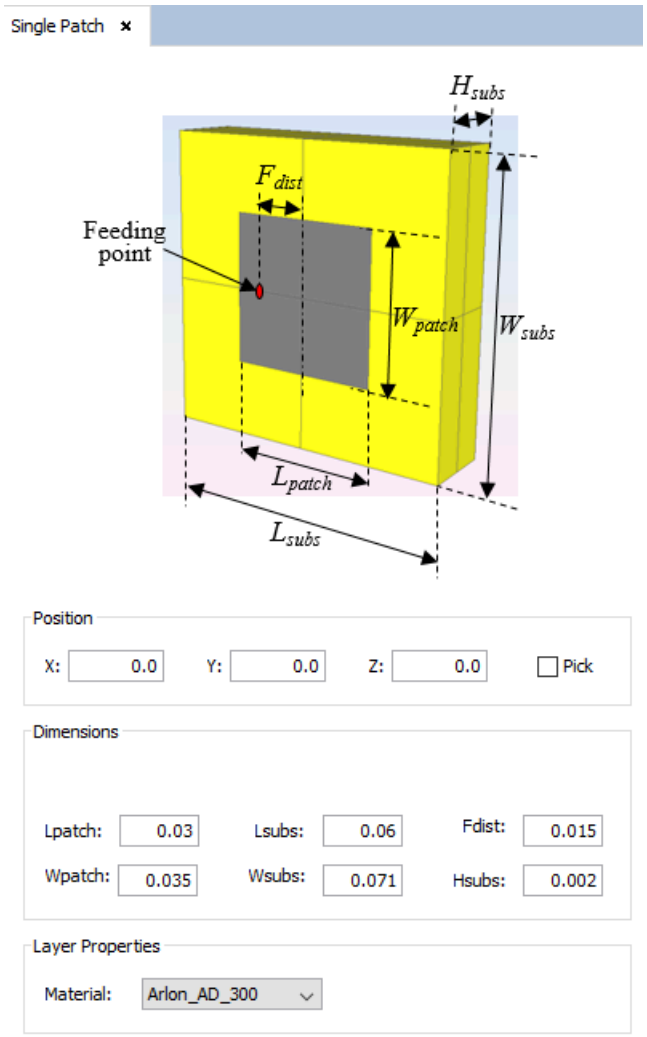


Figure 365: Single Patch panel

Position set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.

Dimensions set the values of the length and width of the patch (Lpatch and Wpatch), the position of the feeding point from the center of the patch (Fdist) as well as the length, width and height of the substrate material (Lsubs, Wsubs and Hsubs).

Layer Properties set the substrate material from the material library of newFASANT.

For each modification on parameters panel, this previsualization will be updated for the new parameters.

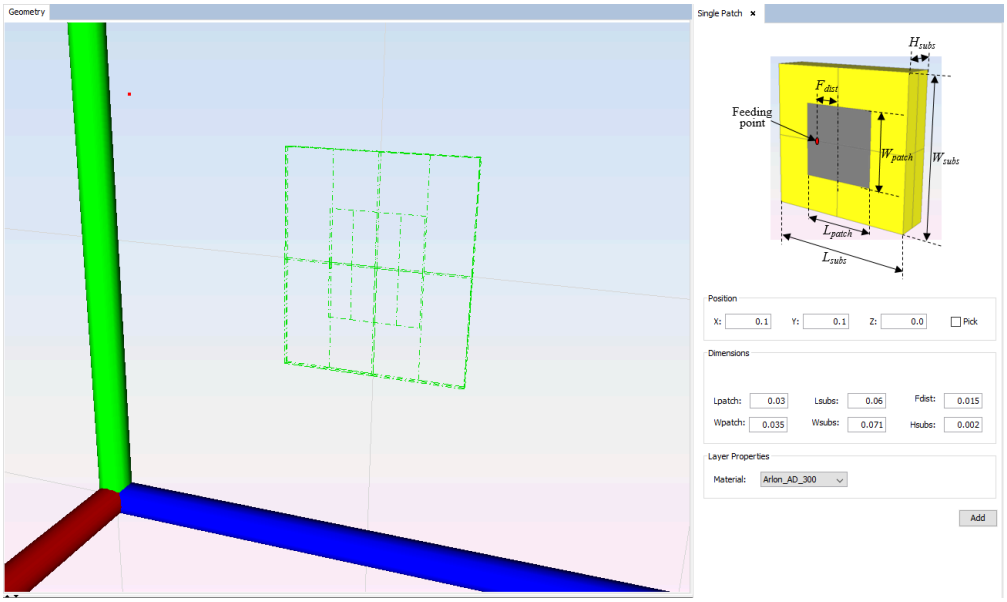


Figure 366: Single Patch previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for the coaxial feed will be shown.

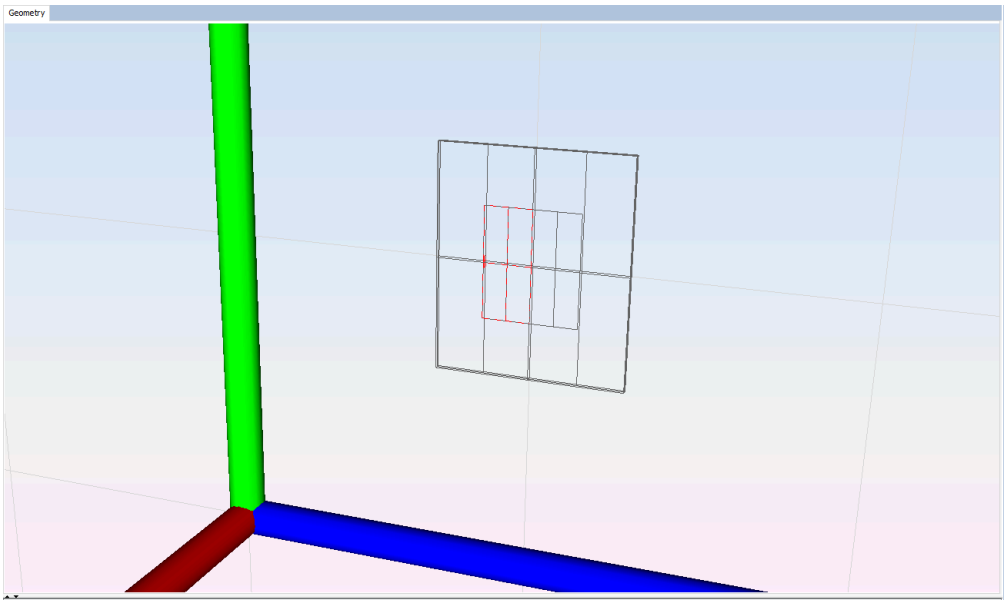


Figure 367: Single patch visualization

Planar Array

This section explains how to create a planar patch array antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

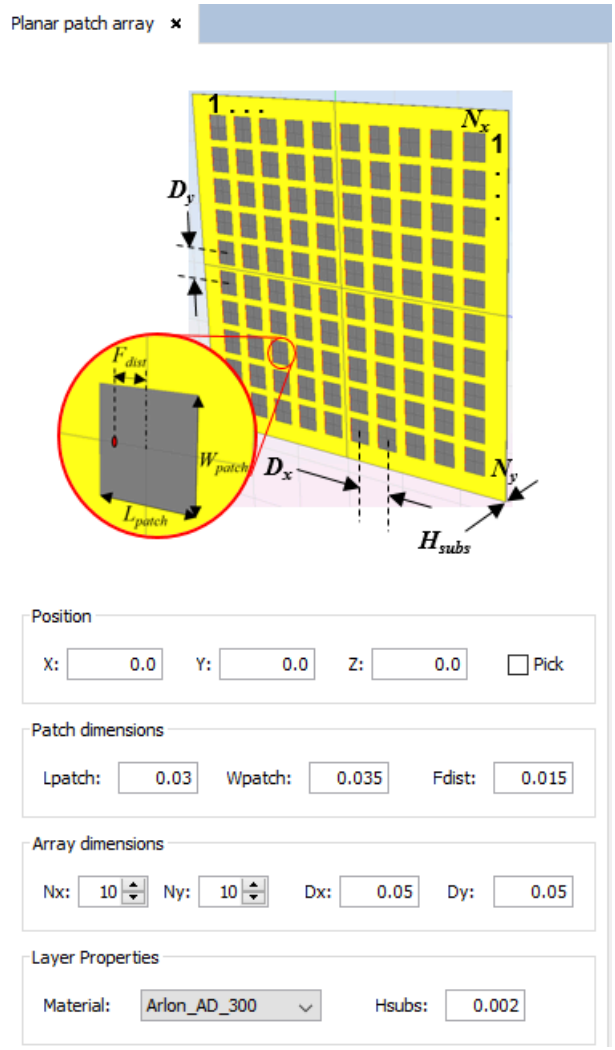


Figure 368: Planar patch array panel

- Position: set the Cartesian coordinates for the position of the antenna. For the antenna, user can select the position with the mouse by selecting the 'Pick' check box and clicking any point in the geometry panel.
- Patch dimensions: set the values of the length and width of the patches (Lpatch and Wpatch) and the position of the feeding point from the center of the patch (Fdist).
- Array dimensions: set the values of the number of patches in X (Nx), the number of patches in Y (Ny), the distance between patches in X (Dx) and the distance between patches in Y (Dy).
- Layer Properties: set the substrate material from the material library of newFASANT and the height of the substrate material (Hsubs).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

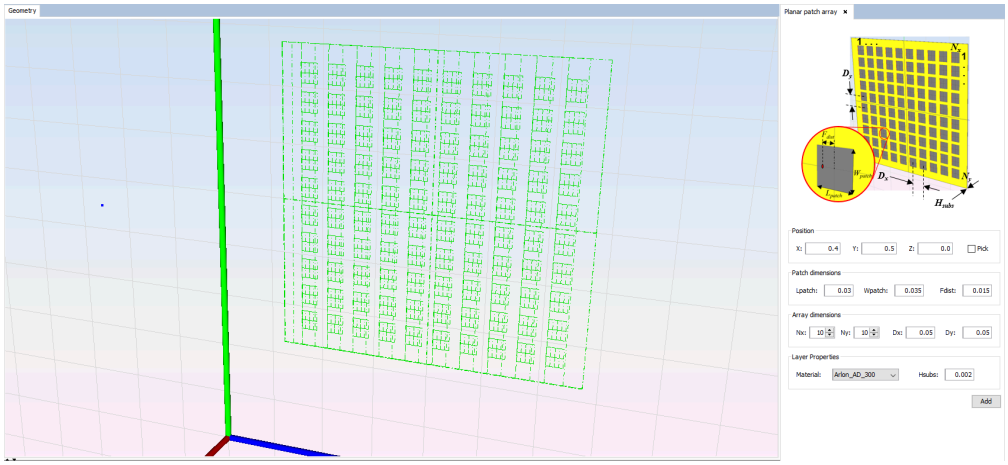


Figure 369: Planar patch array previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for every coaxial feed will be shown.

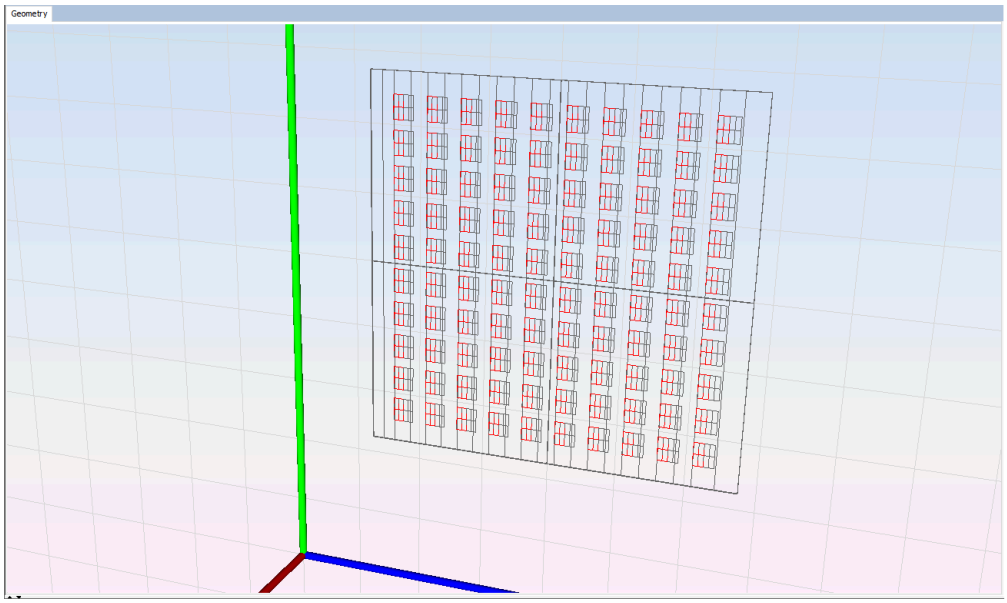


Figure 370: Planar patch array visualization

Circular Array

This section explains how to create a circular patch array antenna.

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

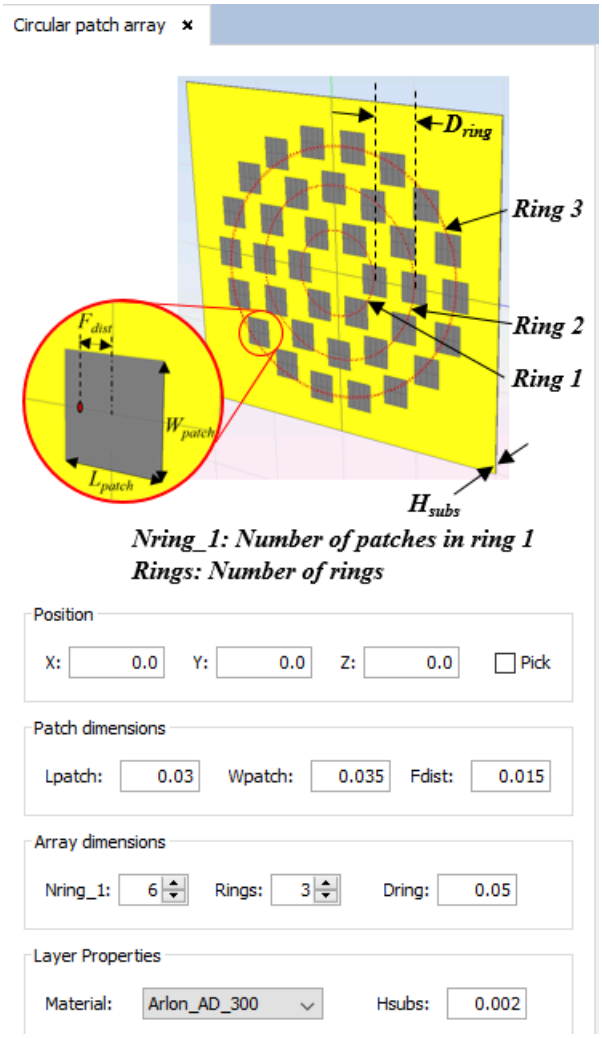


Figure 371: Circular patch array panel

- Patch dimensions set the values of the length and width of the patches (Lpatch and Wpatch) and the position of the feeding point from the center of the patch (Fdist).
- Array dimensions set the values of the number of patches in the first ring (Nring_1), the number of rings (Rings) and the distance between rings (Dring).
- Layer Properties set the substrate material from the material library of newFASANT and the height of the substrate material (Hsubs).

For each modification on parameters panel, this previsualization will be updated for the new parameters.

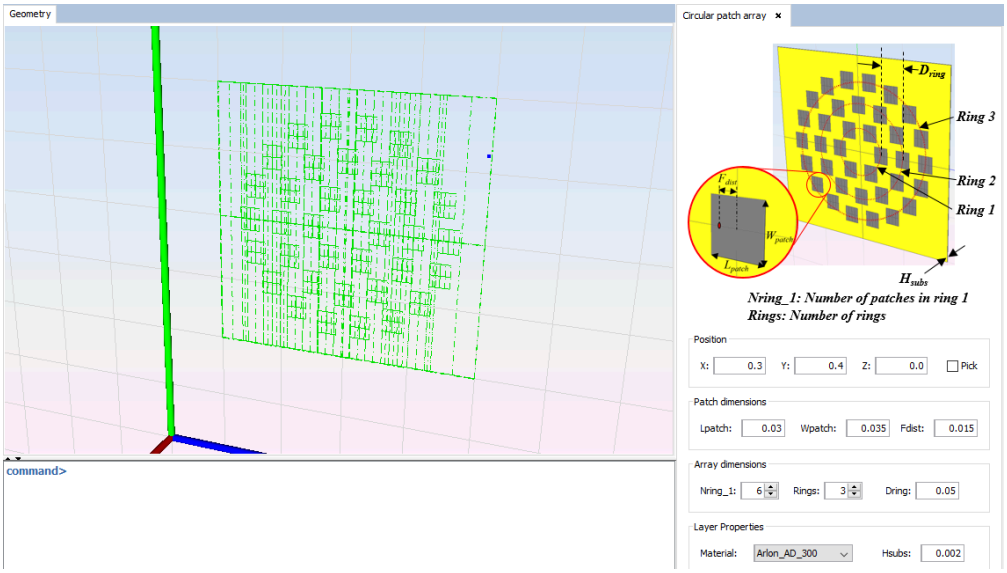


Figure 372: Circular patch array previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for every coaxial feed will be shown.

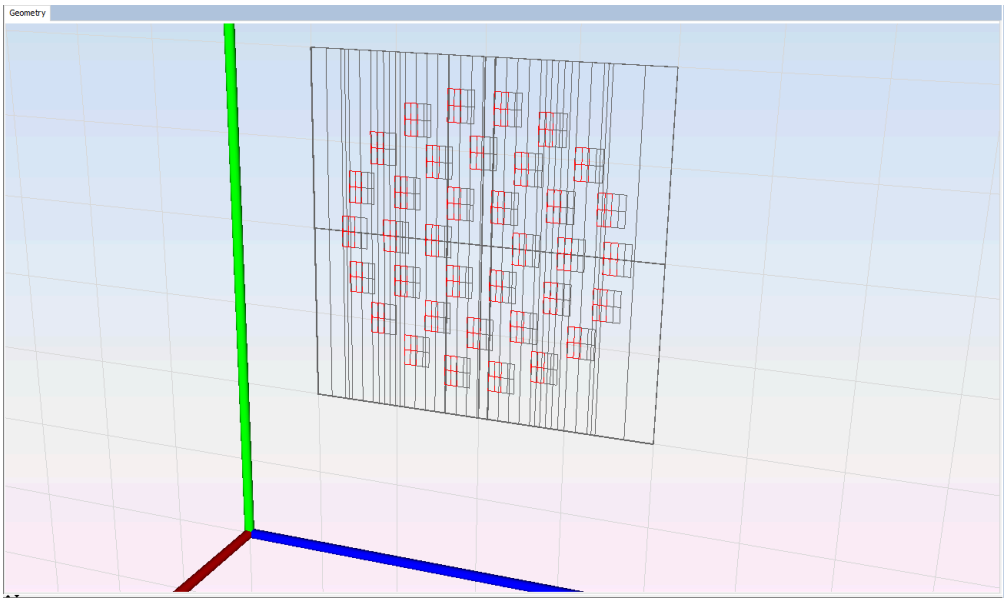


Figure 373: Circular patch array visualization

Planar Patch Array (Circular GND)

This section explains how to create a planar patch array antenna.

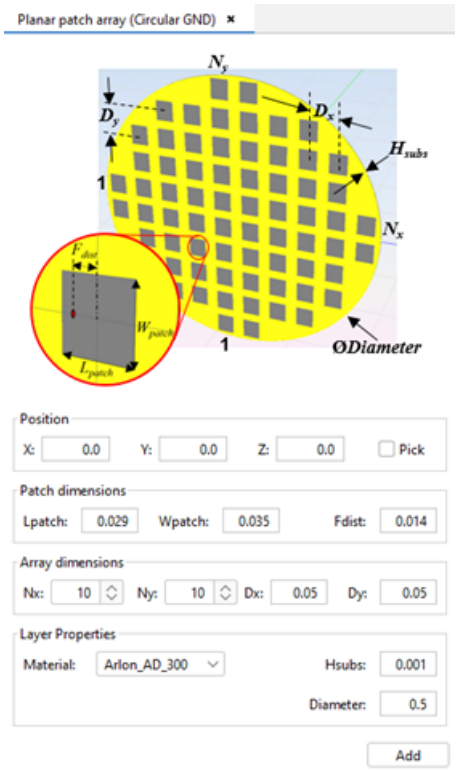


Figure 374: Planar Patch Array Panel

When the option is selected, the next panel and a previsualization of the antenna with the parameters on the panel are displayed.

- **Position**: set the Cartesian coordinates for the position of the antenna.
- **Patch dimensions**: set the values of the length and width of the patches (Lpatch and Wpatch) and the position of the feeding point from the center of the patch (Fdist).
- **Array dimensions**: set the values of the number of patches in X (Nx), the number of patches in Y (Ny), the distance between patches in X (Dx) and the distance between patches in Y (Dy).
- **Layer Properties**: set the substrate material from the material library of newFASANT and the height of the substrate material (Hsubs). The diameter specifies the longitude of the circumference.

For each modification on parameters panel, this previsualization will be updated for the new parameter.

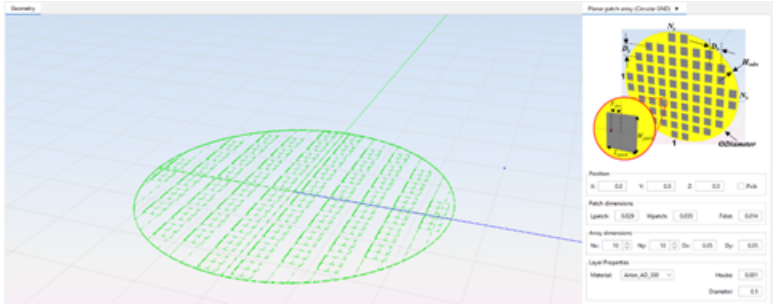


Figure 375: Planar Patch Array Previsualization

Clicking on 'Add' button, the final visualization for the antenna with a red dashed line for every coaxial feed will be shown.

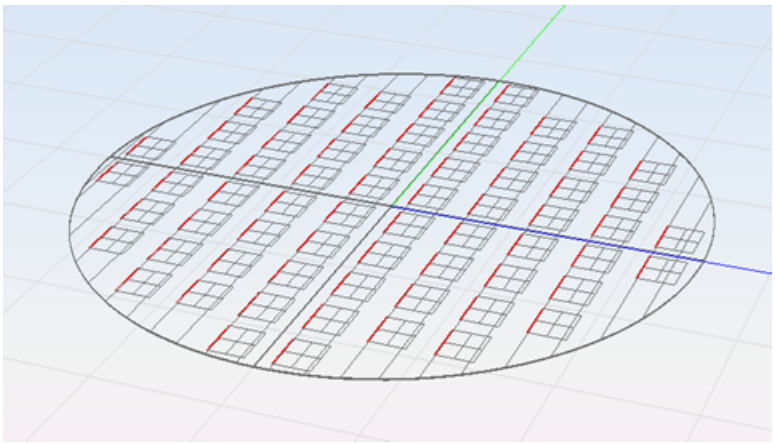


Figure 376: Planar Patch Array Visualization

4.8 Radome

The Radome menu contains three options: "Define Interface Radome", “Define Volumetric Radome” and "FSS Primitives".

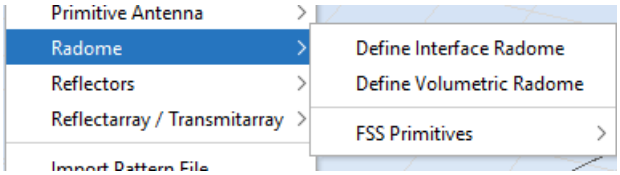


Figure 377: Radome menu

4.8.1 Define Interface Radome

The "Define Interface Radome" option allows the user to generate a radome, including interfaces and materials. Every radome layer is bounded by two surrounding interfaces and the material defined on the inner interface. Click on to add a new radome.

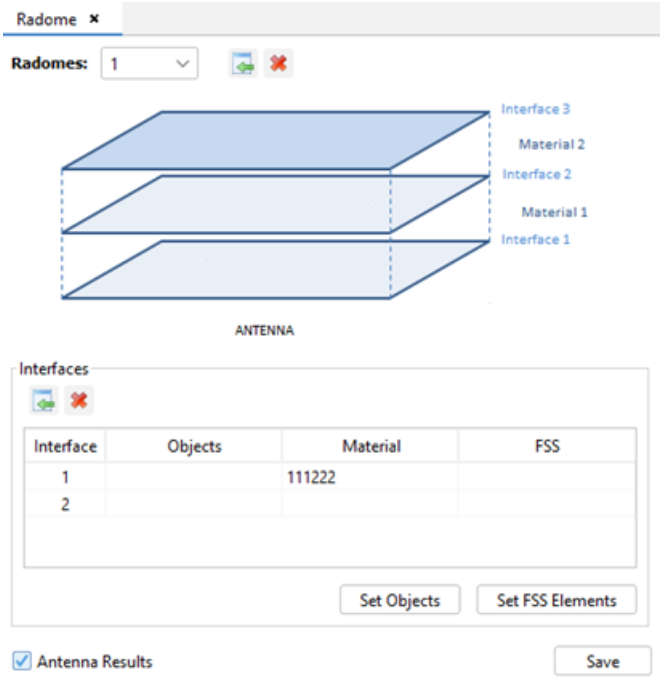


Figure 378: Interface Radome Definition Tab

"Interface 1" is the most inner one, select its geometry and select the interface on the table, then click on Set Objects.

Repeat the process with all interfaces. Then select the layer material clicking on the material cell.

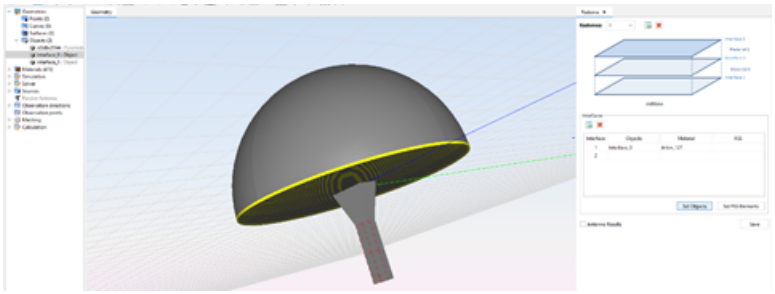


Figure 379: Set Objects on Interface 1

FSS geometry can be added to the interface. First, create the FSS geometry using arrayOnSurface command. Select its geometry and select the interface on the table, then click on Set FSS Elements.

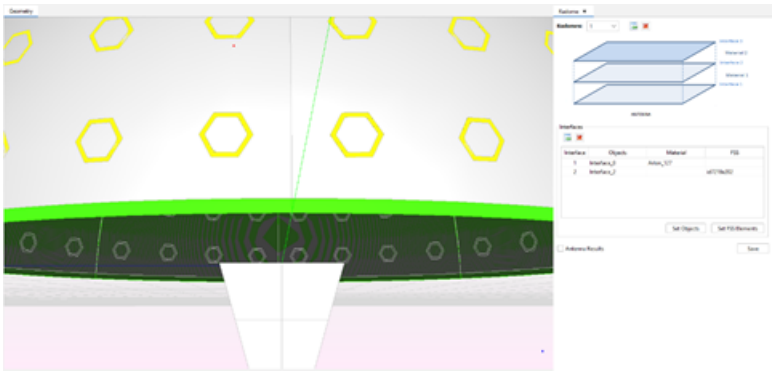


Figure 380: Set FSS Elements on interface 2

The "Antenna Results" checkbox enables meshing and calculating the antenna system without the radome structure. This option enables "Insertion Losses" and "Boresight Error" and allows to compare the results when the radome is considered or not.

Click on "Save" button to save the radome parameters.

4.8.2 Define Volumetric Radome

The "Define Volumetric Radome" option allows the user to generate a volumetric radome, including materials. Every radome layer is bounded by two surrounding interfaces and the material defined on the inner interface. Click on to add a new radome.

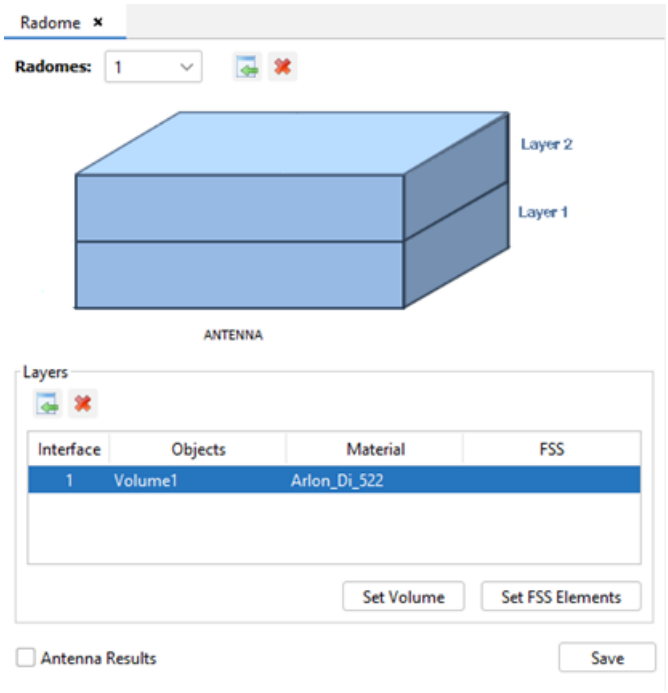


Figure 381: Volumetric Radome Definition Tab

Select the desired object (must be a valid geometric volume) and click the Set Volume button to add the Volume to the radome. Click on the material cell to specify the radome material. Aditionally the user can add FSS Elements in the same manner as in an Interface Radome as mentioned in the previous point.

The "Antenna Results" check box enables meshing and calculating the antenna system without the radome structure. This option enables "Insertion Losses" and "Boresight Error" and allows to compare the results when the radome is considered or not.

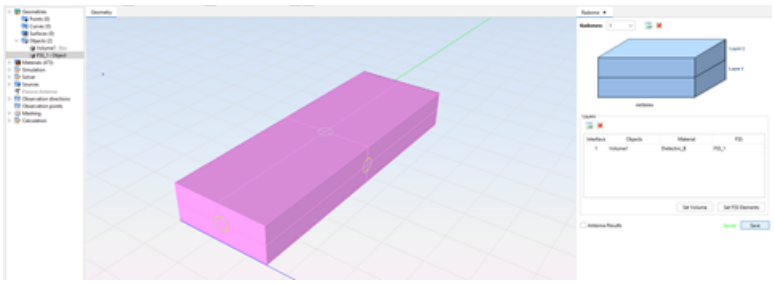


Figure 382: Set FSS Elements on Interface 1

4.8.3 Define Radome by Material Surface

Define Radome by One Material Surface

Input or set the antenna. Set the surface of the internal face of the radome. The radome will be simulated by modelling this surface by a material surface.

Figure 383: Antenna and an internal surface of the radome layers.

If the material for modelling the radome is not available, add a new material for modelling the radome layers and FSS. This material considers the reflection/transmission coefficients. A file with these coefficients is obtained using PERIODICAL STRUCTURE (PS) module with a unit cell that models the radome layers/FSS , see Cell Menu and Training Examples.

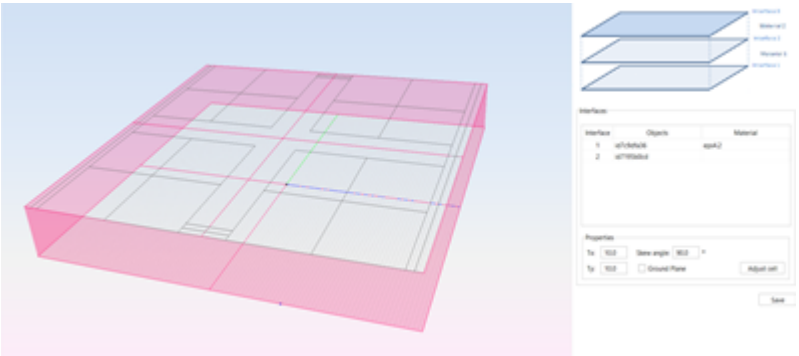


Figure 384: Example of the unit cell for modelling a radome surface composed by a dielectric layer of 1.4 mm of thickness, of material epsilon 4.2 that has a relative permittivity of 4.2 and is covered by a perfect electrical conducting surface with a cross-slot. Unit cell dimensions: 10x10 mm. Cross dimensions: total arms lengths = 9.25mm; wide = 1.20 mm. The upper face of the unit cell (metal with cross slot) corresponds with the internal face of the radome layer.

Get and output the Rx/Tx matrices in the PS module.

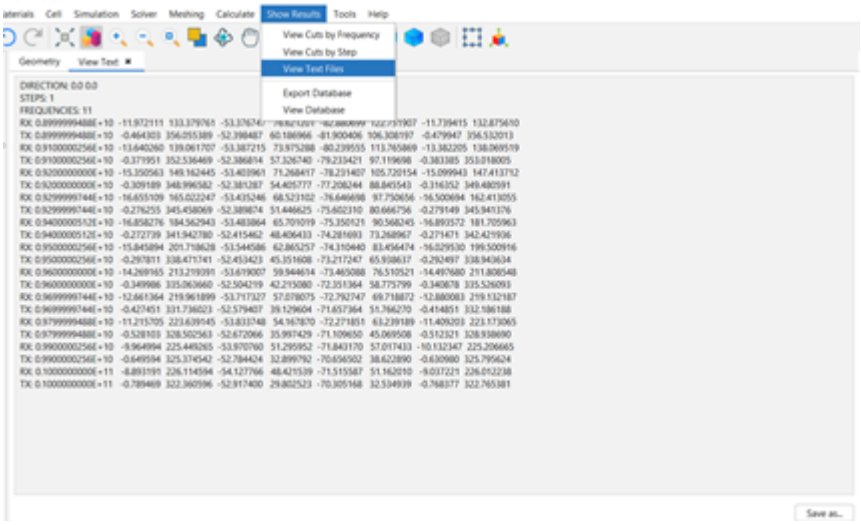


Figure 385: Save the file with the Rx/Tx matrices obtained in the frequency range from 9.0 to 10.0 GHz with 0.1 GHz Steps.

Define in MOM module a material with the Rx/Tx matrices generated in PS module. This material must be saved for future simulations.

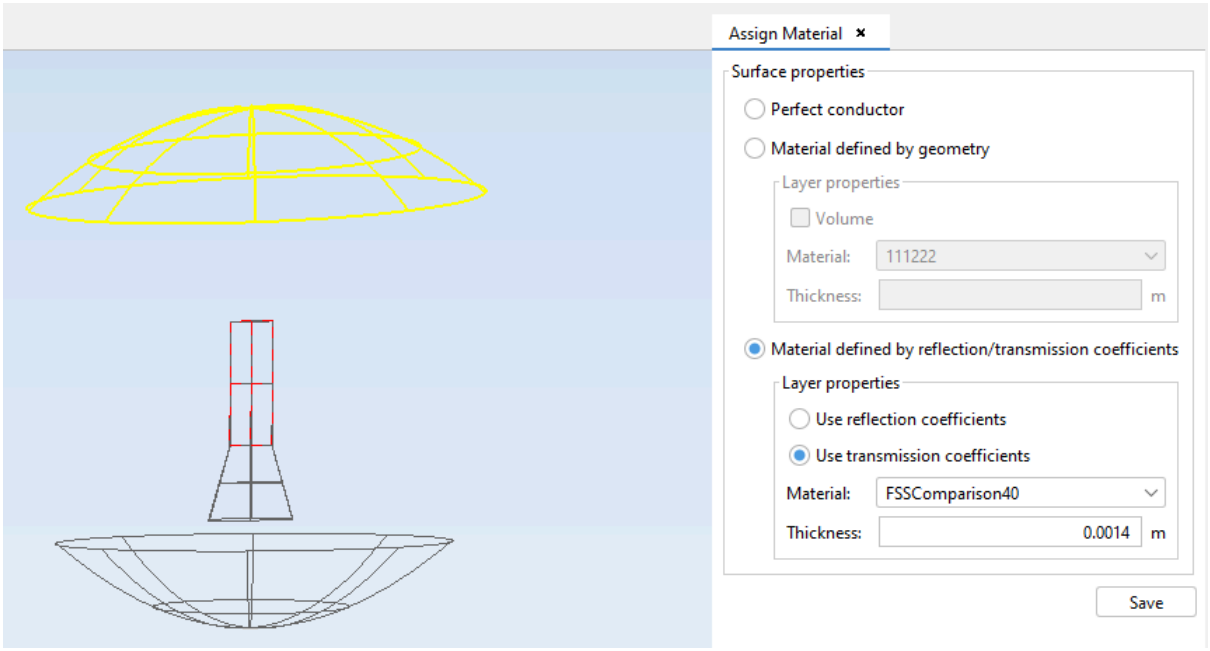


Figure 386: Assign the new material to the radome.

Select **Use transmission coefficients** and in the **Thickness** field, set the thickness of the unit cell.

Mesh and run the case.

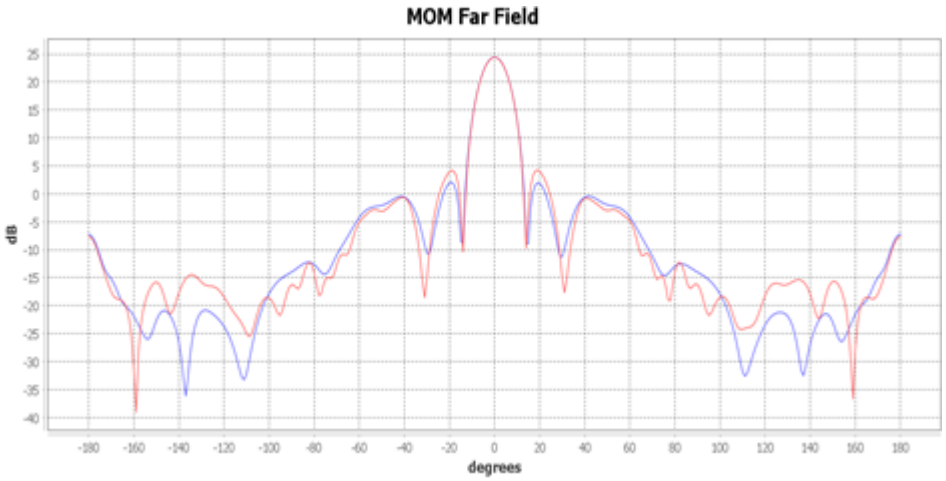


Figure 387: Assign Material to the radome surface.

Define Radome by Several Material Surfaces

User has the option to split complex thick layers into two or more layers structures, in such a way that these structures together compose the original structure. Figure 388 shows an example of this splitting.

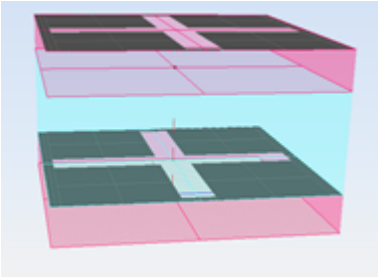


Figure 388: Example of the unit cell for modelling a radome surface composed, starting from the top by metal with cross slot, a dielectric layer of epsilon 4.2 and thickness 1.4 mm, a layer of foam (relative permittivity 1.05) of 3.6 mm of thickness, metal with cross slot and in the bottom a dielectric layer of epsilon 4.2 and thickness 1.4 mm. Notice that the total height of the unit cell is 6.4 mm. The surface at the top of the cell corresponds with the internal face of the radome.

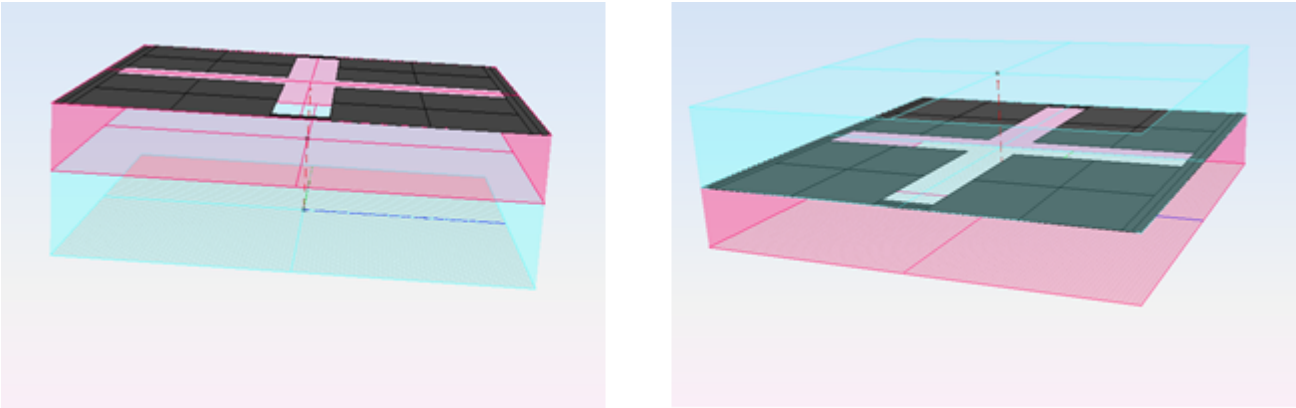


Figure 389: The radome layers is split into two unit cells: Unit cell (a) in the left is composed by, starting from the top by metal with cross slot, a dielectric layer of epsilon 4.2 and thickness 1.4 mm and a layer of foam of 1.8 mm of thickness; Unit cell (b) in the right composed by a layer of foam of 1.8 mm of thickness, metal with cross slot and a dielectric layer of epsilon 4.2 and thickness 1.4 mm.

The radome structure is modeled by a set of material surfaces, each one of them corresponding to a structure resulting from the splitting. Figure 390 shows the case of the structure splitting of previous figures.

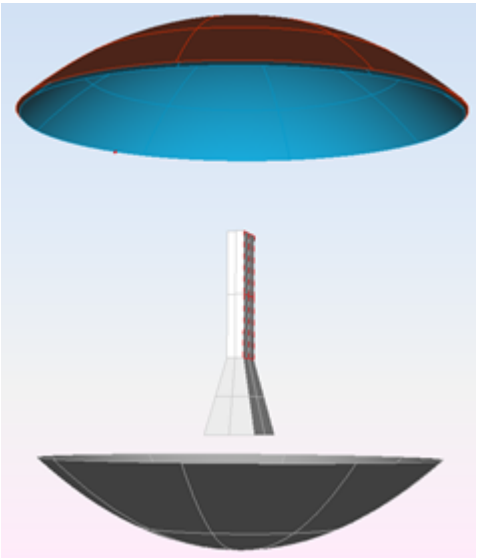


Figure 390: The radome is modeled by two material surfaces. The lower surface, of blue color has assigned a material that corresponds with unit cell (a) and the upper surface, of maroon color, corresponds with unit cell (b).

Figure 391 compares results for radiation pattern cuts with and without the splitting of the radome structure.

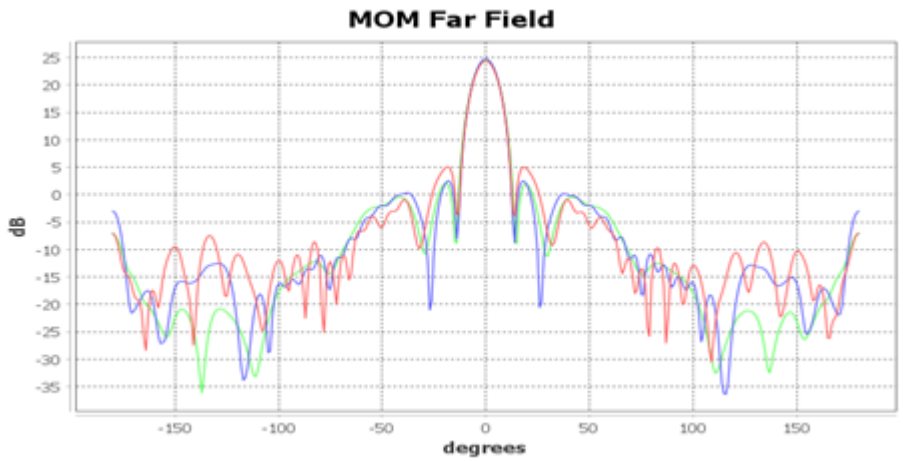


Figure 391: Radiation pattern cut, phi=0, at 9.5 GHz of the antenna with radome modeled by two material surfaces (red line), one material surface (blue line) and the antenna alone (green line).

4.8.4 FSS Primitives

In this menu the user can find a variety of periodical structures primitives to use in conjunction with the radome tool mentioned in the previous chapters.

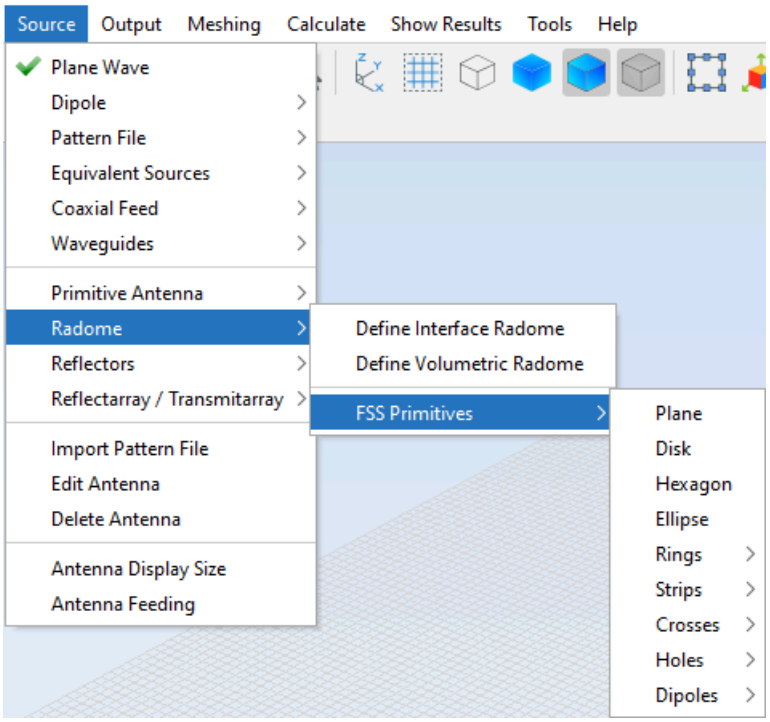


Figure 392: FSS Primitives Menus

4.9 Reflectors

This section will explain how to create different types of reflectors.

4.9.1 Simple Reflector

This option allows the user to create a paraboloid reflector. This is a reflector that has a paraboloid shape. It can be created by selecting Source > Reflectors > Simple Reflector on the menu bar or by issuing the command "simpleReflector".

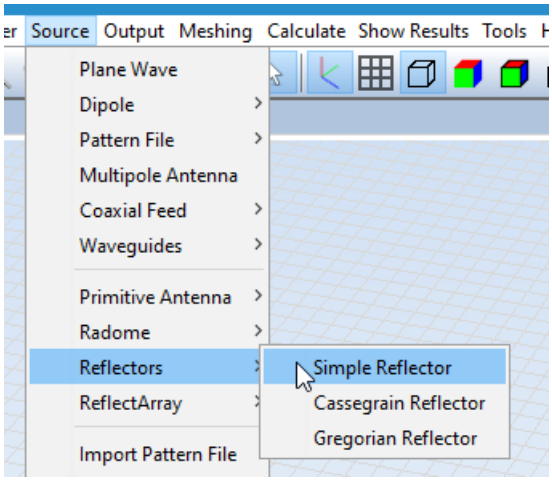


Figure 393: Simple Reflector menu item.

The following panel will be visible:

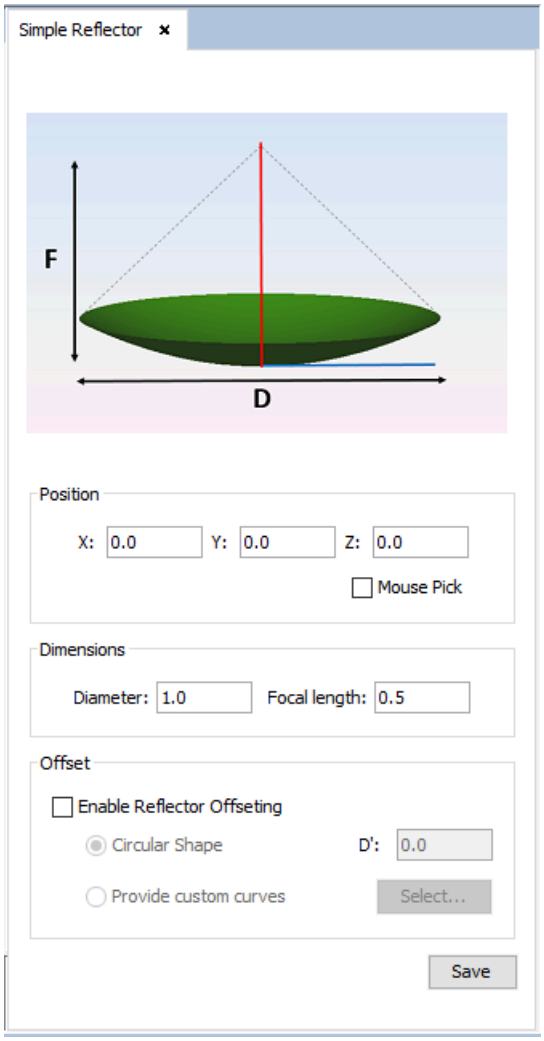


Figure 394: Add Simple Reflector

Using this panel the user is able to create and preview a simple reflector. The following options are available.

- Position where to place the reflector. The geometry center is located at the vertex point of the paraboloid. Therefore, when the position is set, it is done for the paraboloid vertex.
- Diameter the diameter for the paraboloid. This sets the diameter for the upper circle border of the paraboloid; or more concisely, the diameter of the circular projection made by the paraboloid itself.
- Focal length this sets the focal length of the paraboloid.
- Enable reflector offsetting by checking this option, the paraboloid will be offset. The user can provide the shape for the offset paraboloid. The circular shape is already provided as an option. The behavior of offset paraboloids is further detailed below.

While the data is being introduced, a preview will be displayed in the Geometry panel. Once the data is set, press Save to generate the final Geometry that will be added to the project.

As described above, the user can offset the paraboloid. The offsetting process will generate a different kind of paraboloid, as shown in the following figure.

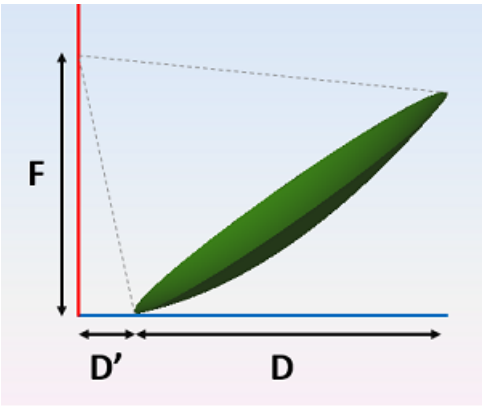


Figure 395: Offset paraboloid

Either a circle or a custom shape can be selected as offsetting primitives.

- Circular Shape is provided as a fast method to create offset paraboloids that still have a circular shape. The D' parameter sets the distance between the dish and the focal point.
- To create a custom offset geometry, a set of curves can be created and used as an offsetting cut. Select the curves and press the Select button on the panel to cut the dish to the curves projection.

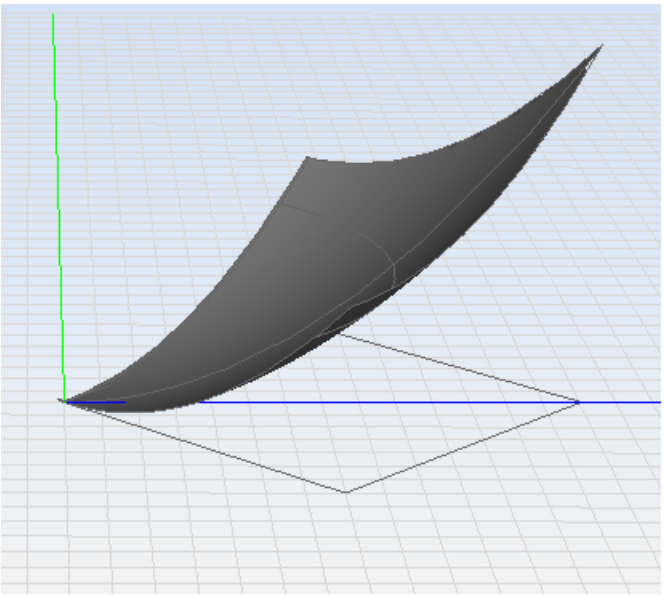


Figure 396: Paraboloid dish offset using custom curves

4.9.2 Cassegrain Reflector

To create a Cassegrain Reflector antenna, select Source > Reflectors > Cassegrain Reflector from the menu bar, or issue the command cassegrainReflector on the command line.

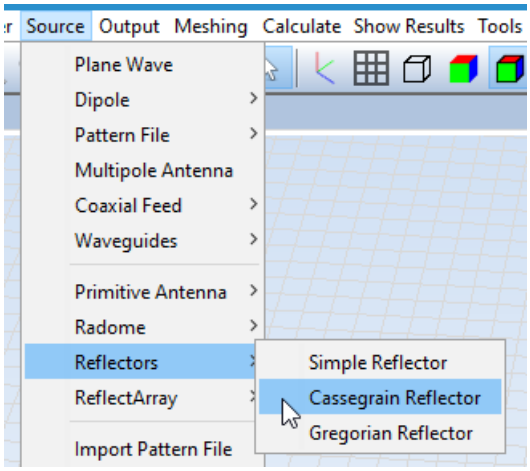


Figure 397: Cassegrain reflector menu item.

This will display the Cassegrain Reflector panel, used to create these reflectors.

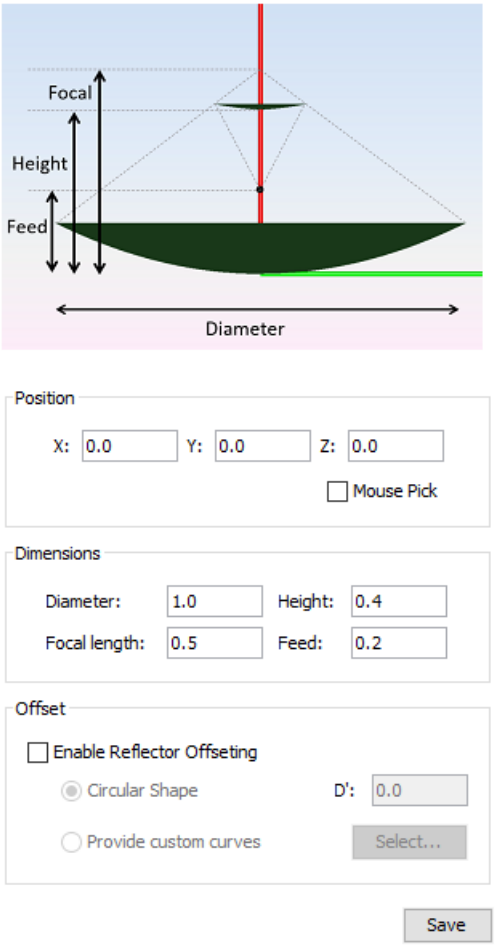


Figure 398: Cassegrain Reflector panel.

The following options can be given:

- Position the coordinates for the Cassegrain center. This is the geometrical point that will be used as a reference when the Cassegrain reflector is created. The center is located at the vertex from the main paraboloid reflector.
- Diameter this sets the diameter for the main paraboloid. This value sets the diameter of the circular projection of the main paraboloid reflector.
- Focal length this sets the focal length for the main paraboloid.
- Feed height this sets the height of the feed point. It is described as a height above the position center; so, as an example, if the position is **(x, y, z)** and the feed height is **p**, the actual feed point will be **(x, y, z+p)**.
- Height this is the height the secondary hyperboloid reflector will be placed.

Additionally, the user is able to select if they want to create a front-fed Cassegrain reflector or an offset-fed one. By default, a front-fed Cassegrain reflector system will be created, and by checking the "Offset-fed" option the user can create an offset-fed Cassegrain reflector.

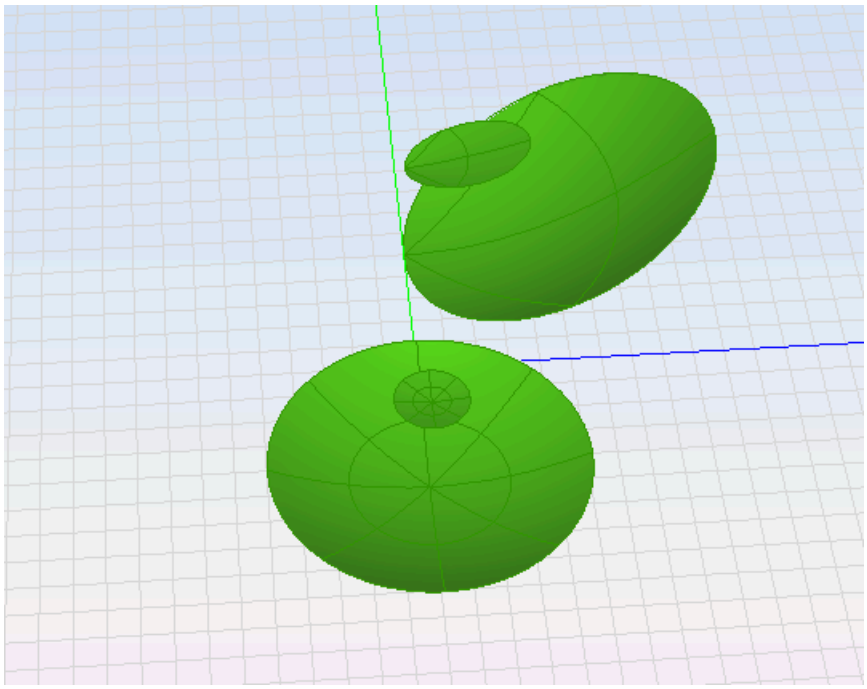


Figure 399: An offseted Cassegrain reflector and a simple Cassegrain reflector.

There are two ways of creating an offset-fed Cassegrain reflector:

- Circular shape Selecting this option will cause the sub reflector to have a circular projection on the XZ plane. This mode of creating the reflector requires an additional parameter the separation (also called D'), along the X axis, between the left border of the main reflector and the vertex of the paraboloid it is extracted from.
- Custom shape This option allows the user to customize the shape of the main reflector (and consequently, the shape of the secondary reflector) by selecting a closed set of curves that define the projection of the main reflector on the XZ plane. To do this, the user needs to select the "Custom shape" option, select the set of curves and press the "Select curves" button (the preview will be updated if the selected curves are adequate).

In order to create the geometry of the Cassegrain reflector system, the user needs to press the "Save" button.

4.9.3 Gregorian Reflector

This option allows the user to create a Gregorian reflector system. When this option is selected, or the "gregorianReflector" command is called without parameters, the following panel will be shown:

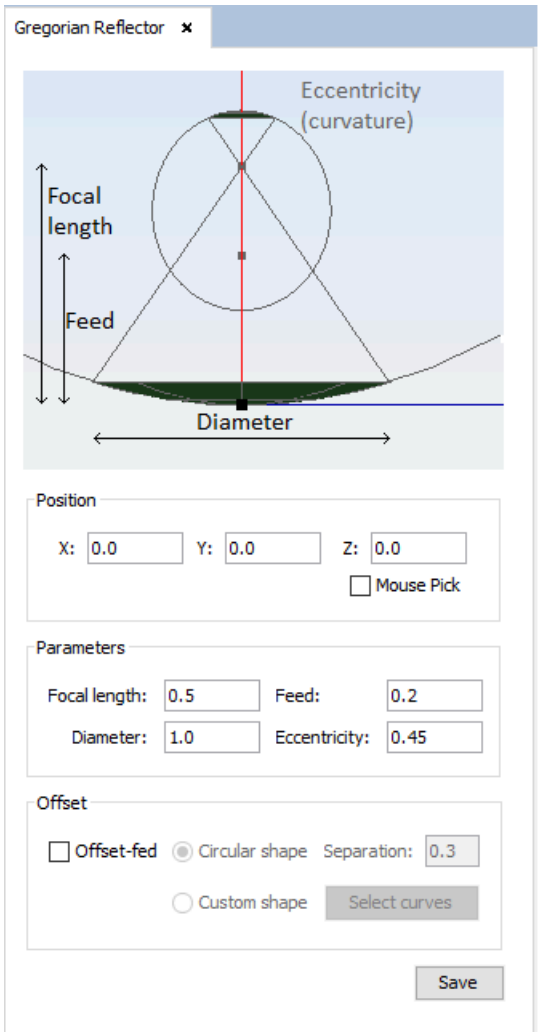


Figure 400: "Add Gregorian Reflector" panel

Additionally, a preview of the antenna (using the current parameters of the panel) is shown in the geometry panel. This preview will be shown in green and will be updated every time a parameter is modified in the panel.

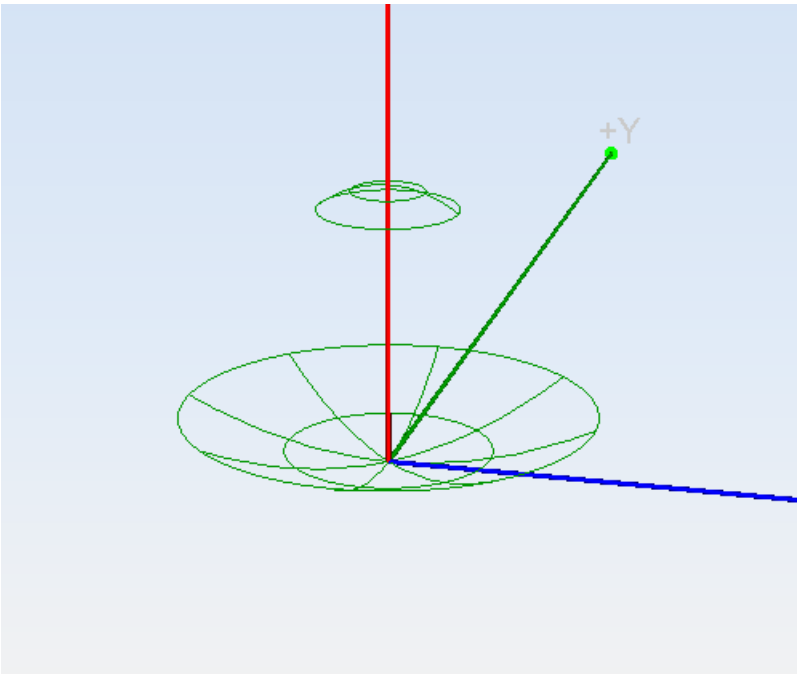


Figure 401: Preview of the Gregorian reflector system in the geometry

A Gregorian reflector system is composed of a main (paraboloidal) reflector and a secondary (ellipsoidal) reflector. The geometry of the main reflector is a portion taken from a paraboloid with its axis pointing in the same direction as the Z axis. The portion is determined by the offset the main reflector of a front-fed reflector system (no offset) will be centered at the vertex of the paraboloid. In an offset-fed reflector system, the main reflector will be a portion of the paraboloid separated by a distance D' from the vertex of the paraboloid.

On the other hand, the geometry of the secondary reflector is defined by the geometry of the main reflector and the eccentricity of the ellipsoid the secondary reflector is based on.

The following parameters define a Gregorian reflector system:

- **Position** This point will be the vertex point of the paraboloid that defines the main reflector. Note that the vertex point does only match the geometric center of the main reflector if the Gregorian reflector is front-fed (i.e. no offset).
- **Focal length** Defines the distance, along the positive Z axis, from the vertex point to the focal point of the paraboloid. This point will also be one of the foci of the ellipsoid.
- **Feed** Defines the distance, along the positive Z axis, from the vertex point to the point where the antenna feed should be located. This distance should be smaller than the focal length. The point defined by this parameter will be the second focus of the ellipsoid.
- **Diameter** Defines the diameter of the main reflector. Note that this is the diameter of the circular projection of the main reflector on the XY plane (projected along the Z axis).
- **Eccentricity** Defines the eccentricity of the ellipsoid used to create the secondary reflector. Therefore, this parameter will affect the curvature of the secondary reflector (values closer to 0 will cause the secondary reflector to be more spherical-shaped, while values closer to 1 will cause the secondary reflector to be more ellipsoidal-shaped).

Additionally, the user is able to select if they want to create a front-fed Gregorian reflector or an offset-fed one. By default, a front-fed Gregorian reflector system will be created, and by checking the "Offset-fed" option the user can create an offset-fed Gregorian reflector:

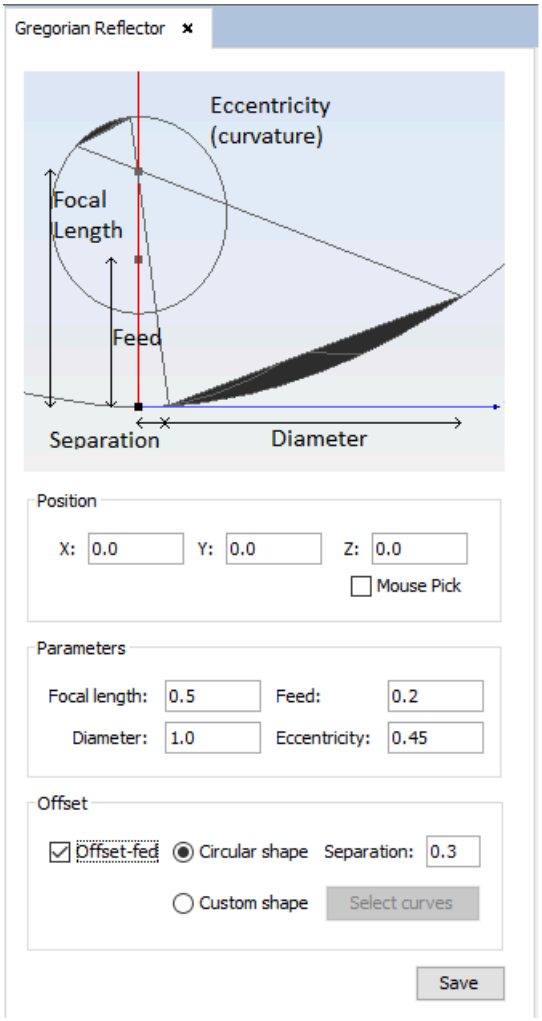


Figure 402: "Add Gregorian Reflector" panel when the "Offset-fed" option is selected

There are two ways of creating an offset-fed Gregorian reflector:

- **Circular shape** Selecting this option will cause the subreflector to have a circular projection on the XZ plane. This mode of creating the reflector requires an additional parameter the separation (also called D') along the X axis between the left border of the main reflector and the vertex of the paraboloid it is extracted from.

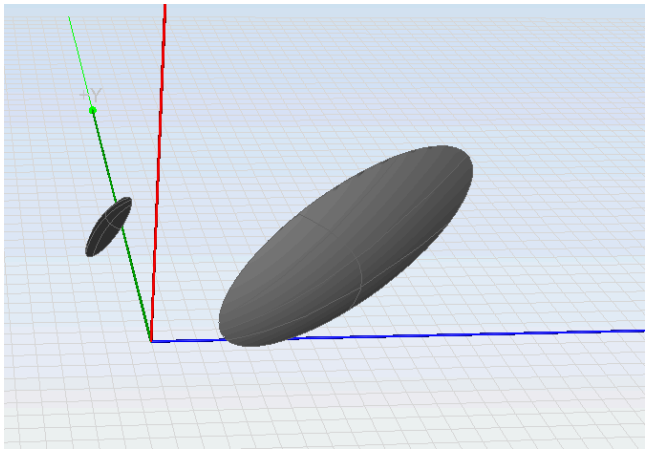


Figure 403: Offset-fed Gregorian reflector with circular shape

- **Custom shape** This option allows the user to customize the shape of the main reflector (and consequently, the shape of the secondary reflector) by selecting a closed set of curves that define the projection of the main reflector on the XZ plane. To do this, the user needs to select the "Custom shape" option, select the set of curves and press the "Select curves" button (the preview will be updated if the selected curves are adequate).

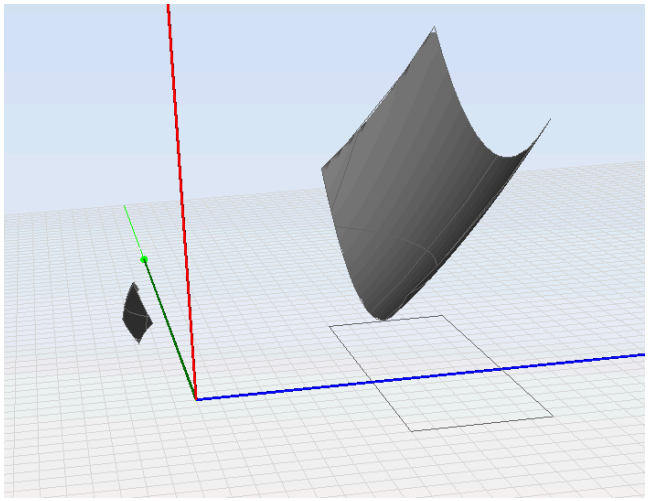


Figure 404: Offset-fed Gregorian reflector with custom shape, and the curves used to define the shape

In order to actually create the geometry of the Gregorian reflector system, the user needs to press the "Save" button.

4.9.4 Equivalent Antenna

This option allows the user to create a reflector with a rectangular aperture area and with the size to give the selected gain. When this option is selected, or the "equivalentAntenna" command is called without parameters, the following panel will be shown:

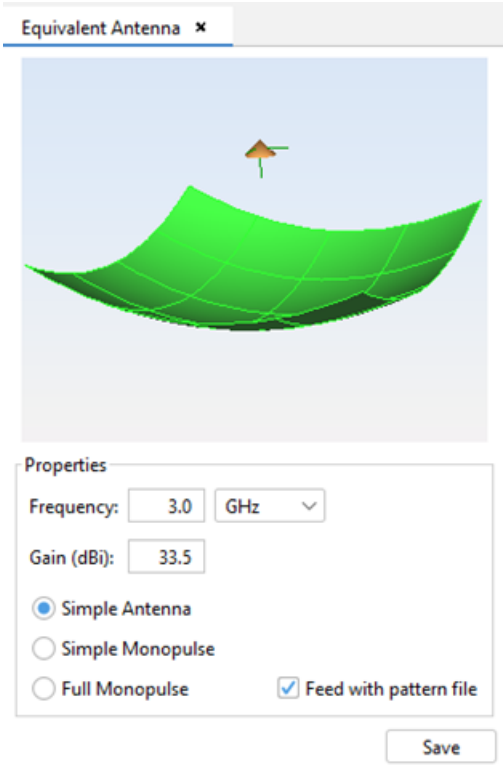


Figure 405: Equivalent Antenna Panel

- **Frequency**: Specify the antenna's working frequency, given in the desired unit. (By default will be in GHz).
- **Gain**: Indicates the antenna's resulting gain, in dBi.
- **Simple Antenna**: Creates a simple antenna with a feeding point over it.
- **Simple Monopulse**: Create two rectangular reflectors for simple monopulse (only one plane) applications.
- **Full Monopulse**: Create four rectangular reflectors for full monopulse (two planes) applications.
- **Feed with pattern File**: Indicate the antenna feeding via `.dia` file (diagram). If selected, a new `.dia` file will be generated alongside the antenna.

Additionally, a preview of the antenna (using the current parameters of the panel) is shown in the geometry panel. This preview will be shown in green and will be updated everytime a parameter is modified in the panel.

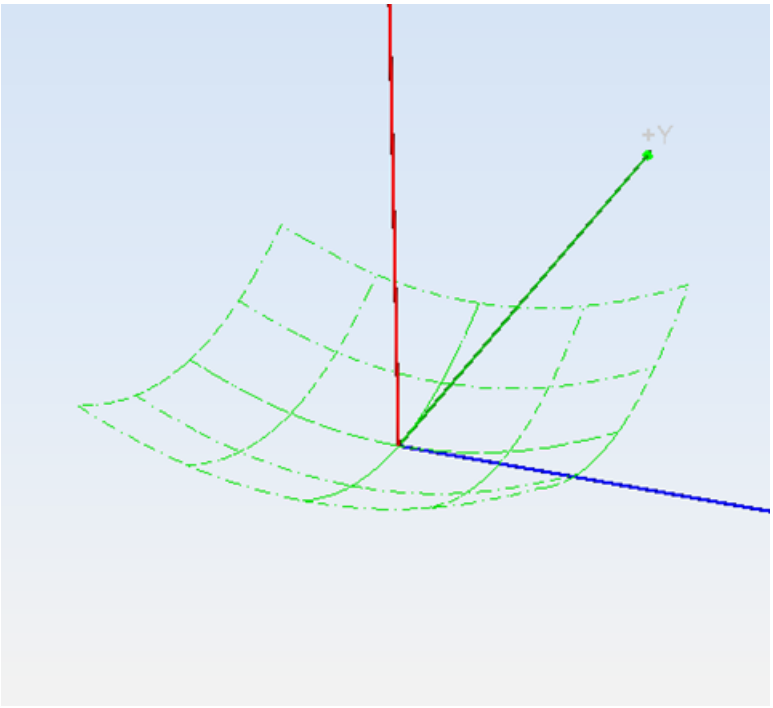


Figure 406: Equivalent Antenna Previsualization (Simple Antenna)

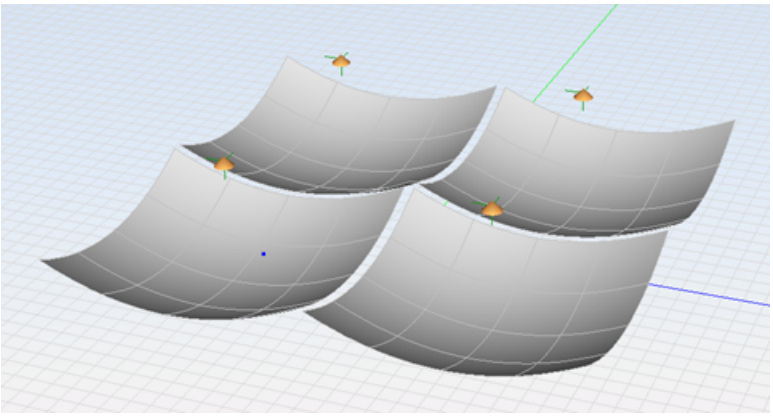


Figure 407: Equivalent Antenna Visualization(Full Monopulse)

4.10 Reflectarrays

4.10.1 New Layout

This option allows the user to create a new reflectarray or transmitarray layout. Three elements are required to create a reflectarray layout:

- Database

The database contains all the geometric cells available for the reflect array generation, together with their field values by frequencies and components. It must have been created by using the Periodical Structures module (see Annex 1: Creating a Reflectarray Database for an example).

- Target surfaces.

Single layer that defines the real shape of the top reflect array layout the upper interface of the reflect array will be placed on the target surfaces, and the next interfaces will be automatically generated. Both planar and curved surfaces are allowed, but the generation for planar surfaces is faster. In the case of using curved surfaces, a soft curvature is recommended.

The layout is generated by considering a planar grid on the current XY system (see Reference Plane) and projecting it onto the target surfaces with the reference Z direction, so it is important to have a good agreement between the surfaces orientation and the reference system.

After the layout generation, the original surfaces are deleted.

- Cell Selection Function.

The best available cell is assigned at a point of the grid on the real geometry, considering as the best cell the one that provides the closest phase to the given by the selection function.

The user may set its own selection function programmed in Java. The next variables can be read from a custom function, all of them with their name starting with the "$" character, and obtained from the reflectarray generation parameters:

- $freq Working frequency, given in Hz.
- $x, $y, $z Cell position where the function is evaluated.
- $focusX, $focusY, $focusZ Focus position.
- $theta, $phi main beam direction, where the reflectarray must be pointing to.

The program function has to return a double value corresponding to the optimal phase at the given point, in degrees.

Include the customized functions in the **/functions/** directory corresponding to the installation path.

See further information about the custom functions programmatic in User Functions.

The fields for the layout generation are explained in the following.

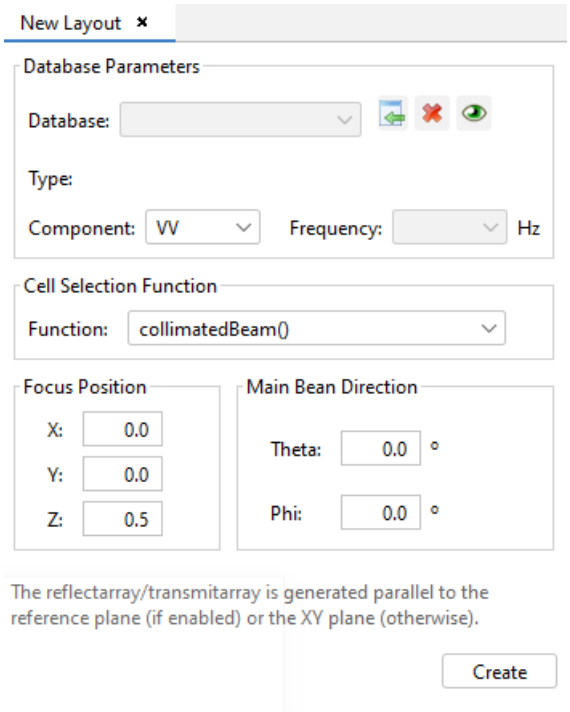


Figure 408: New Layout Parameters tab

- **Database**: Shows the selected database file to create the new layout. With the button a file chooser window will prompt the user to add a database file. Opens a database information tab on the main panel of the application. With this button, it is possible to visualize all the relevant parameters of the database: cell dimensions, cell replications, material considered, skew angle, geometrical models, plots of the phase cells,…The button detaches the specified database from the layout.
- **Component**: Allows the user to select the desired polarization component of the database to generate the layout.
- **Frequency**: Allows the user to select the desired frequency of the database to generate the layout.
- **Function**: Allows the user to select the desired user function to assign one cell of the database to every position of the base geometry.
- **Focus Position**: Allows the user to define the point of the source.
- **Main Bean Direction**: Allows the user to define the output direction of the main beam.

Is possible to specify more than one database in order to create more complex layouts. In order to do so, user must specify the parameters of each database file separately.

When all parameters have been specified for each database, clicking the Create button will generate the desired reflectarray/transmitarray layout.

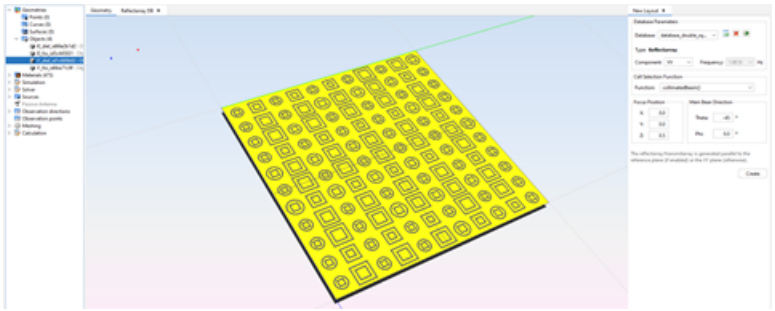


Figure 409: Multi database ReflectArray Layout

4.11 Import Pattern File

This section explains how to import one or more radiation pattern files to the current project. This files can be used for the definition of pattern file antennas.

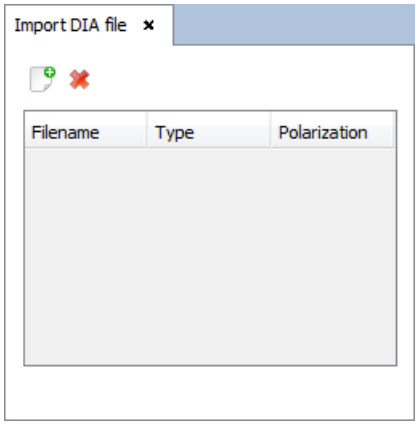


Figure 410: Import Pattern File panel

- Adds a new radiation pattern file selecting the file on a panel.
- Delete the selected radiation pattern file.

newFASANT allows the user to create their pattern files (`.dia`) and edit existing files. To see more information about creating or editing pattern files see DIA Files.

4.12 Edit Antenna

This section explain how to edit an antenna.

Selecting this option with a previous selection of an antenna will show a panel with editable parameters of the antenna.

An alternative way to edit an antenna is selecting the "Edit" option under the context menu of the antenna.

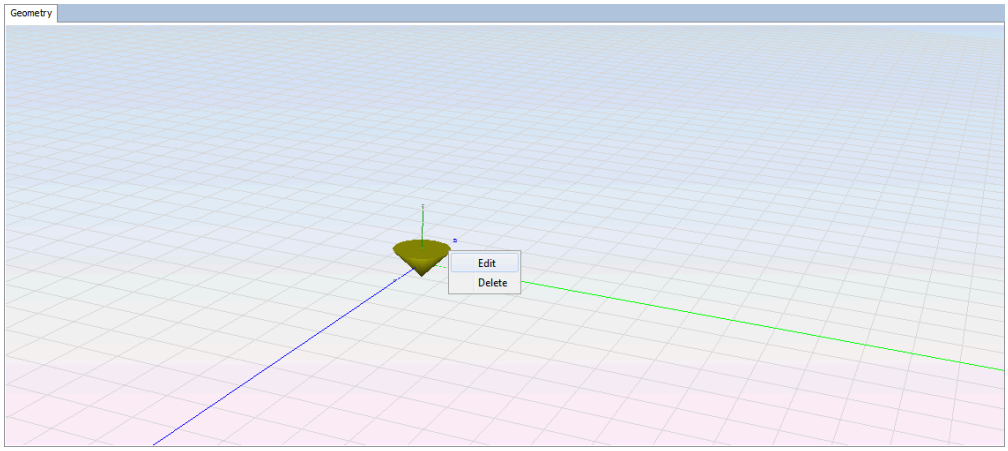


Figure 411: Context menu of an antenna

4.13 Delete Antenna

This section explain how to delete an antenna.

If the user selects an antenna in the geometry panel and selects this option, the antenna will be deleted.

An alternative way of deleting an antenna is selecting the "Delete" option in the context menu of the antenna.

4.14 Antenna Size

This parameter allows changing the size for displaying the logical antennas, such as the dipole antenna (both active and passive), multipole antennas, 3D pattern... It just modifies the visualization size of the antennas that is in the magnitude of the selected value, but the radiating behavior keeps invariable.

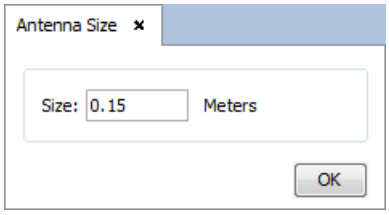


Figure 412: Antenna Size

4.15 Antenna Feeding

When the user selects "Antenna Feeding" in the Source menu, the following panel is shown:

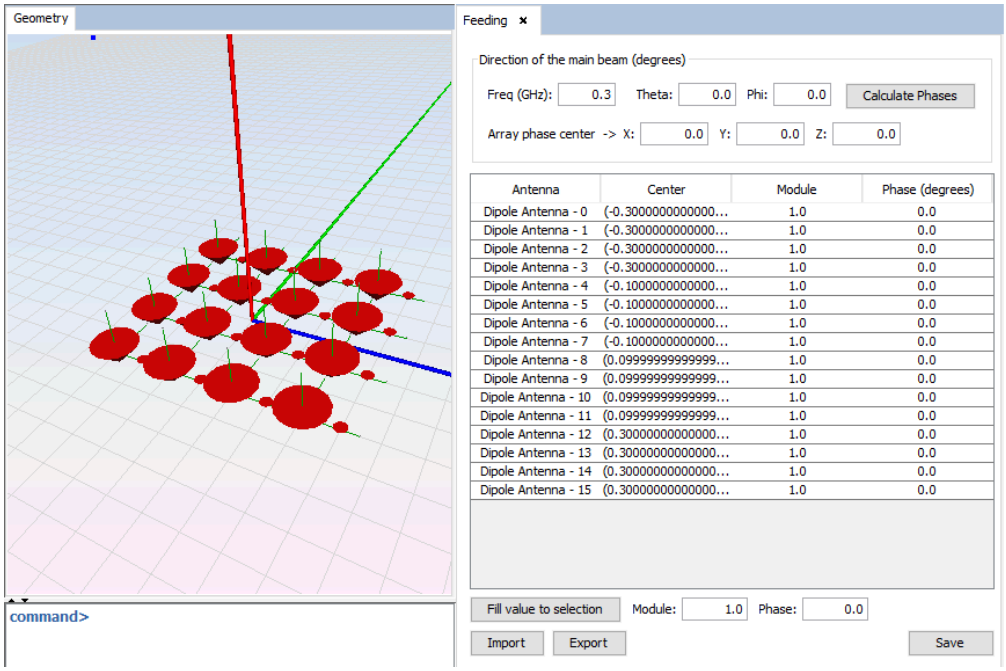


Figure 413: Antenna Feeding panel

This panel allows the user to modify the antenna feeding for each individual antenna (even if it is part of an array). The table shows the position, module and phase of each antenna, all of which can be modified independently.

In the panel labelled as "**Direction of the main beam (degrees)**", the user can set global parameters for the feeding of all antennas. By pressing the "Calculate Phases" button, the application will calculate the module and phase for each antenna based on the given array phase center, frequency and angle of the beam (both theta and phi must be given in **degrees**).

It is also possible to set an specific module and phase to an antenna or group of antennas. To do this, the user should fill the "**Module**" and "**Phase**" textboxes below the table, select the row or rows of the table they want to modify and press the "Fill value to selection" button.

In order to export a text file with the list of modules and phases of the antennas, the user needs to click the "**Export**" button. Reciprocally, in order to import a text file with the list of modules and phases, the user needs to click the "**Import**" button.

The format of the input/output text files used by the import/export functionality is the following:

```
<Number of antennas (N)>
<Module of antenna 1> <Phase of antenna 1 (degrees)>
<Module of antenna 2> <Phase of antenna 2 (degrees)>
...
<Module of antenna N> <Phase of antenna N (degrees)>
```

Materials

5

Materials can be defined, edited, deleted and assigned to specific surfaces or objects.

This chapter covers the following:

- 5.1 Materials Menu (MOM Module) (p. 386)
- 5.2 Materials Menu (MONCROS Module) (p. 394)
- 5.3 Materials Menu (PO Module) (p. 400)
- 5.4 Materials Menu (GTD and GTD-PO Modules) (p. 408)
- 5.5 Materials Menu (Periodical Structures Module) (p. 415)
- 5.6 Materials Menu (US Module) (p. 418)
- 5.7 APPENDIX A: Color Selection (p. 424)

5.1 Materials Menu (MOM Module)

In the following sections, all the options of the materials menu will be discussed:

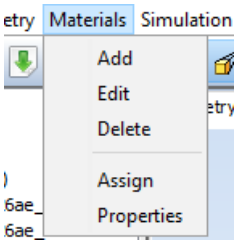


Figure 414: Materials Menu

5.1.1 Add

After selecting the Add option the following panel will be shown on the right side of the window:

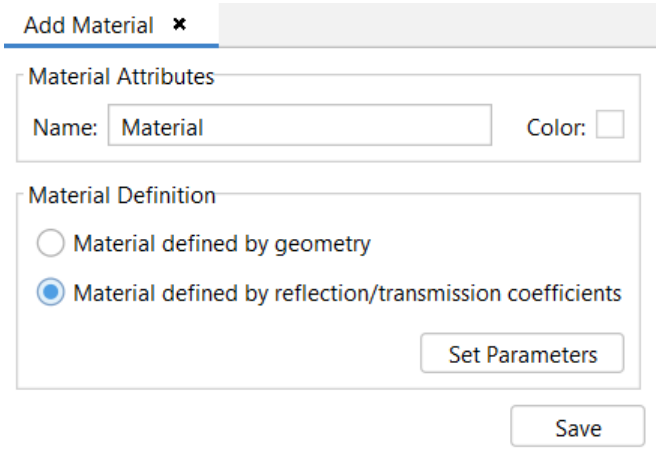


Figure 415: Add Material panel

This option allows the user to create a new material. The following options are available:

Material Attributes-Name: The user may want to provide a name for this material for future reference.

Material Attributes-Color: For more information, see APPENDIX A: Color Selection.

Material Definition - Material defined by geometry: Used to define a material according to its physical properties. After clicking on the Set Parameters button, two different options are available in the **Material defined by Geometry** window:

Constant

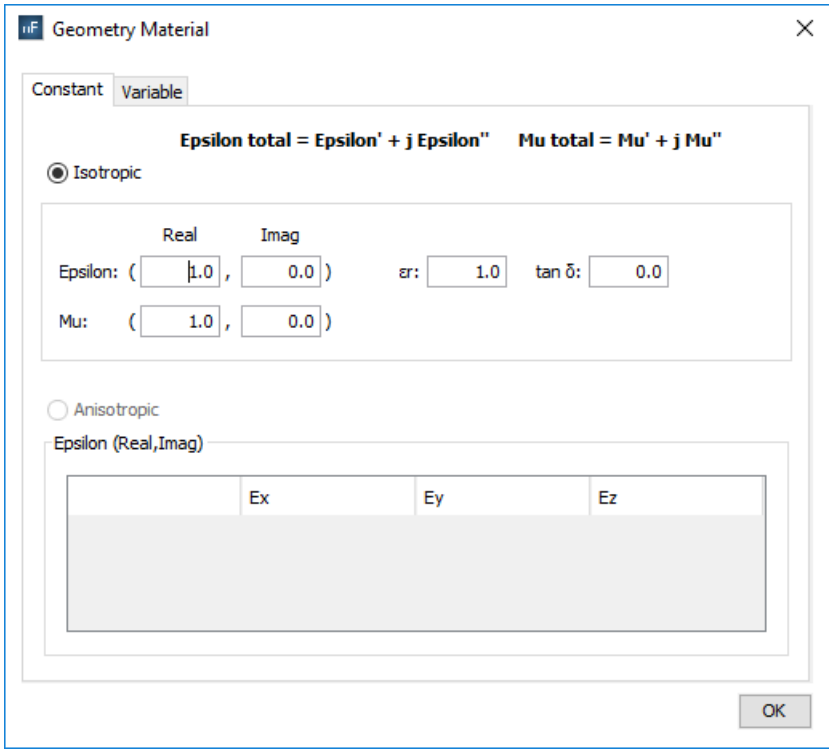


Figure 416: Material defined by geometry

§ **Real part**: common values are greater than or equal to 1.0.

§ **Imaginary part**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

o **Isotropic**: Selecting this option sets the epsilon (e' - permittivity) and mu (μ' - permeability) properties immutable. The user will need to specify both real and imaginary components of both properties or loss tangent and relative permittivity. Common real values for epsilon and mu are greater of equal to 1.0. If the imaginary part is negative, the material is considered to have losses; otherwise, it's considered to be a material with gain.

o **Anisotropic**: This option allows the user to set the material properties depending on the volumetric subdomain direction (i.e. the dielectric coefficient may change depending on the direction of the subdomains). The epsilon parameter consists of a 3x3 matrix where all Cartesian combinations can be modified. The materials using this option are only designed to be applied to volumes.

Variable

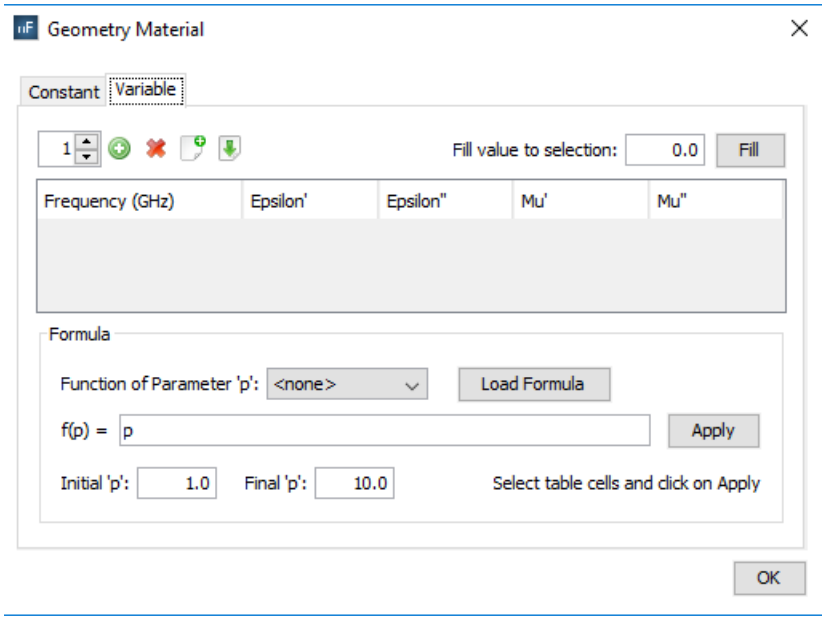


Figure 417: Material defined by geometry, tab 'Variable'

To assign the physical properties of the material according to the frequency.

§ **Frequency (in GHz)**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.

§ **Real part (')**: common values are greater than or equal to 1.0.

§ **Imaginary part (")**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

Figure 418: Add new frequencies to the material properties

§ **Remove**: to remove existing frequencies to the material properties

§ **Import/Export**: to import table with values from a text file or export the table into a text file.

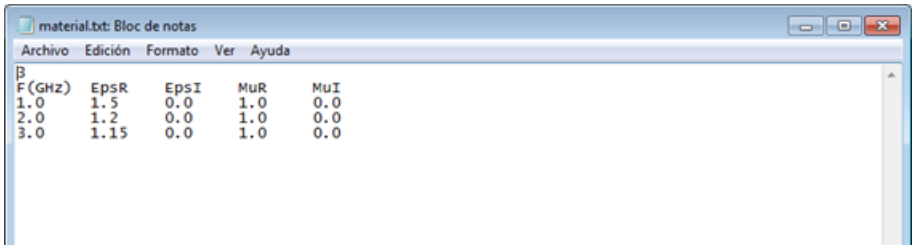


Figure 419: File Format

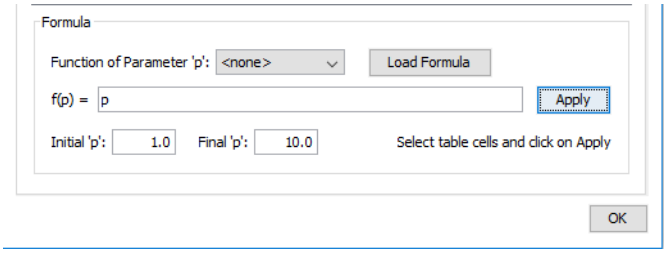
The information of this file can be modified by the user. Be sure not to modify the format and not to introduce any additional line break.

*Figure 420: The **Fill value to selection** option*

§ **Fill value to selection**: This option allows the user to automatically change the value of the selected cells of the table with the inserted value

These parameters can also be set using formulas in function of the frequency values.

§ **Formula:**



This option serves for completing the selected cells of the table with values depending on the selected function. After choosing the function that we want to apply, pressing the Load Formula button will load the formula *f(p)* in the field. The next step is choosing the initial and the final value for the function. The final step is selecting the cells that we want to apply the function to and press the button Apply. The cells will be auto-completed with the values created by the chosen function starting with the initial value and ending with the final one.

Material Definition - Material defined by reflection/transmission coefficients: Used to define a material according to its reflection coefficient. After clicking on the Set Parameters button, a window appears to define the reflection coefficient:

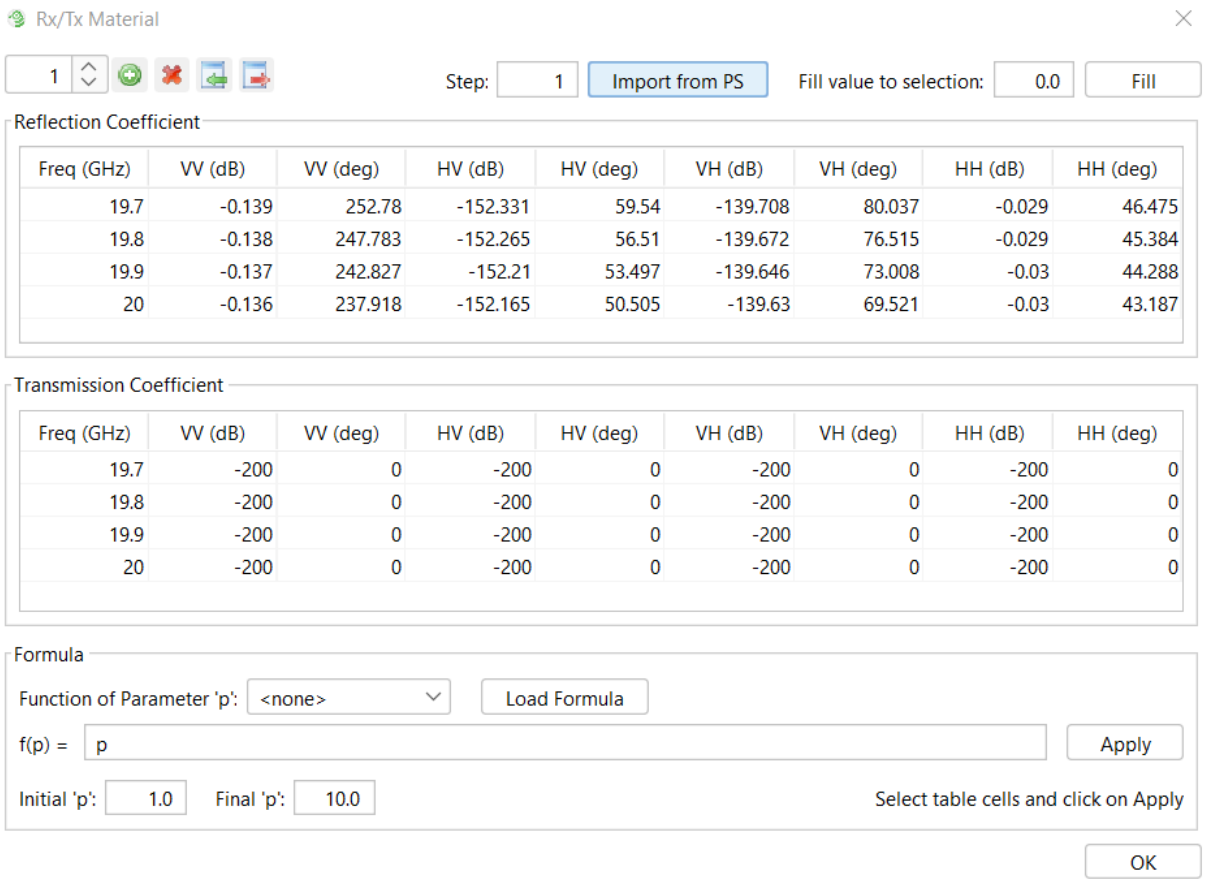


Figure 421: Material defined by reflection and transmission coefficients

To assign the reflection and transmission coefficients according to the frequency.

- **Frequency (GHz)**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched to use the most suitable one for the working frequency.

Import from PS: This option allows the user to import a matrix of reflections and transmission coefficients computed in the periodical structure module.

§ **Fill value to selection**: This option allows the user to automatically change the value of the selected cells of the table with the inserted value.

These parameters can also be set using formulas depending on the parameter *p*. Where *p* is defined from an initial value to a final value.

§ **Formula:**

This option serves for completing the selected cells of the table with values depending on the selected function. After choosing the function that we want to apply, pressing the Load Formula button will load the formula *f(p)* in the field. The next step is choosing the initial and the final value for the function. The final step is selecting the cells that we want to apply the function and pressing the button Apply. The cells will be auto-completed with the values created by the chosen function starting with the initial value and ending with the final one.

5.1.2 Edit

When this option is selected, a panel similar to the Add panel will be shown:

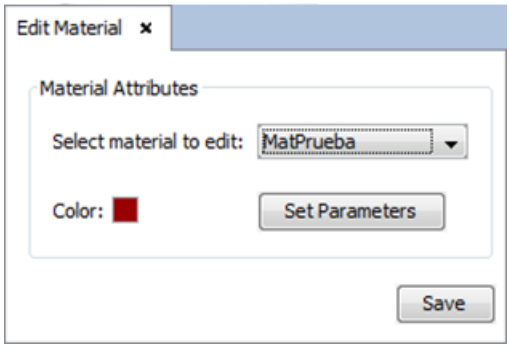


Figure 422: Edit Material panel

This panel allows the user to select an existing material to edit its properties. Refer to Materials Menu (MOM Module) for details on setting the color and physical parameters of materials.

5.1.3 Delete

Selecting this option shows the following panel on the right side of the window:

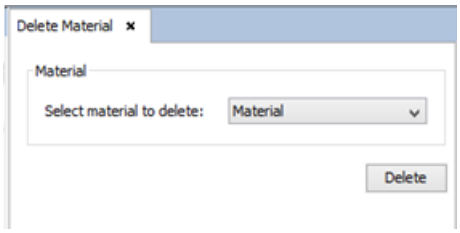


Figure 423: Delete Material panel

In this panel, the user may select the name of the material to remove and then left-click the Delete button.

5.1.4 Assign

This option allows the user to assign existing materials to objects and surfaces previously selected from the geometry tab. Selecting this option shows the following panel on the right side of the screen.

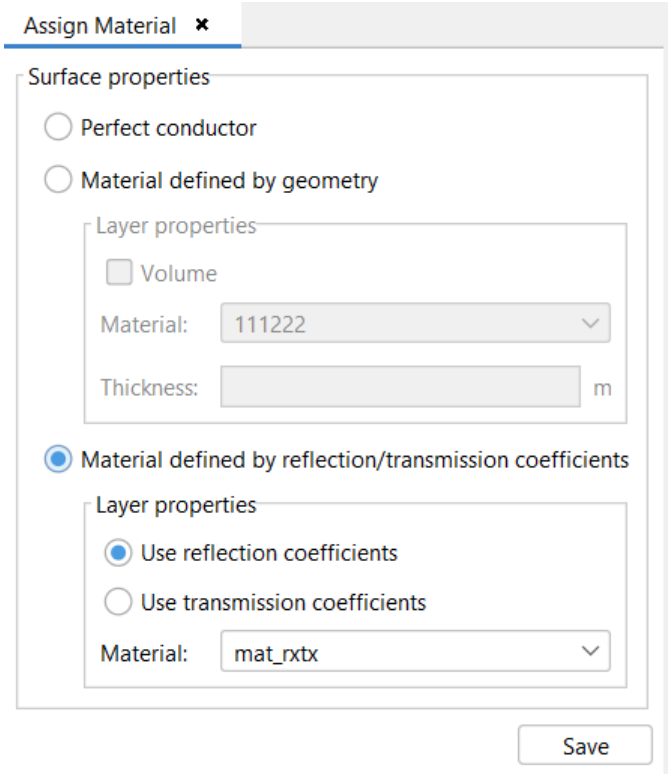


Figure 424: Assign Material panel

The following operations can be applied to surfaces:

Perfect Conductor
: Type of material that is selected by default. It corresponds to the metallic material.

Material defined by geometry
: This option allows the user to select a custom material and a thickness value for the selected objects and/or surfaces. If the 'Volume' option is selected, the objects (not surfaces) will be transformed to a volume and this property will be considered for the meshing and simulation.

Material defined by reflection/transmission coefficient
: This option allows the user to select a custom material for the selected objects and/or surfaces.

In order to assign a material to a surface or an object, first, select this one on the screen and then choose the material to be assigned. Finally, click on **Save** button.

5.1.5 Material Properties

This option displays the material assigned to different surfaces. The first table shows the selected surfaces, the color of the assigned material and the number of layers. The second table shows the selected surface of the first table with the materials of each layer.

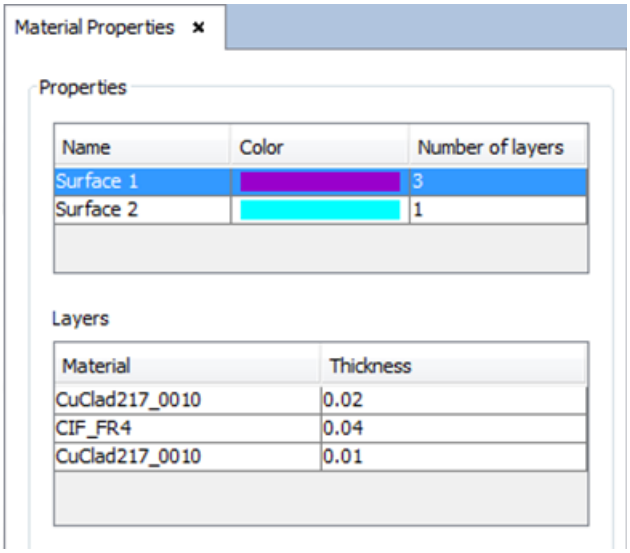


Figure 425: Material properties

PO module supports the multilayer material. Other modules only have one layer of a material.

5.2 Materials Menu (MONCROS Module)

In the following sections, all the options of the materials menu will be discussed:

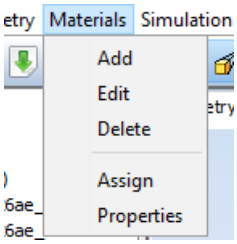


Figure 426: Materials Menu

5.2.1 Add

After selecting the Add option the following panel will be shown on the right side of the window:

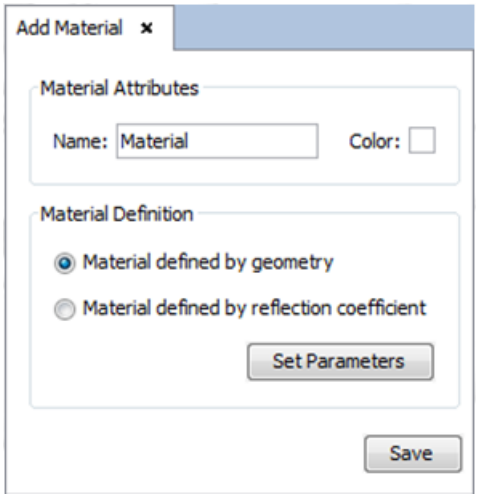


Figure 427: Add Material panel

This option allows the user to create a new material. The following options are available:

Material Attributes-Name: The user may want to provide a name for this material for future reference.

Material Attributes-Color: For more information, see APPENDIX A: Color Selection.

Material Definition - Material defined by geometry: Used to define a material according to its physical properties. After clicking on the Set Parameters button, two different options are available in the **Material defined by Geometry** window:

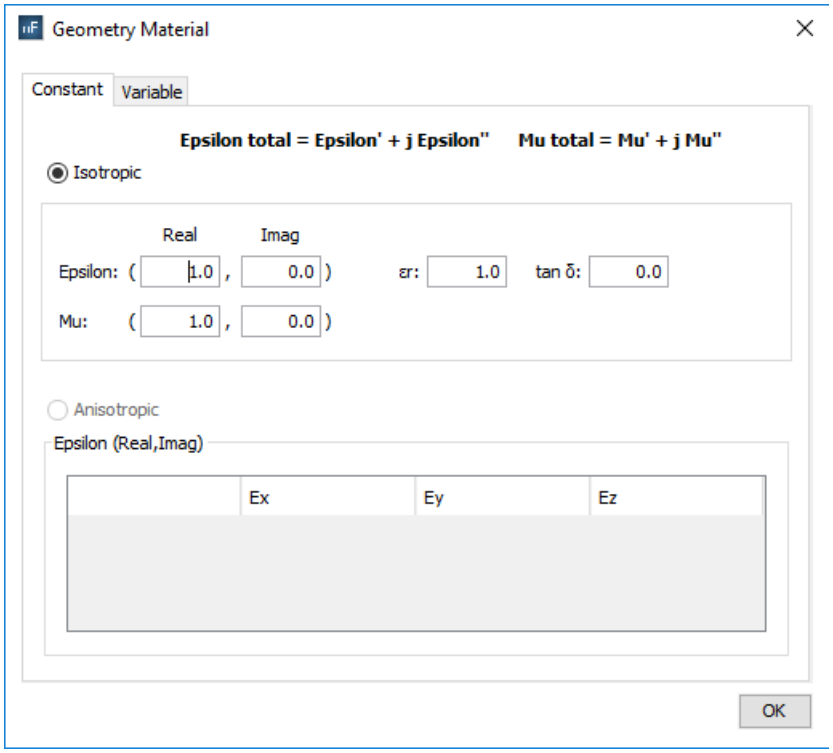


Figure 428: Material defined by geometry

Constant

§ **Real part**: common values are greater than or equal to 1.0.

§ **Imaginary part**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

o **Isotropic**: Selecting this option sets the epsilon (e' - permittivity) and mu (μ' - permeability) properties immutable. The user will need to specify both real and imaginary components of both properties or loss tangent and relative permittivity. Common real values for epsilon and mu are greater of equal to 1.0. If the imaginary part is negative, the material is considered to have losses; otherwise, it's considered to be a material with gain.

o **Anisotropic**: This option allows the user to set the material properties depending on the volumetric subdomain direction (i.e. the dielectric coefficient may change depending on the direction of the subdomains). The epsilon parameter consists of a 3x3 matrix where all Cartesian combinations can be modified. The materials using this option are only designed to be applied to volumes.

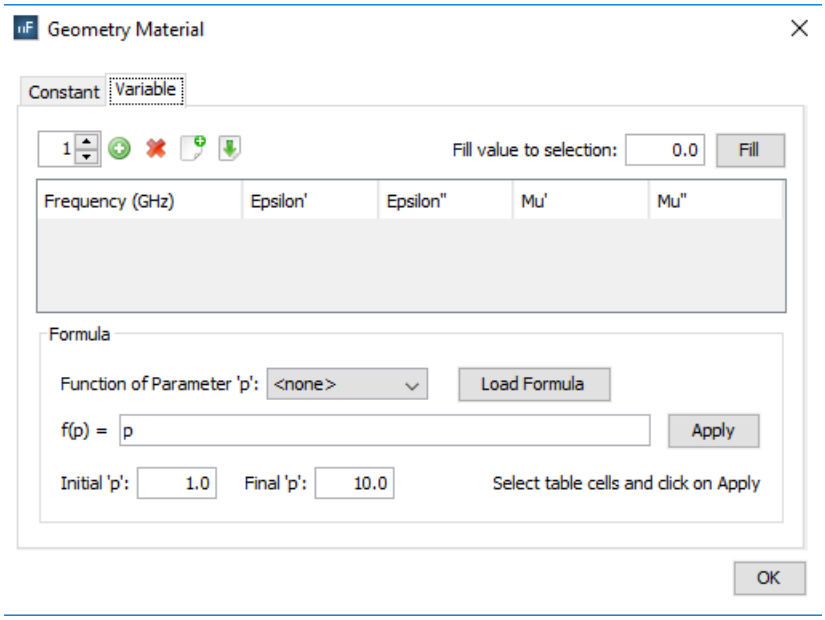


Figure 429: Material defined by geometry, tab 'Variable'

Variable

To assign the physical properties of the material according to the frequency.

§ **Frequency (in GHz)**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.

§ **Real part (')**: common values are greater than or equal to 1.0.

§ **Imaginary part (")**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

Figure 430: § Add:to add new frequencies to the material properties.

§ **Remove**: to remove existing frequencies to the material properties

§ **Import/Export**: to import table with values from a text file or export the table into a text file.

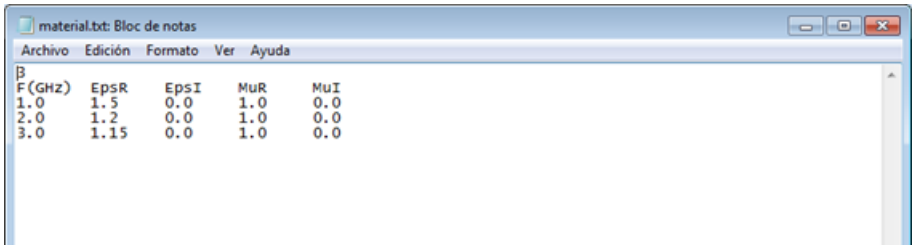
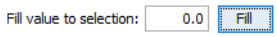


Figure 431: File Format

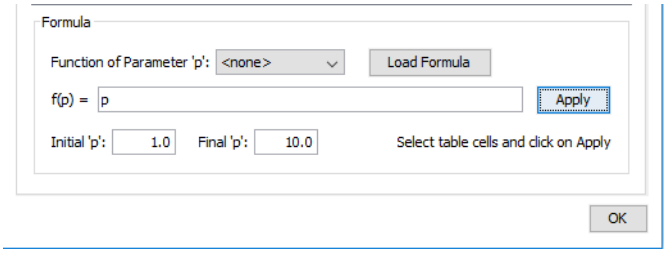
The information of this file can be modified by the user. Be sure not to modify the format and not to introduce any additional line break.



§ **Fill value to selection**: This option allows the user to automatically change the value of the selected cells of the table with the inserted value

These parameters can also be set using formulas in function of the frequency values.

§ **Formula:**



This option serves for completing the selected cells of the table with values depending on the selected function. After choosing the function that we want to apply, pressing the Load Formula button will load the formula *f(p)* in the field. The next step is choosing the initial and the final value for the function. The final step is selecting the cells that we want to apply the function to and press the button Apply. The cells will be auto-completed with the values created by the chosen function starting with the initial value and ending with the final one.

Material Definition - Material defined by reflection coefficient: Used to define a material according to its reflection coefficient. After clicking on the Set Parameters button, a window appears to define the reflection coefficient:

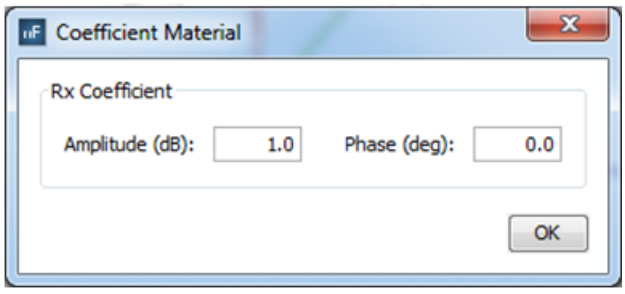


Figure 432: Material defined by reflection coefficient

5.2.2 Edit

When this option is selected, a panel similar to the Add panel will be shown:

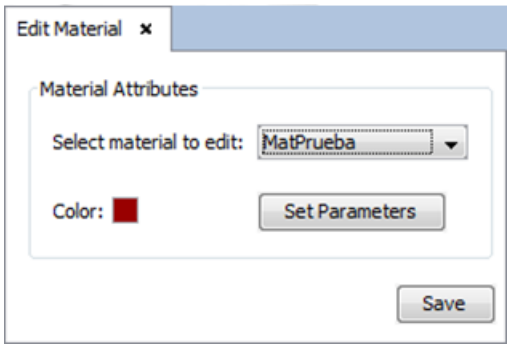


Figure 433: Edit Material panel

This panel allows the user to select an existing material to edit its properties. Refer to Materials Menu (MOM Module) for details on setting the color and physical parameters of materials.

5.2.3 Delete

Selecting this option shows the following panel on the right side of the window:

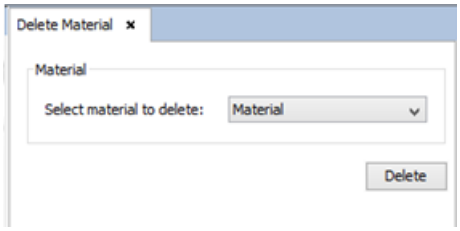


Figure 434: Delete Material panel

In this panel, the user may select the name of the material to remove and then left-click the Delete button.

5.2.4 Assign

This option allows the user to assign existing materials to objects and surfaces previously selected from the geometry tab. Selecting this option shows the following panel on the right side of the screen.

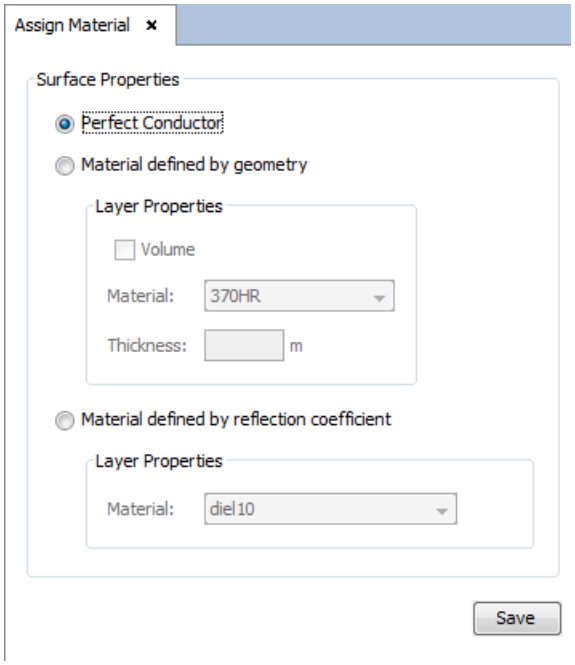


Figure 435: Assign Material panel

The following operations can be applied to surfaces:

Perfect Conductor
: Type of material that is selected by default. It corresponds to the metallic material.

Material defined by geometry
: This option allows the user to select a custom material and a thickness value for the selected objects and/or surfaces. If the 'Volume' option is selected, the objects (not surfaces) will be transformed to a volume and this property will be considered for the meshing and simulation.

Material defined by reflection coefficient
: This option allows the user to select a custom material for the selected objects and/or surfaces.

In order to assign a material to a surface or an object, first, select this one on the screen and then choose the material to be assigned. Finally, click on **Save** button.

5.3 Materials Menu (PO Module)

In this section, all the features available in the PO (Physical Optic) module Materials menu are explained. The available options in the Materials menu are shown in the next Figure.

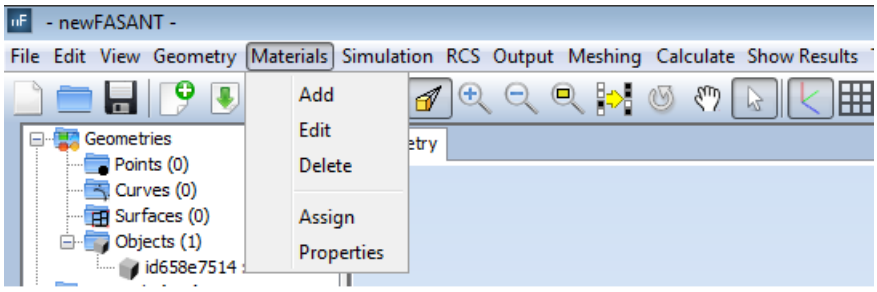


Figure 436: Materials Menu

In the following sections, all the options of the Materials Menu are detailed.

5.3.1 Add

When Add is selected, the Add Material window appears, as shown in the next Figure.

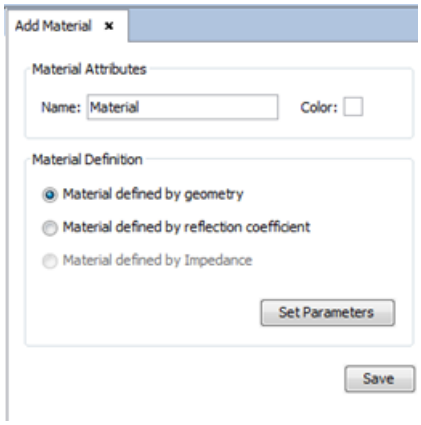


Figure 437: Add Material panel

Select the required parameters and click on Save. When clicking on this button, if the material exists, then a message is displayed informing the user that the material has already been defined. Otherwise, the material definitions are saved.

In the Add Material window, the following options are available:

Material Attributes - Name: The user specifies the desired name for the new material.

Material Attributes - Color: For more information, see APPENDIX A: Color Selection.

Material definition - Material defined by geometry: Used to define a material according to its physical properties. After clicking on the Set Parameters button, two different options are available in the Material defined by Geometry window:

Constant

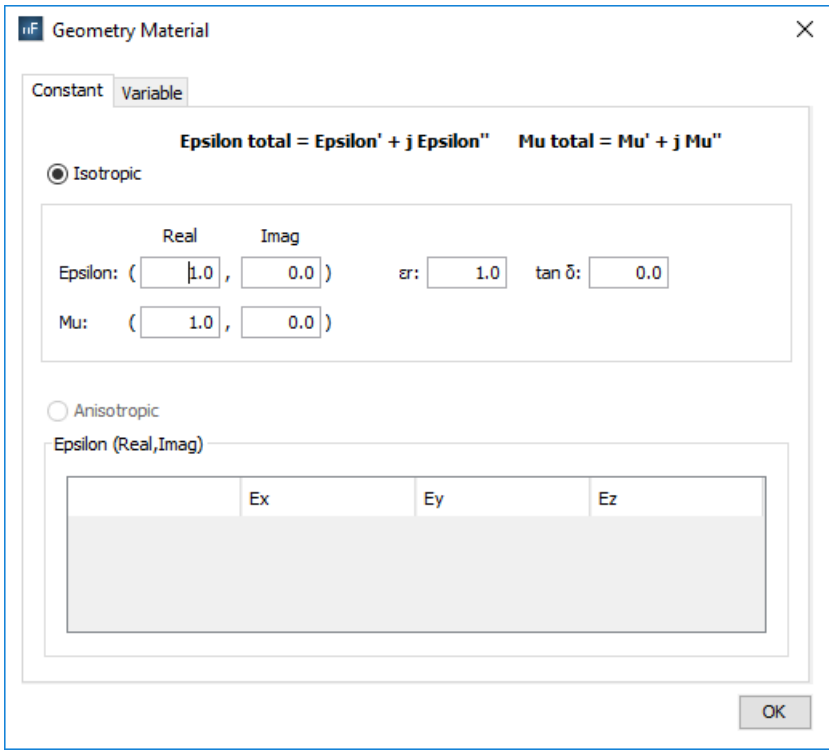


Figure 438: Material defined by geometry, tab 'Constant'

To set the material properties immutable. Epsilon (e') and Mu (μ') are defined with the same restrictions.

- **Real part**: common values are greater than or equal to 1.0.
- **Imaginary part**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.
    - Variable

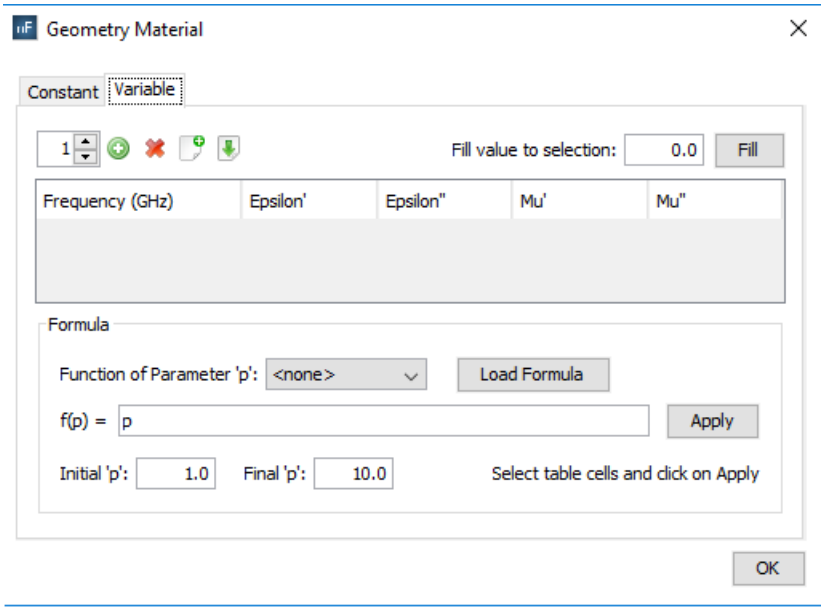


Figure 439: Material defined by geometry, tab 'Variable'

To assign the physical properties of the material according to the frequency.

- **Frequency (in GHz):**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.
- **Real part (')**: common values are greater than or equal to 1.0.
- **Imaginary part (")**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

- **Add**: to add new frequencies to the material properties.
- **Remove**: to remove existing frequencies to the material properties.
- **Import/Export**: to import table with values from a text file or export the table into a text file.

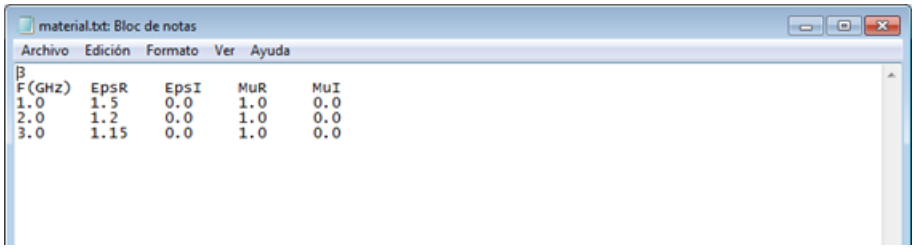


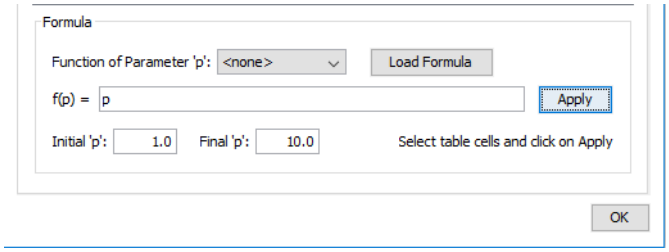
Figure 440: File Format

The information of this file can be modified by the user. Be sure not to modify the format and not to introduce any additional line break.

- **Fill value to selection**: This option allows the user to automatically change the value of the selected cells of the table with the inserted value

These parameters can also be set using formulas in function of the frequency values.

- **Formula**:



This option serves for completing the selected cells of the table with values depending on the selected function. After choosing the function that we want to apply, pressing the Load Formula button will load the formula f(p) in the field. The next step is choosing the initial and the final value for the function. The final step is selecting the cells that we want to apply the function to and press the button Apply. The cells will be auto-completed with the values created by the chosen function starting with the initial value and ending with the final one.

Material Definition - Material defined by reflection coefficient. Used to define a material according to its reflection coefficient. After clicking on the Set Parameters button, two different options are available in the Material defined by reflection window (the following Figure):

- **Constant**

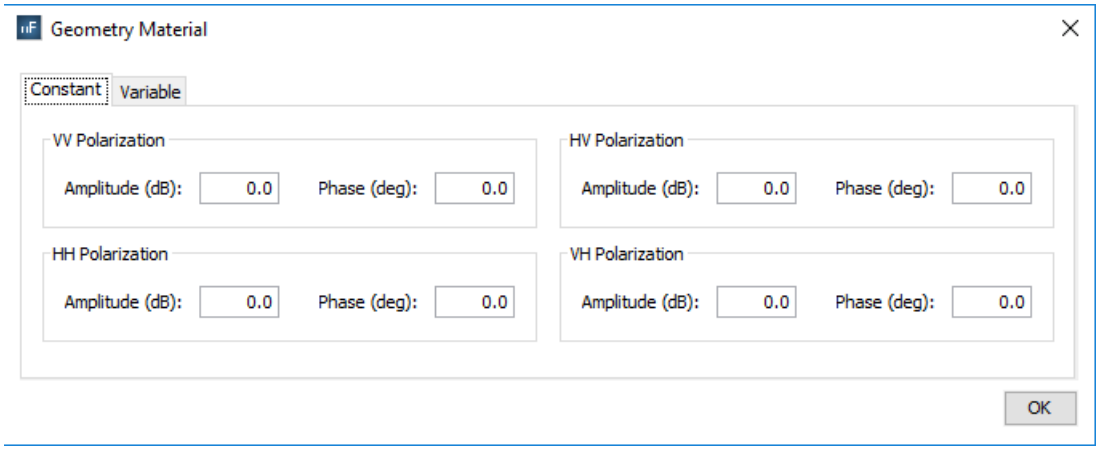


Figure 441: Material defined by coefficient, tab 'Constant'

To set the material properties invariable. Amplitude and Phase are defined using decibels (dB) and degrees (degrees).

- **Variable with frequency and angle**

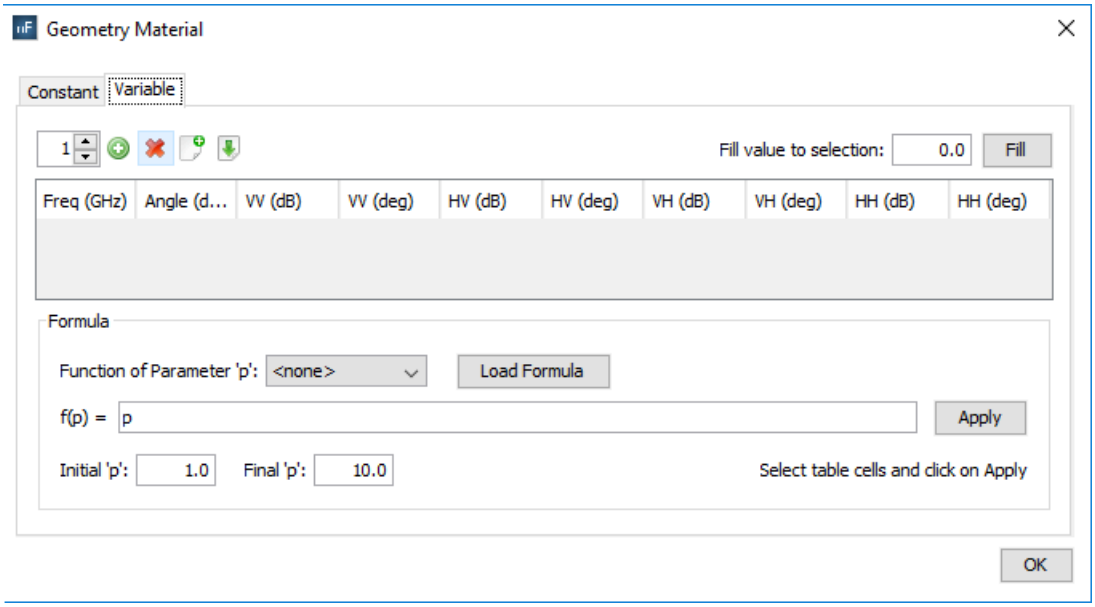


Figure 442: Material defined by coefficient, tab 'Variable'

To assign the refraction properties of the material according to the frequency and angle.

- **Frequency (in GHz):**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.
- **Angle (in degrees):**: Sets the associated angle to the physical properties.

5.3.2 Edit

This option allows the user to modify a previously defined Material. When the Materials –Edit option is selected the Edit Material window (following Figure) appears. When the material to be edited has been selected in the Select material combo box, its color can be modified. It is important to remark that the properties of the materials may be modified, but not their type (defined by geometry, defined by electrical impedance, or defined by reflection coefficient). To edit the material parameters, click on the Set Parameters button and the corresponding window (following Figure) will appear. The rest of the editing process may continue as explained in section Add.

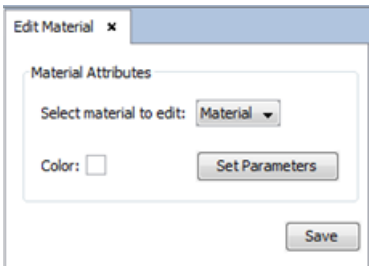


Figure 443: Edit Material panel

5.3.3 Delete

This option allows the user to remove a previously defined Material. When the Materials –Delete option is selected, the Delete Material window (following Figure) appears. When the material to be deleted has been selected in the Select material combo box, the action can be confirmed by clicking on Delete.

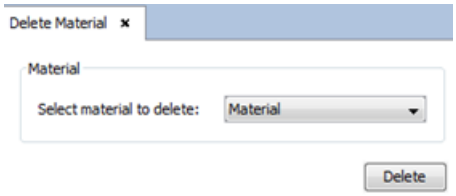


Figure 444: Delete Material panel

5.3.4 Assign

The geometry properties assigned to materials can be edited by clicking on Materials Assign. After selecting this option, the surfaces/objects assigned to specific materials must be selected, and confirm the selection with the "Assign" button as shown in next Figure.

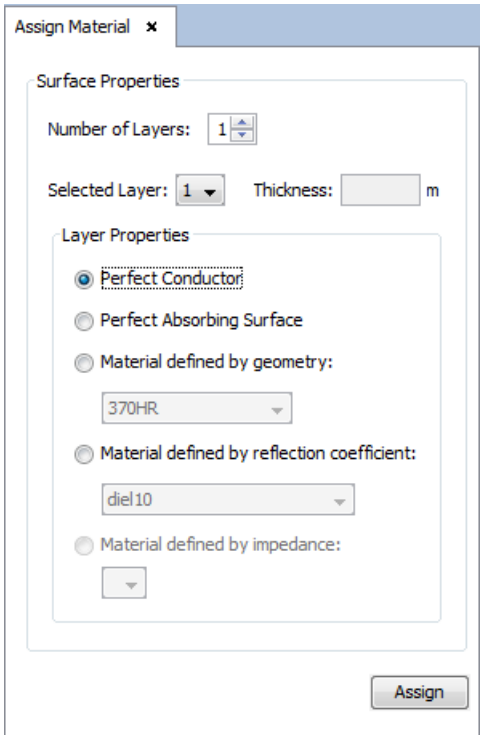


Figure 445: Assign Material panel

The following material assignment options can be applied to surfaces:

Number of Layers
to specify the number of layers to be considered in this object.

Selected Layer
: to select the layer that is being edited.

Thickness
: to assign a thickness to each layer.

For each layer, the user can select one of the following material types but can’t assign material properties defined by geometry, material properties defined by reflection coefficient, and material properties defined by electrical impedance to the same surface.

Perfect Conductor
: The objects of this type of material are metallic or PEC (Perfect Electrical Conductor). This is the material selected by default.

Perfect Absorbing Surface
: The objects of this type of material are perfectly absorbing.

Material defined by geometry
: If there are materials defined with this type, it is possible to assign them by selecting from the combo box.

Material defined by reflection coefficient
: If there are materials defined with this type, it is possible to assign them by selecting from the combo box.

Material defined by electrical impedance
: If there are materials defined with this type, it is possible to assign them by selecting from the combo box. (Note This feature is under development at this moment.)

Assign button
: To confirm the changes.

In order to assign a material to a surface or an object, first, select this one on the screen and then choose the material to be assigned. Finally, click on **Save** button.

5.3.5 Material Properties

This option displays the material assigned to different surfaces. The first table shows the selected surfaces, the color of the assigned material and the number of layers. The second table shows the selected surface of the first table with the materials of each layer.

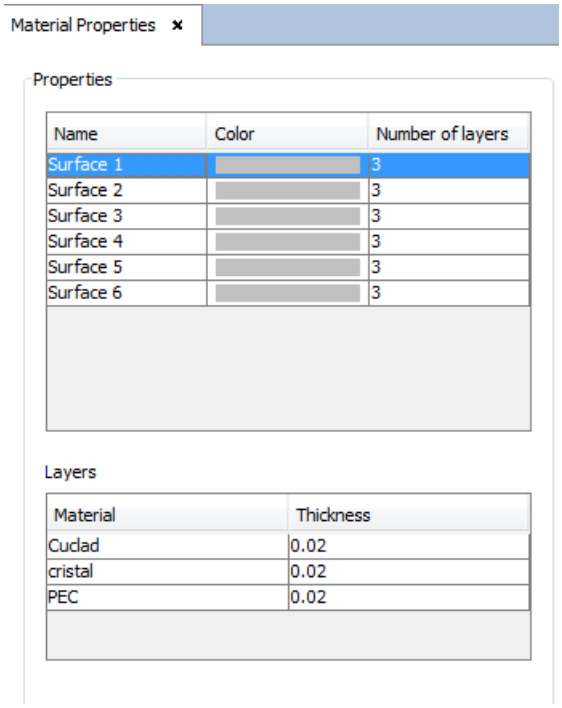


Figure 446: Material properties

PO module supports the multilayer material. Other modules only have one layer of a material.

5.4 Materials Menu (GTD and GTD-PO Modules)

In this section all the features available in the GTD (Geometrical Theory of Diffraction) module Materials menu are explained. The available options in the Materials menu are shown in the next Figure.

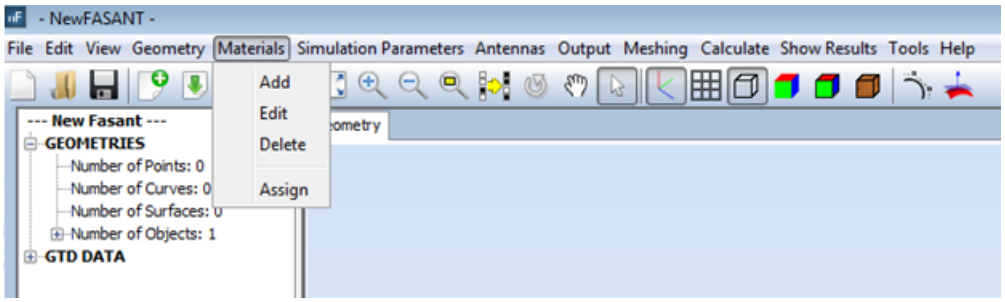


Figure 447: Materials Menu

In the following sections all the options of the Materials Menu are detailed.

5.4.1 Add

When Add is selected, the Add Material window appears, as shown in the next Figure.

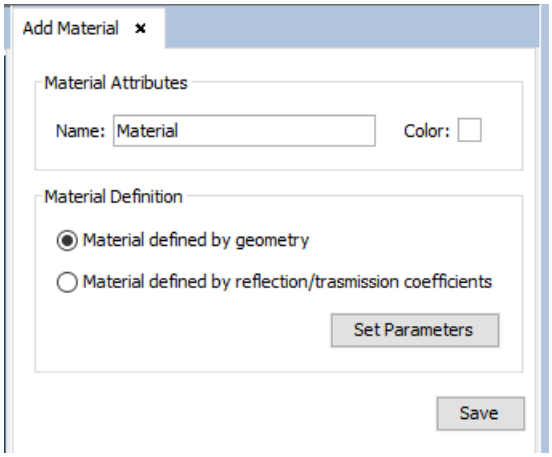


Figure 448: Add Material panel

Select the required parameters and click on Save. When clicking on "save" to save the material, if it exists, then a message is displayed informing the user that the material has already been defined. If not, then the material definitions are saved.

In the Add Material window, the following options are available:

Material Attributes - Name. The user specifies the desired name for the new material.

Material Attributes - Color. For more information, see APPENDIX A: Color Selection.

Material Definition- Material defined by geometry. Used to define a material according to its physical properties. After clicking on the Set Parameters button, two different options are available in the Material defined by Geometry window:

- **Constant**

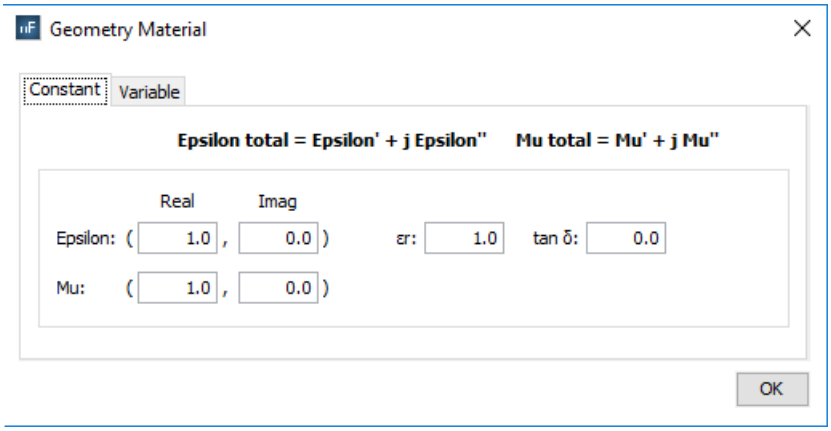


Figure 449: Material defined by geometry, tab 'Constant'

To set the material properties invariable. Epsilon (e') and Mu (μ') are defined with the same restrictions.

§ **Real part**: common values are greater than or equal to 1.0.

§ **Imaginary part**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

Variable

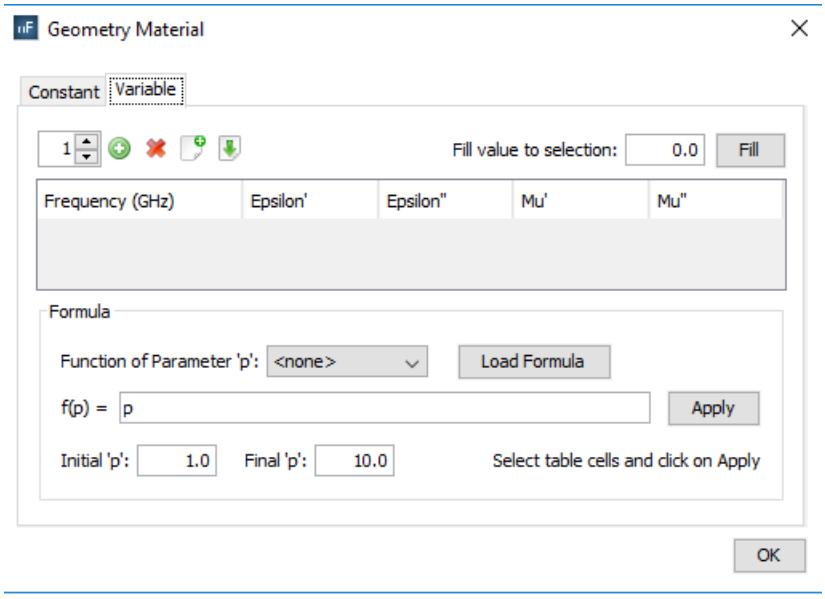


Figure 450: Material defined by geometry, tab 'Variable'

To assign the physical properties of the material according to the frequency.

§ **Frequency (in GHz)**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.

§ **Real part (')**: common values are greater than or equal to 1.0.

§ **Imaginary part (")**: if the imaginary part is negative, the material is considered to have losses; otherwise it would be equivalent to a material with gain.

§ **Add**: to add new frequencies to the material properties.

§ **Remove**: to remove existing frequencies to the material properties

§ **Import/Export**: to import table with values from a text file or export the table into a text file.

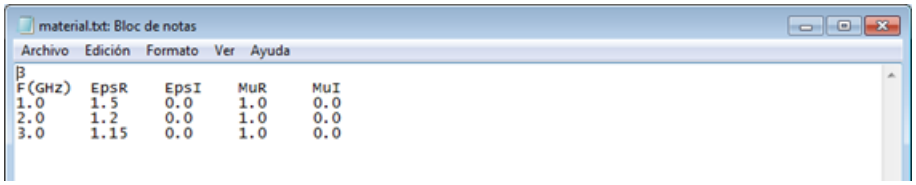
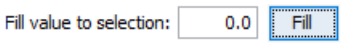


Figure 451: File format

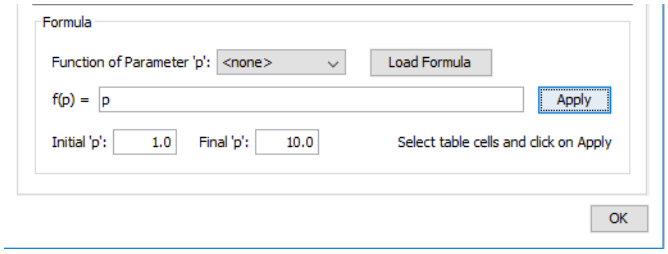
The information of this file can be modified by the user. Be sure not to modify the format and not to introduce any additional line break.



§ **Fill value to selection**: This option allows the user to automatically change the value of the selected cells of the table with the inserted value

These parameters can also be set using formulas in function of the frequency values.

§ **Formula**:



This option serves for completing the selected cells of the table with values depending on the selected function. After choosing the function that we want to apply, pressing the Load Formula button will load the formula f(p) in the field. The next step is choosing the initial and the final value for the function. The final step is selecting the cells that we want to apply the function to and press the button Apply. The cells will be auto-completed with the values created by the chosen function starting with the initial value and ending with the final one.

Material Definition - Material defined by reflection/transmission coefficient. Used to define a material according reflection coefficient. After clicking on the Set Parameters button, three different options are available in the Material defined by reflection window (following Figure):

- **Constant**

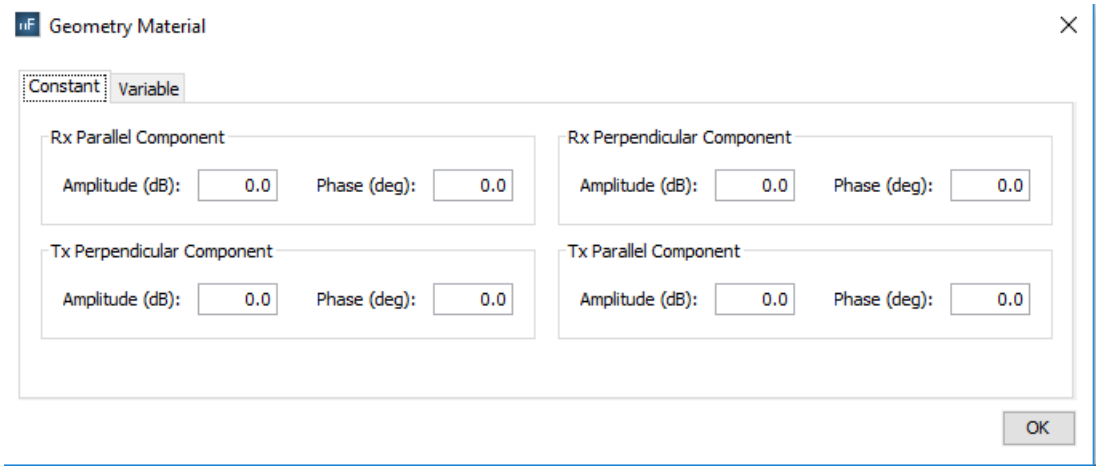


Figure 452: Material defined by coefficient, tab 'Constant'

To set the material properties invariable. Amplitude and Phase are defined with decibels (dB) and degrees (degrees).

- **Variable with frequency and angle**

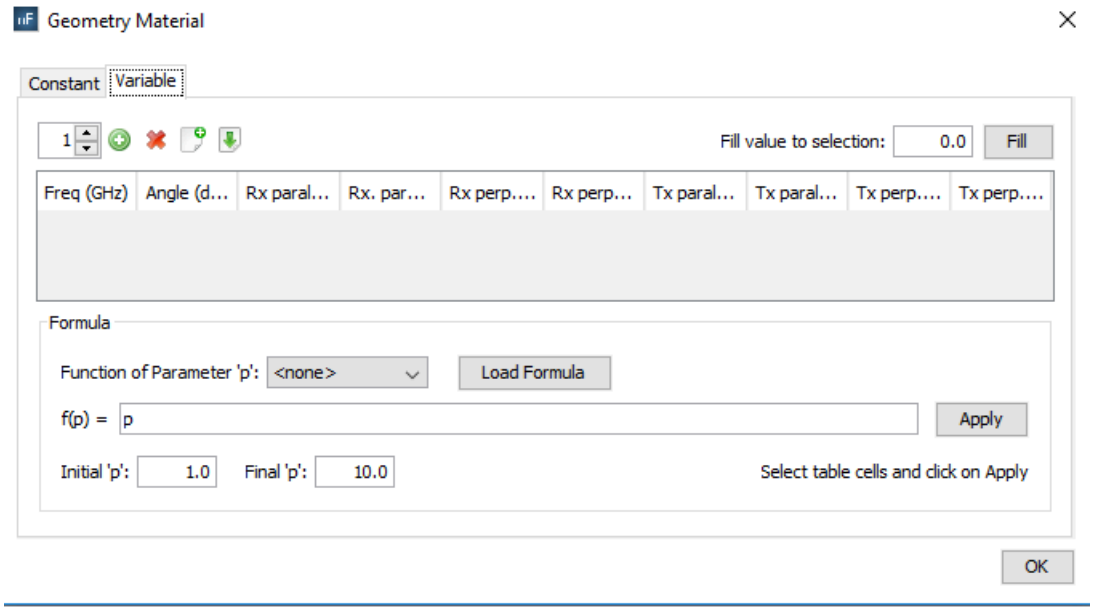


Figure 453: Variable with frequency and angle

To assign the refraction and transmission properties of the material according to the frequency and angle.

§ **Frequency (in GHz)**: Sets the associated frequency to the physical properties. The frequency values for each material are interpolated when the simulation is launched in order to use the most suitable one for the working frequency.

§ **Angle (in degrees)**: Sets the associated angle to the physical properties.

5.4.2 Edit

This option allows the user to modify a previously defined Material. When the Materials –Edit option is selected the Edit Material window (following Figure) appears. When the material to be edited has been selected from the Select material combo box, its color can be modified. It is important to remark that the properties of the materials may be modified, but not their type (defined by geometry, defined by electrical impedance, or defined by reflection coefficient). To edit the material parameters, click on the Set Parameters button and the corresponding window (following Figure) will appear. The rest of the editing process may continue as explained in section Materials Menu (GTD and GTD-PO Modules).

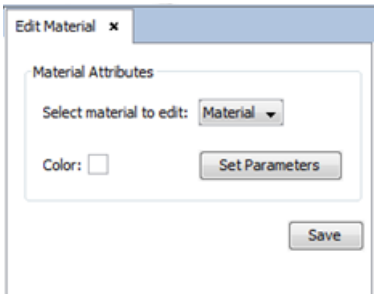


Figure 454: Edit Material panel

5.4.3 Delete

This option allows the user to remove a previously defined Material. When the Materials –Delete option is selected, the Delete Material window (following Figure) appears. When the material to be deleted has been selected in the Select material combo box, the action can be confirmed by clicking on Delete.

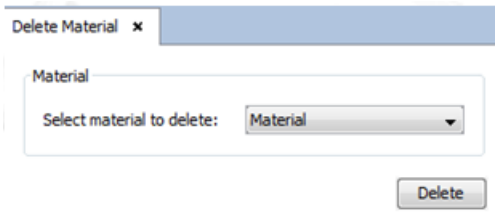


Figure 455: Delete Material panel

5.4.4 Assign

The geometric properties assigned to materials can be edited by clicking on Materials - Assign. After selecting this option, the surfaces/objects assigned to specific materials must be selected, and confirm the selection with the "Assign" button as shown in next Figure.

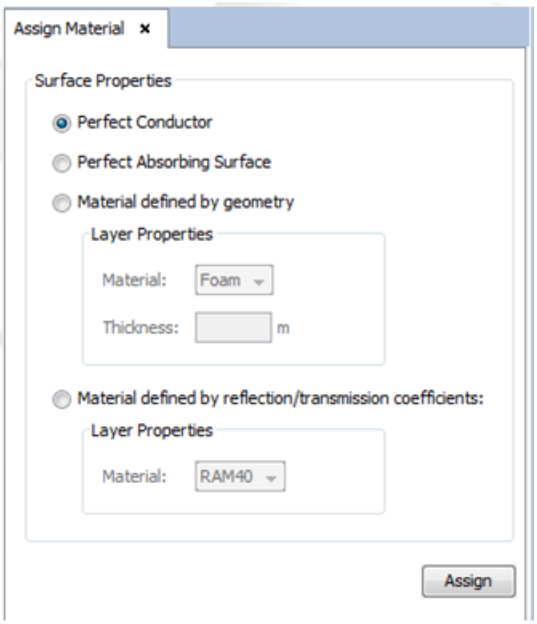


Figure 456: Assign Material panel

The following material assignment options can be applied to surfaces:

Perfect Conductor
: The objects of this type of material are metallic, or PEC (Perfect Electrical Conductor). This is the material selected by default.

Perfect Absorbing Surface
: The objects of this type of material are perfectly absorbing.

Material defined by geometry
: If there are materials defined with this type, it is possible to assign them by selecting from the combo box and to specify the thickness.

Material defined by reflection/transmission coefficient
: If there are materials defined with this type, it is possible to assign them by selecting from the combo box.

Assign button
: To confirm the changes.

In order to assign a material to a surface or an object, first, select this one on the screen and then choose the material to be assigned. Finally, click on **Save** button.

5.4.5 Material Properties

This option displays the material assigned to different surfaces. The first table shows the selected surfaces, the color of the assigned material and the number of layers. The second table shows the selected surface of the first table with the materials of each layer. This options is only available in GTD module.

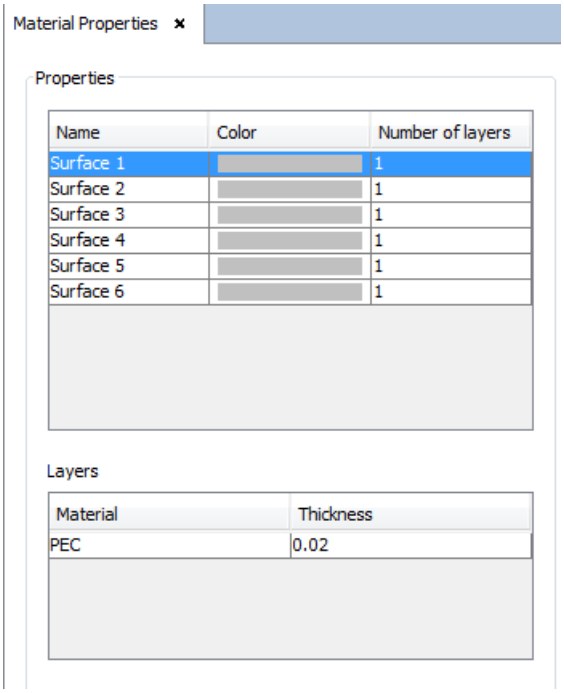


Figure 457: Material properties

PO module supports the multilayer material. Other modules only have one layer of a material.

5.5 Materials Menu (Periodical Structures Module)

In this section, the different options of the "Materials" menu from the Periodical Structures module will be detailed. The following figure shows the "Materials" menu of this module:

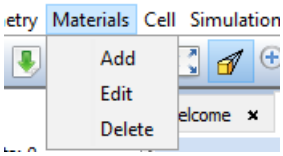


Figure 458: "Materials" menu of the Periodical Structures module

5.5.1 Add

The "Add" option of the materials menu allows the user to add a new material to the list of materials. When this option is selected, the following tab will appear:

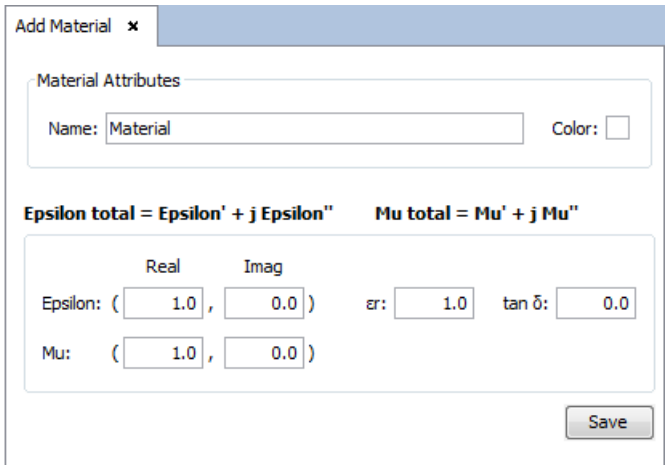


Figure 459: "Add Material" tab

In this tab, the user can set the parameters for the material they want to create. The "**Material Attributes - Name**" option allows the user to give a name for the created material for future reference, and the "**Material Attributes - Color**" allows to give the material a representative color by clicking the small rectangle with the current material color (white, by default). On more details about color selection for a material, see APPENDIX A: Color Selection.

The user should modify the Epsilon (permittivity) and Mu (permeability) values associated with the material, specifying their real parts (on the left text field) and imaginary parts (on the right text field), or the loss tangent and relative permittivity values. Common real parts for these values are greater or equal to "1.0". Additionally, materials with positive imaginary parts are considered to have gains, while materials with negative imaginary parts are considered to have losses.

When the user is satisfied with the properties given to the material, they should click the "Save" button to confirm the creation of the material.

5.5.2 Edit

The "Edit" option of the materials menu allows the user to edit the properties of already existing materials. When selected, this option will show the following tab to the user:

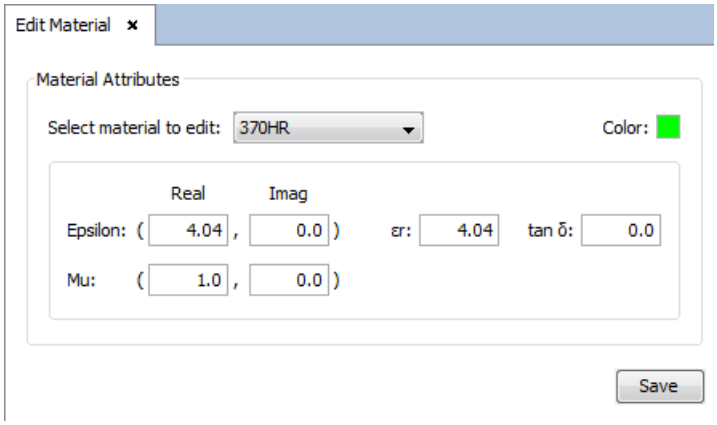


Figure 460: "Edit Material" tab

The user needs to select the material they want to edit in the "Select material to edit" dropdown menu. The user can modify the material color by clicking the small rectangle filled with the material color, as well as modify the Epsilon and Mu values. The meaning of these values is the same as in the "Add Material" tab, detailed in the previous section.

When the user has finished editing the parameters for the material, they need to click the "Save" button in order to confirm the changes.

5.5.3 Delete

The "Delete" option of the "Materials" menu allows the user to delete already existing materials. When the user clicks on this option, the following tab will be presented:

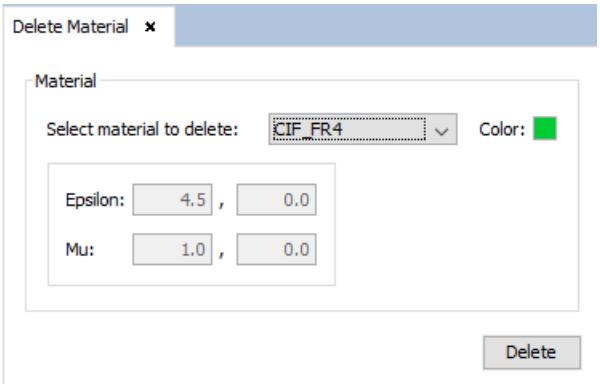


Figure 461: "Delete Material" tab

The user needs to select the material to delete in the "Select material to delete" dropdown menu. The properties of the selected material, that is, the material color and the Epsilon/Mu values, will be shown

in this tab. If the user wants to delete the selected material, they only need to click the "Delete" button. Note the material deletion cannot be undone.

5.6 Materials Menu (US Module)

This section describes the options of the Materials menu available in US projects.

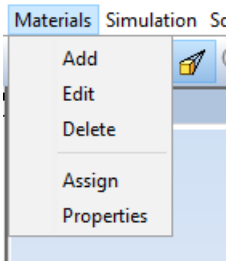


Figure 462: Materials menu (US module)

5.6.1 Add

The "Add" option allows the user to create new materials to be assigned to objects and surfaces in the simulation. When the user selects this option, the following panel appears:

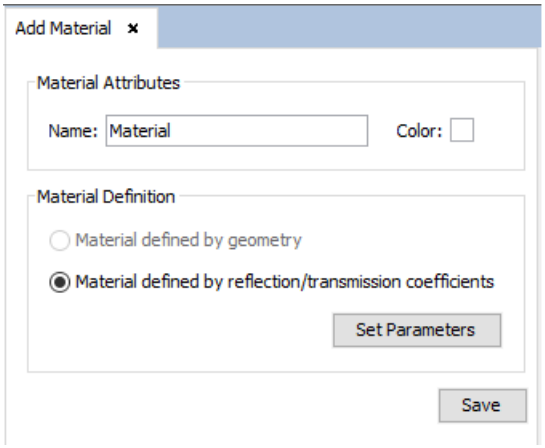


Figure 463: Add Material panel

Currently, the only possibility is the creation of materials given by their reflection/transmission coefficients. In this panel, the user can set the name of the new material, as well as the color. To set the color, the user needs to double-click the box next to the "Color:" label. For more information about setting a material's color, check APPENDIX A: Color Selection.

In order to set the parameters of the material, the user needs to press the "Set Parameters" button. The following dialog will appear:

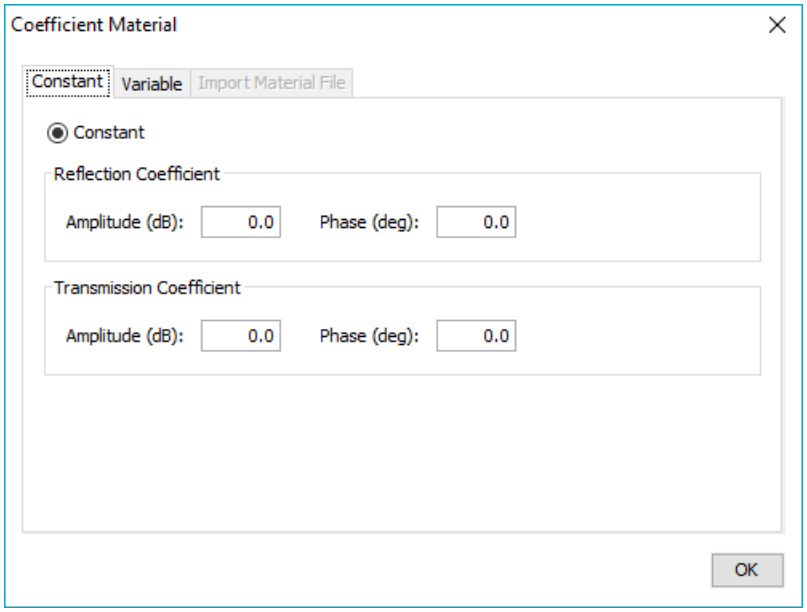


Figure 464: Set Parameters dialog

There are two types of coefficient-defined materials:

- **Constant coefficient materials** These are materials with reflection and transmission coefficients that are constant (not depending on frequency nor angle of incidence). To define this kind of material, it is necessary to check the "Constant" option under the "Constant" tab and introduce the complex values of the coefficients. Please note that the amplitudes of the coefficients cannot be larger than 1.

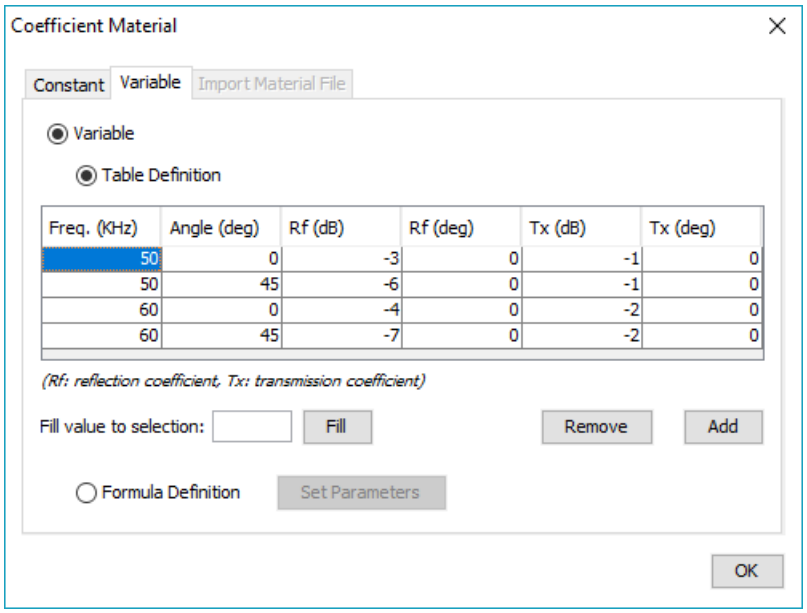


Figure 465: Set Parameters dialog (variable coefficients)

- **Variable coefficient materials** These are materials with reflection/transmission coefficients that depend on the simulation frequency and the incidence angle of the ray in the surface. To define this

kind of material, the user needs to select the "Variable" option under the "Variable" tab of the "Set parameters" dialog. There are two different ways of creating variable coefficient materials:

- **Table definition** This mode of creating variable coefficient materials allows the user to manually specify the reflection and transmission coefficient for different frequencies and angles of incidence. To add and remove rows, the user needs to use the "Add" and "Remove" buttons located below the table. The user can edit the content of the cells of the table directly. However, it is possible to bulk update the value of multiple cells at once. To do that, the user needs to select the cells of the table, introduce the value in the "Fill value to selection" field and then press the "Fill" button.
- **Formula definition** This mode allows the user to specify formulas used to generate the values of the amplitudes and phases of the coefficients. In the formula definition mode, the user needs to set a sweep of frequency values and a sweep of angle of incidence values, used to generate the data points where the formulas will be evaluated. The user also needs to specify the formulas for each of the amplitudes and phases of the coefficients. The available parameters and functions are shown in the drop-down lists at the top of the "Formula Parameters" panel. The "Apply" buttons next to the formula boxes allows the user to add the function to the formula.

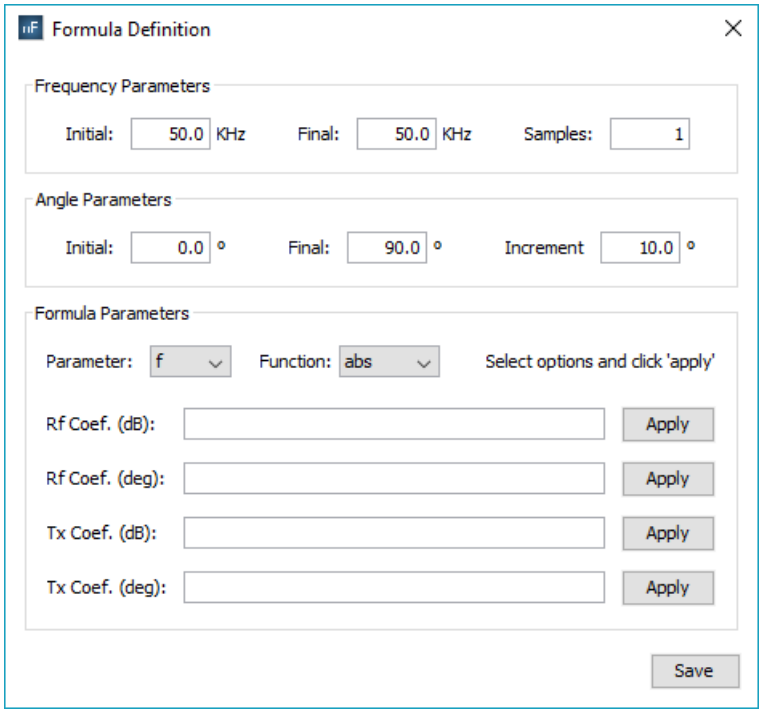


Figure 466: Set Parameters dialog (formula definition)

Once the parameters has been set, the user needs to press the "OK" button on the "Coefficient Material" dialog and then press the "Save" button in the "Add Material" button in order to save the new material.

5.6.2 Edit

This option allows the edition of already created materials. When the user selects this option, the following panel appears:

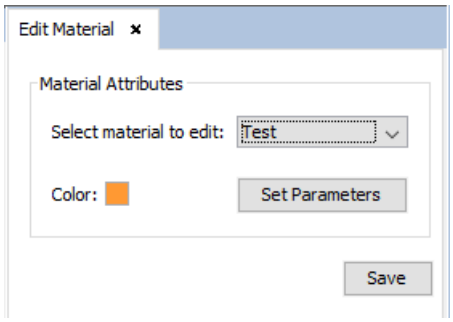


Figure 467: Edit Material panel

The user can select the material to edit in the drop-down list next to the "Select material to edit:" label. It is possible to edit the color of the selected material by double-clicking the box next to the "Color" label. The "Set Parameters" button allows the user to modify the reflection/transmission coefficients of the materials. For more details on the edition of the parameters of a material, refer to the "Add" section.

5.6.3 Delete

This option allows the user to delete a material. When the user selects this option, the following panel appears:

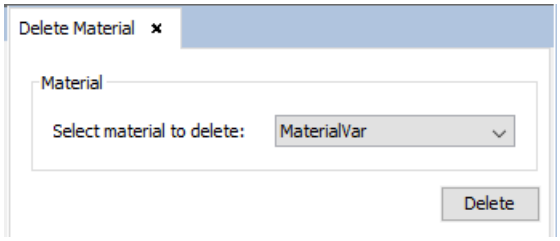


Figure 468: Delete Material panel

To delete a material, the user needs to select the material's name from the drop-down list and then press the "Delete" button. Please note that, if the material to delete is already assigned to at least one object or surface, the following warning will appear:

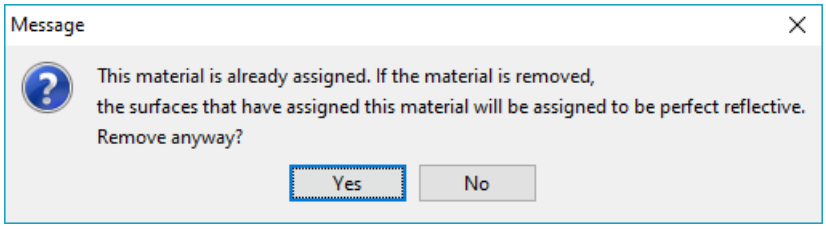


Figure 469: Warning when deleting an assigned material

If the user selects "Yes", the material will be deleted and the surfaces that had the material assigned will be assigned a perfect reflective material instead. If the user selects "No", the material won't be deleted and no action will be performed.

5.6.4 Assign

This option allows the user to assign a material to one or more surfaces. When this option is selected, the following panel appears:

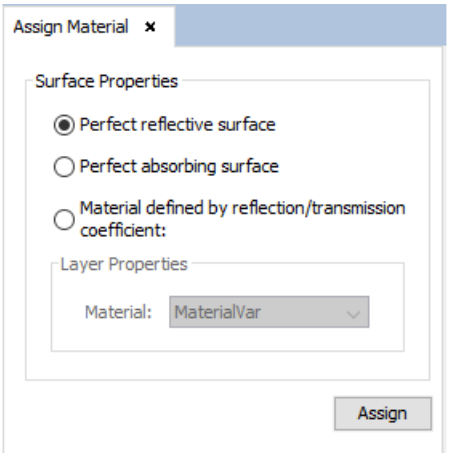


Figure 470: Assign Material panel

There are several options when assigning materials:

- **Perfect reflective surface**: Represents a material that perfectly reflects any wave that gets into it (that is, without attenuation).
- **Perfect absorbing surface**: Represents a material that perfectly transmits any wave that gets into it (that is, without attenuation).
- **Material defined by reflection/transmission**: This option allows the user to select a user-defined material to be assigned to a surface or object.

To assign a material, the user needs to select (in the geometry panel) the objects and surfaces they want to assign the material to, and then press the "Assign" button.

5.6.5 Properties

This option allows the user to visualize the materials assigned to surfaces, as well as bulk assign materials and visualize the coefficients of the assigned materials. The user needs to select the objects and surfaces in the geometry panel and then select the "Properties" option. The following panel will appear:

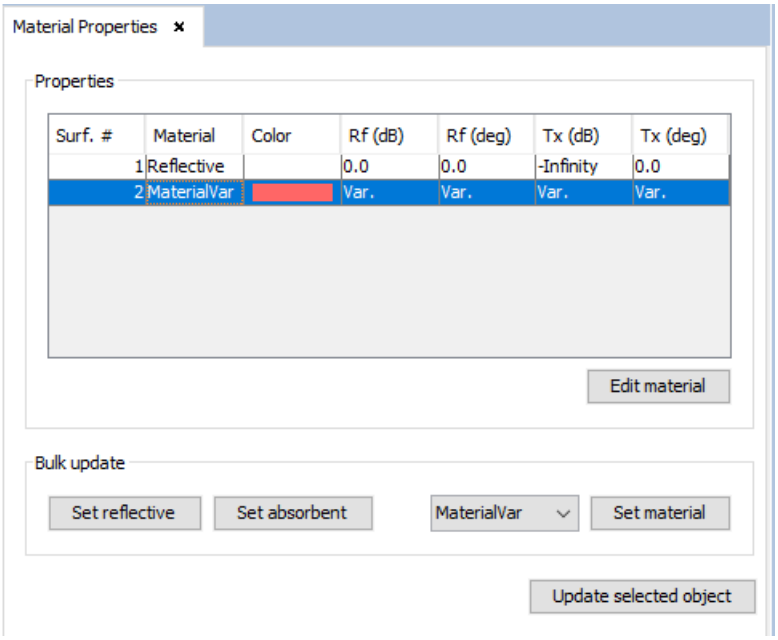


Figure 471: Material Properties panel

It is possible to edit the material of the selected surface in the table (only if it isn't a perfectly reflective or absorbent material) by pressing the "Edit material" button. For more details on how to edit the parameters of a material, refer to the "Add" section.

The "Set reflective", "Set absorbent" and "Set material" buttons allow the user to bulk-assign a material to multiple surfaces (to select multiple surfaces in the table, hold the Ctrl key while clicking surfaces). The "Set material" allows to assign a user-defined material to one or multiple surfaces. The material that is assigned corresponds to the one selected in the drop-down list next to the button.

If the user wants to update which surfaces appear in the table, they need to select the appropriate objects and surfaces in the geometry panel and then press the "Update selected object" button.

5.7 APPENDIX A: Color Selection

The box shows the currently selected color for the material. Left-clicking the box will originate the Choose Material Color dialog, which allows the user to select a color in a variety of color modes:

Swatches
: This tab shows a grid of predefined colors from which the user can choose.

HSV
: In this tab the user can pick a color specifying its hue, saturation, and value (HSV) properties. The user can also specify a transparency or alpha value for the color.

HSL
: Likewise, in this tab the user can pick a color specifying its hue, saturation, and lightness (HSL) values and also a transparency value.

RGB
: This tab allows the user to pick a color specifying its RGBA (red, green, blue, and alpha) components.

CMYK
: This tab allows the user to pick a color specifying its CMYK (cyan, magenta, yellow, and black) components, and also an alpha component.

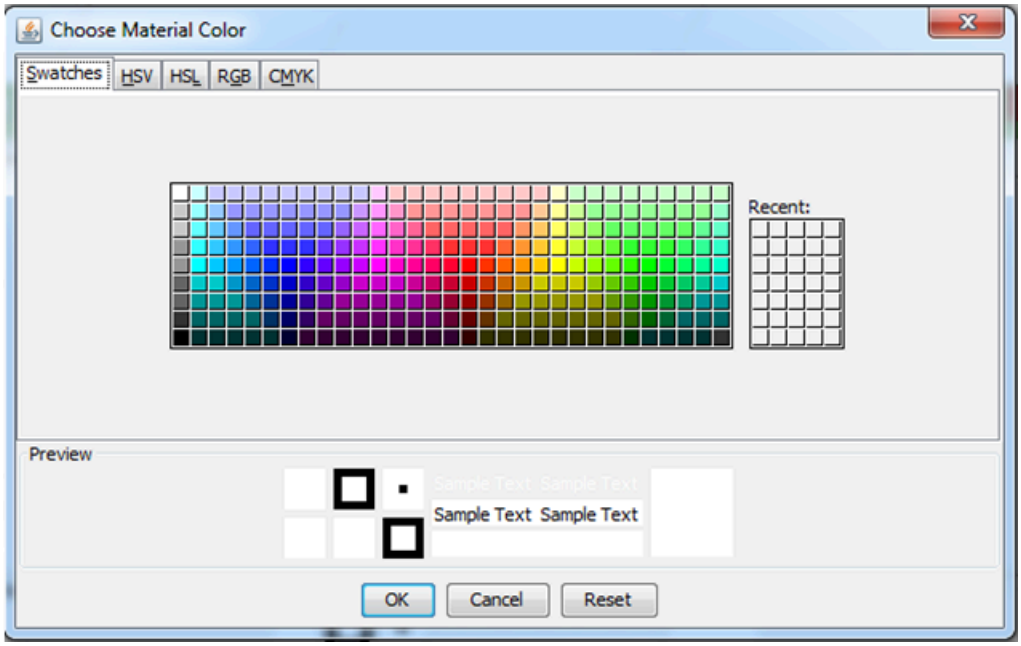


Figure 472: Choose Material Color using the 'Swatches' tab

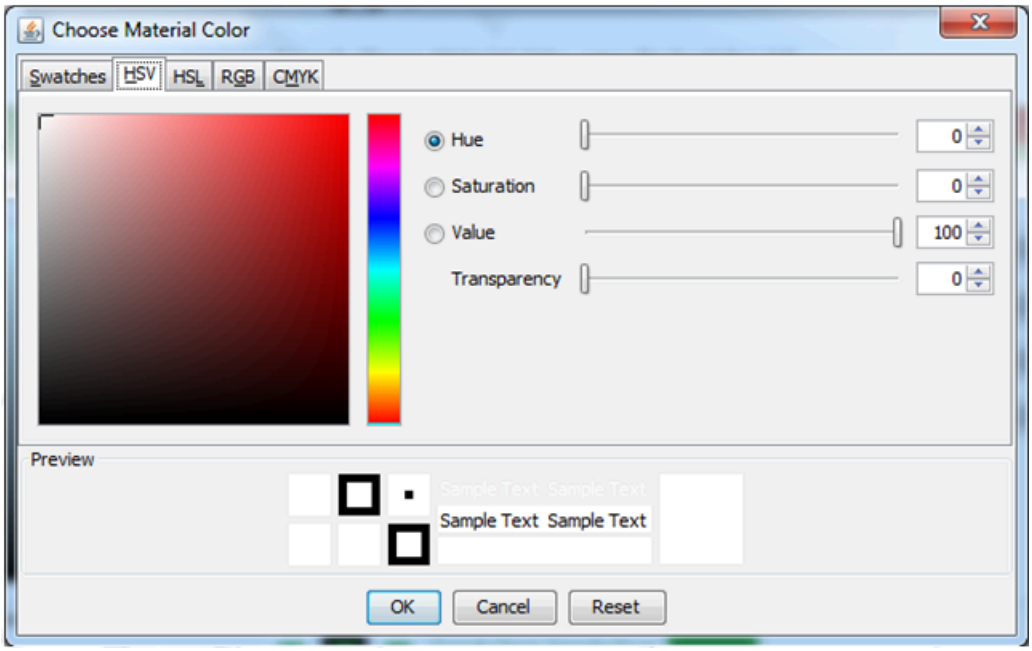


Figure 473: Choose Material Color using the 'HSV' tab

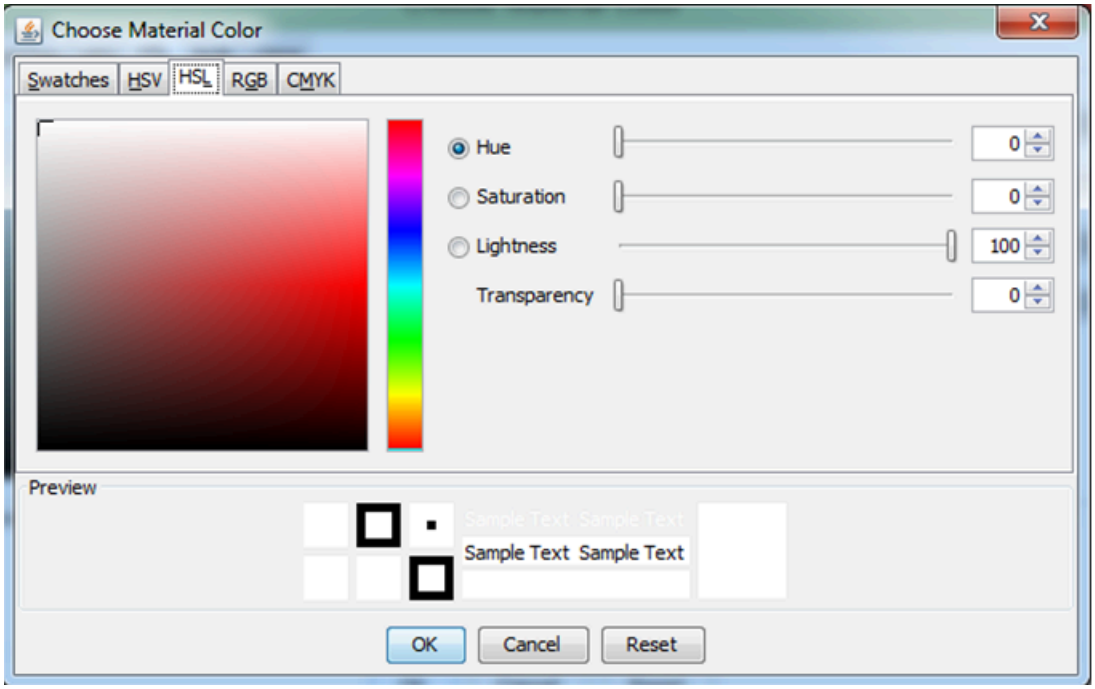


Figure 474: Choose Material Color using the 'HSL' tab

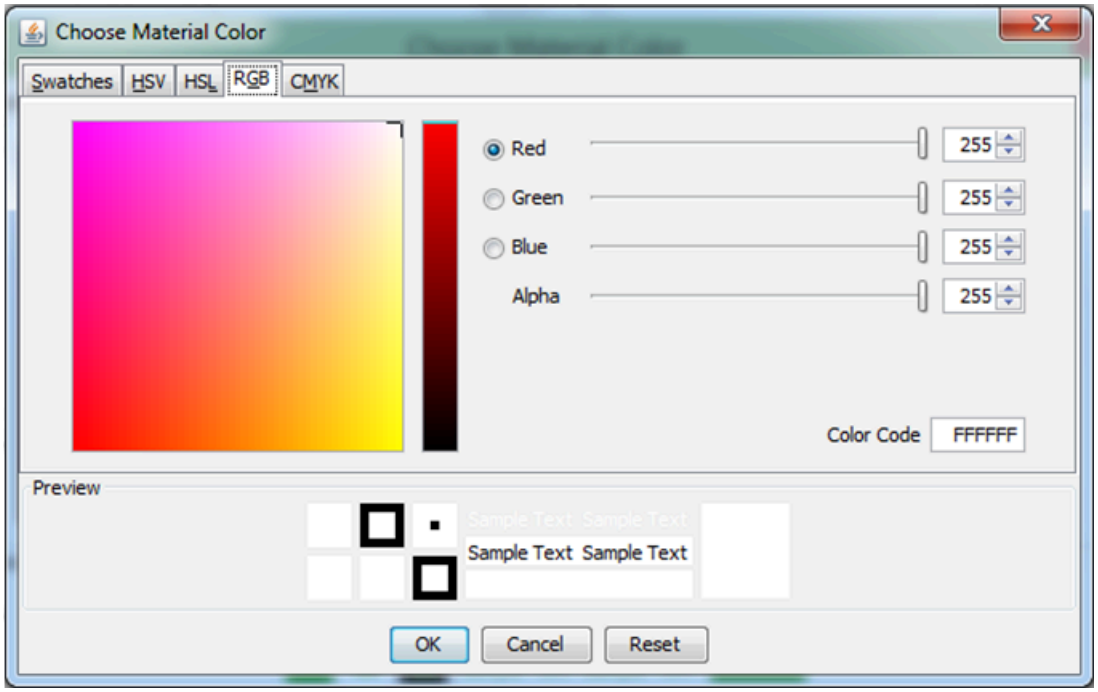


Figure 475: Choose Material Color using the 'RGB' tab

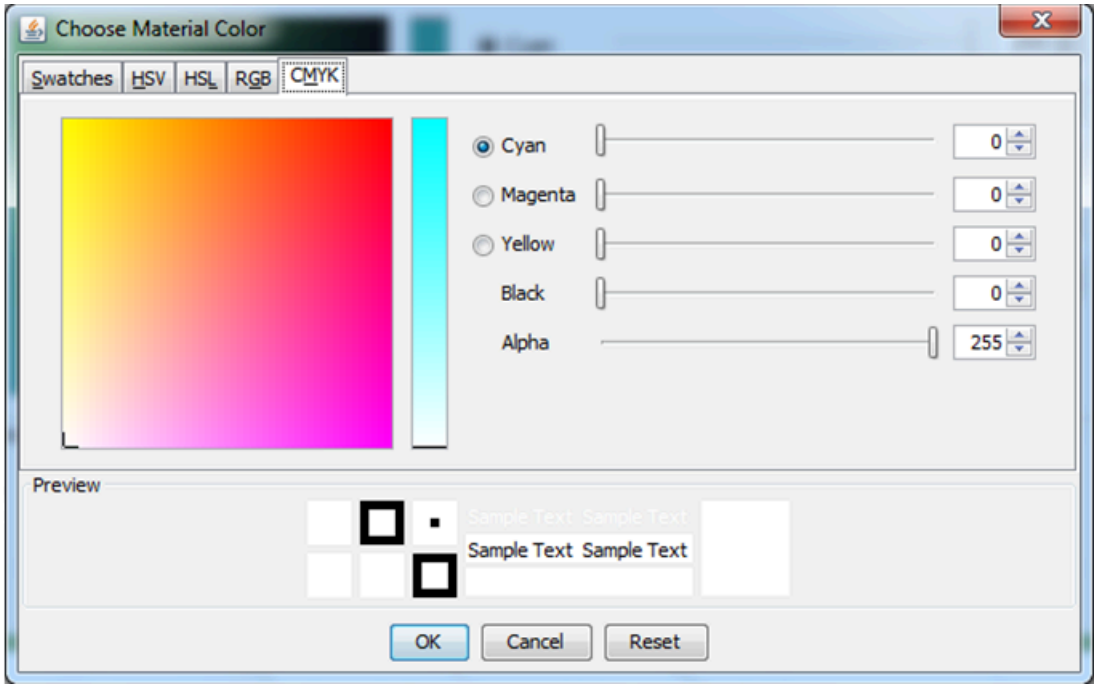


Figure 476: Choose Material Color using the 'CMYK' tab

Command Line

6

The command line can be used to draw geometries or change the GUI configuration.

This chapter covers the following:

- 6.1 Using the Command Line (p. 428)
- 6.2 List of Commands Ordered by Functionality (p. 429)
- 6.3 Command Reference (p. 433)
- 6.4 Appendix I: List of Functions (p. 654)

6.1 Using the Command Line

The command line provides numerous commands that can be used for multiple purposes, like drawing geometries or changing the GUI configuration.

The command line panel is located at the bottom of the main window. In this panel the user can issue commands to be processed by the application.

Figure 477: Calling the box command using interactive mode

Most commands (especially those related to geometry) have two modes of invocation **inline mode** and **interactive mode**. In the inline mode, the command is called by specifying the command name followed by all the arguments the command needs to work. In interactive mode, commands are called by specifying their name (in some cases, a modifier can also be specified to alter its behaviour) and then the command line asks for the required parameters, one by one. The latter mode is easier to use when the user does not know the order of the parameters used by the inline mode (which can be queried by calling the command with the **-h** modifier). It is possible to abort a command while in interactive mode by pressing the Esc key, or to jump into the previous step by pressing the **Ctrl+Left** keys combination. The previous figure shows the box command called in interactive mode, whereas the following figure shows the same command called using the inline mode. Both forms of invocation achieve the same result.

Figure 478: Calling the box command using inline mode

Commands that are used to create geometric primitives are called in a similar way. The -n modifier is optional and, if it's present, it must be followed by a user-given name for the created geometry. This name can be used to refer to the created object in commands that represent geometric operations (like copy, rotate or array). If the -n modifier is not present, the created object will be given an automatically generated name.

In the other hand, the -p modifier is mandatory and must be followed by the parameters needed for the creation of the geometry. In case of doubt about the meaning and the order of the parameters needed to create a certain primitive, it is recommended to run the command with only the -h modifier to obtain help.

For further information about the commands general behaviour, use the **help** command.

6.2 List of Commands Ordered by Functionality

The next table represents an exhaustive list of the commands supported in newFASANT. These commands are ordered by the function they perform.

Category	Commands
Surfaces	bentDipoles: Generates a pair of bent dipoles. capacitance: Generates a capacitance-shaped surface. circularHole: Generates a plane with a circular hole in it. cornerpoints: Generates a surface from 3 or 4 given points cross: Draws a cross-like surface. crossHole: Generates a plane with a cross-shaped hole in it. disk: Draws a circular surface. hexagon: Generated a solid hexagonal surface. hyperboloid: Draws a hyperboloid. interdigitalLines: Generates a stripline interdigital filter. openRing: Generates a ring-shaped surface with an opening. paraboloid: Draws a paraboloid. plane: Draws a plane. rebuildSurface: Rebuilds the selected surface. ring: Generates a ring-shaped surface. spiralCross: Generates a spiral-shaped cross surface. splitRing: Generates a ring-shaped surface with two symmetrically located openings. spiralCross: Generates a hollow cross-shaped surface. stripHexagon: Generates a hollow hexagonal surface. stripTrifilar: Generates a hollow trifilar surface. surfacesurface: Creates a surface with control points. surfaceEdges: Generates the bound curves of the surface selected. surfaceEllipse: Generates a solid ellipse surface. surfaceFunction: Creates a surface with three functions that define points in Cartesian coordinates. surfaceInterpolation: Creates a surface that goes through a set of points specified by the user.

Category	Commands
	surfaceIsoparamCurve: Extracts an isoparametric curve of the selected surface. surfaceSpiral: Generates a surface-spiral from its center. tCross: Generates a cross with T-shaped segments. trifilar: Generates a cross with three segments. rectangularHole: Generates a plane with a rectangular hole in it. threeDipoles: Generates three straight dipoles. threeDipolesUnion: Generates three, joined, straight dipoles.
Objects	box: Draws a box. cone: Draws a cone. cylinder: Draws a cylinder. ellipsoid: Draws an ellipsoid. ogive: Draws a tangent ogive. flare: Draws a flare. sphere: Draws a sphere. torus: Draws a torus.
Mesh	meshFromSurfaces: Groups NURBS surfaces in mesh (degree one and untrimnurbs). meshToSurfaces: Explodes mesh object in NURBS surfaces.
Geometric operations	array: Copies objects in each of the directions of the axes. arrayOnSurface: This option creates an array from planar elements conformed to the selected surface. coons: Creates a coons patch through a selected loop of curves. copy: Copies the selected objects from start to end points. delete: Deletes objects on screen. duplicate: Creates a copy of the selected objects at the same position. explode: Explodes the selected objects on screen. This generates an specific number of surfaces. extendHole: Extends a hole created by a surface and a loop of curves. extrude: Creates a surface by sweeping a curve along a vector.

Category	Commands
	extrudeNormal: Creates an object by sweeping the selected surfaces along their normal vectors. extrudeSurface: Creates an object by sweeping a surface along a vector. group: Groups a set of surfaces into a single object. invertNormals: Inverts the normal vectors of curves and surfaces. localTransform: Transforms a object or more objects (assuming they are designed with respect to the absolute coordinate system) to the coordinate system defined by the current reference plane. move: Moves objects by applying a translation vector. pipe: Creates a circular cross-section tube that surrounds the selected curve. revolve: Creates a surface by revolving a curve an specific number of degrees around a rotational axis. rotate: Rotates objects a certain angle using a start point and an end point. scale: Scales objects with a specific scale factor regarding the reference plane or ignoring it. scale1D: Scales only one of the X, Y or Z dimension of objects with a specific scale factor regarding the reference plane or ignoring it. scale2D: Scales two of the X, Y or Z dimensions of objects with a specific scale factor regarding the reference plane or ignoring it. scale3D: Scales the three X, Y or Z dimension of objects with a specific scale factor regarding the reference plane or ignoring it. scaleNonUniform: Scales the three X, Y or Z dimension of objects with a specific scale factor for each dimension regarding the reference plane or ignoring it. skinned: Interpolates a surface through the given curves. surfaceEdges: Creates a surface defined by a closed boundary loop of curves. symmetric: Reflects objects with respect a plane of symmetry. trim: Trims a surface using a curve. untrim: Undoes trim operations.
Interface operations	axis: Changes the configuration of the display axes. exit: Exits the application. export: Exports geometry file (NUR, IGES, DXF, MSH, NAS…). import: Imports geometry file (NUR, IGES, DXF, MSH, NAS…).

Category	Commands
	rename: Rename object from an actual name to a new name. script: Loads script and execute it or save history to script.
Parameter operations	set: Assigns a set of values to a parameter. unset: Undeclares a parameter.
Boolean operations	booleanInside: Regions of an object inside another one booleanIntersection: AND operation between two objects booleanOutside: Regions of an object outside another one booleanSplit: Split two objects booleanDifference: Difference operation between two objects booleanUnion: OR operation between two objects booleanXOR: XOR operation between two objects planarSurface: Generates planar surfaces for closed loops of curves. splitByProjection: Split surfaces with the projection of the curves. splitCurves: Splits between all selected curves. splitSurfaces: Splits between surfaces or by curves. splitSurfaceByIsoparam: Split a surface by an isoparametric line. splitSurfaceByIsoparamGrid: Split a surface by a grid of isoparametric lines.

6.3 Command Reference

- analyzeCurvature
- analyzeEdges
- analyzeNakedBorders
- angle
- arc
- area
- array
- arrayOnSurface
- axis
- bentDipoles
- booleanDifference
- crossHolebooleanInside
- booleanIntersection
- booleanOutside
- crossHole
- booleanSplit
- cross
- booleanUnion
- crossHole
- booleanXOR
- box
- capacitance
- circle
- circularHole
- cone
- connect2Surfaces
- connectCurves
- coons
- copy
- cross
- crossHole
- curve
- curveFunction
- curveInterpolation
- cylinder
- delete

- disk
- distance
- duplicate
- ellipse
- ellipsoid
- exit
- explode
- export
- extendHole
- extrude
- extrudeNormal
- extrudeSurface
- flare
- group
- help
- hexagon
- hyperbola
- hyperboloid
- import
- interdigitalLines
- invertNormals
- length
- line
- localTransform
- meshFromSurfaces
- meshToSurfaces
- move
- ogive
- openRing
- parabola
- paraboloid
- perpendicular
- pipe
- planarSurface
- plane
- point
- pointCloud
- pointsFromCurve
- pointsFromCurveByDistance

- pointsFromSurface
- polygon
- project
- rebuildCurve
- rebuildSurface
- rectangle
- rectangularHole
- rename
- revolve
- rhomboid
- ring
- rotate
- scale
- scale1D
- scale2D
- scale3D
- scaleNonUniform
- script
- segment
- semicircle
- set
- skinned
- sphere
- spiral
- spiralCross
- splitByProjection
- splitCurves
- splitRing
- splitSurfaceByIsoparam
- splitSurfaceByIsoparamGrid
- splitSurfaces
- stripCross
- stripHexagon
- stripTrifilar
- surface
- surfaceCornerPoints
- surfaceEdges
- surfaceEllipse
- surfaceFromEdges

- surfaceFunction
- surfaceInterpolation
- surfaceIsoparamCurve
- surfaceSpiral
- sweep
- symmetric
- tangent
- tCross
- threeDipoles
- threeDipolesUnion
- torus
- trifilar
- trim
- unset
- untrim

6.3.1 analyzeCurvature

This option is useful to identify the curves and surfaces in the geometry that describe large curvatures. Given a desired angle as reference (in degrees), if some of the selected objects describe curvatures bigger than the reference angle the points are generated where the curves has varied their direction every angular step. The detected points may be only visualized or also added to the geometry.

Inline mode usage:

analyzeCurvature -h
Displays the help file that summarizes the parameters for this command.

analyzeCurvature
Identifies the points of curvature according to the given angle.

Interactive mode usage:

Invocation
analyzeCurvature [-all]

Parameters

- Selection of the curves, surfaces and objects to be analyzed.
- Division angle (in degrees).

6.3.2 analyzeEdges

This tool allows the user to detect the edges in the geometry. The edges may be only visualized or also added to the geometry.

Inline mode usage:

analyzeEdges -h
Displays the help file that summarizes the parameters for this command.

analyzeEdges
Shows the detected edges in the whole geometry.

analyzeEdges –s
Shows the detected edges only in the selected geometry.

Interactive mode usage:

Invocation
analyzeEdges -s

Parameters

- Selection of the surfaces and objects to be analyzed.

6.3.3 analyzeNakedBorders

This tool allows the user to detect the naked borders (borders of the surfaces that are not in contact with other ones) in the geometry. The borders may be only visualized or also added to the geometry.

Inline mode usage:

analyzeNakedBorders -h
: Displays the help file that summarizes the parameters for this command.

analyzeNakedBorders
: Shows the detected edges in the whole geometry.

analyzeNakedBorders –s
: Shows the detected edges only in the selected geometry.

Interactive mode usage:

- analyzeNakedBorders -s
- **Parameters**:
  - Selection of the surfaces and objects to be analyzed.

6.3.4 angle

This tool allows the user to calculate the angle (in degrees) between two given vectors. Vectors are specified by an initial point and an end point.

Inline mode usage:

angle -h
Displays the help file that summarizes the parameters for this command.

angle <x1 y1 z1> <x2 y2 z2> <x3 y3 z3> <x4 y4 z4>
Calculates the angle between the vectors. <x1 y1 z1> and <x2 y2 z2> refer to the initial and final Cartesian coordinates of first vector and <x3 y3 z3> and <x4 y4 z4> refer to the initial and final Cartesian coordinates of second vector, respectively.

Interactive mode usage:

Invocation
angle

Parameters

- Initial coordinate for the first vector: x1 y1 z1.
- Final coordinate for the first vector: x2 y2 z2.
- Initial coordinate for the second vector: x3 y3 z3.
- Final coordinate for the second vector: x4 y4 z4.

All coordinates are specified by writing each of their three spatial components separated by a blank space.

Example:

```
>angle
```

```
Select first point for first vector [x y z] 0 0 0
```

```
Select second point for first vector [x y z] 3 0 0
```

```
Select first point for second vector [x y z] 0 0 0
```

```
Select second point for second vector [x y z] 0 2 0
```

```
The angle between given points is 90.0
```

6.3.5 arc

This command allows the user to draw a circumference arc specifying its central point, radius, initial and final angles (in degrees), in this order.

Inline mode usage:

arc -h
Displays the help file that summarizes the parameters for this command.

arc –n <name> -p <x_center> <y_center> <z_center> <radius> <initial_angle> <final_angle>
Draws a circle arc named <name> between the <initial_angle> and the <final_angle> (both specified in degrees), and with the specified radius and center point.

Interactive mode usage:

Invocation
arc

Parameters

- Center point, specified in "x y z" format x y z.
- Radius: radius.
- Initial angle: start_angle.
- Final angle: end_angle.

Both angles are specified in degrees.

Example:

```
>arc
```

```
Center [x y z] 0 1 0
```

```
Radius [double] 3
```

```
Initial angle [degrees] 45
```

```
Final angle [degrees] 100
```

Result:

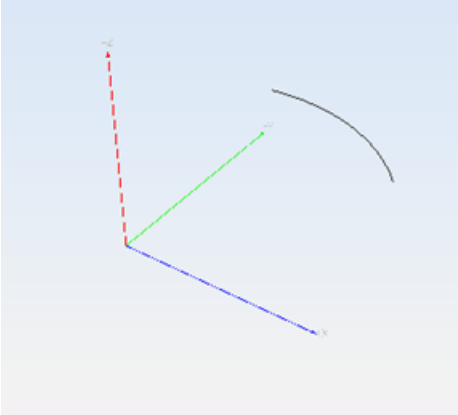


Figure 479: Resulting arc

6.3.6 area

This command returns the total area of the selected surfaces.

Inline mode usage

area -h
Displays the help file that summarizes the parameters for this command.

area -all
The total area of the whole geometry is computed.

Interactive mode usage

Invocation
area

Parameters
Selection of surfaces and objects to be analyzed.

Example

```
command> disk
```

```
Select center [x y z] 0 0 0
```

```
Radius [double] 0.5
```

```
command> area
```

```
Select the surfaces to be analyzed (Press enter when done)
```

```
Please Wait...
```

```
The area of the selected geometry is 0.7853981633982382 square meters
```

```
command>
```

6.3.7 array

Copies an object or a set of objects a given number of times in the direction of each of the axes. This option lets the user easily create a matrix of geometries.

Inline mode usage

array -h
Displays the help file that summarizes the parameters for this command.

array –s <name> [-cir <radius>] –p <i> <j> <k> <x1> <y1> <z1> <x2> <y2> <z2> :
Copy <name> object <i> times in X axis, <j> times in Y axis and <k> times in Z axis, starting with <x1 y1 z1> point and ending in <x2 y2 z2> point.

Interactive mode usage

Invocation

- array -all: Makes an array for all objects.
- array -cir <radius>: Makes a circular array by removing elements that are in a radius greater than the <radius> parameter.
- array: Copy the selected objects on the screen.

Parameters

- Objects to copy (used only when not specifying -h or -all parameters). These objects must be selected by left-clicking on each of them (hold Ctrl to select several objects) or selecting them on the tree view.
- Number of copies in the X axis: i.
- Number of copies in the Y axis: j.
- Number of copies in the Z axis: k.
- Starting point: x1 y1 z1.
- End point: x2 y2 z2.

Both the starting point and the end point are specified by their Cartesian coordinates. These two points define a vector whose projection along each of the axes define the location that each successive copy creates along the axis.

As an example, if the starting point is (1, 2, 3) and the end point is (6, 5, 4), the resulting translation vector will be (5, 3, 1). The projection of this vector along the X axis is the vector (5, 0, 0), so in this case each successive copy along the X axis will be spaced by 5 units. Similar calculations can be done with respect to the Y and Z axis.

Example

A 2x2x2 box was created prior to this operation:

```
> array
```

```
Select objects on screen (Press enter when done) // Left-click on the box appearing
 on the screen display
```

```
Number of copies in X axis: 2
```

```
Number of copies in Y axis: 3
```

```
Number of copies in Z axis: 4
```

```
Select start point [x y z]: 0 0 0
```

```
Select end point [x y z]: 5 4 3
```

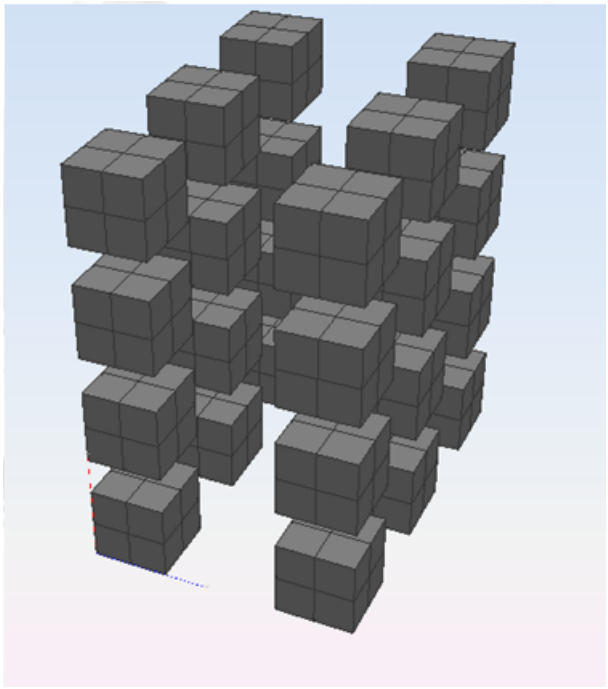


Figure 480: Result of the array operation

6.3.8 arrayOnSurface

Creates an array of planar elements along surface.

Inline mode usage

ArrayOnSurface -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

arrayOnSurface

Parameters

- Select surfaces on screen (Press enter when done)
- Size of cell structure [Tx Ty] (Object with planar elements must be in cell)
- Select array mode [1-Parametric,2-Global](default=2) (Global mode requires Tx=Ty)
- Select array type [1-Capacitive,2-Inductive](default=1)
- Select one object with planar elements (Press enter when done)

Example

The next figure shows the full example of the disk array on the sphere with all the parameters required by default (the original objects have not been deleted).

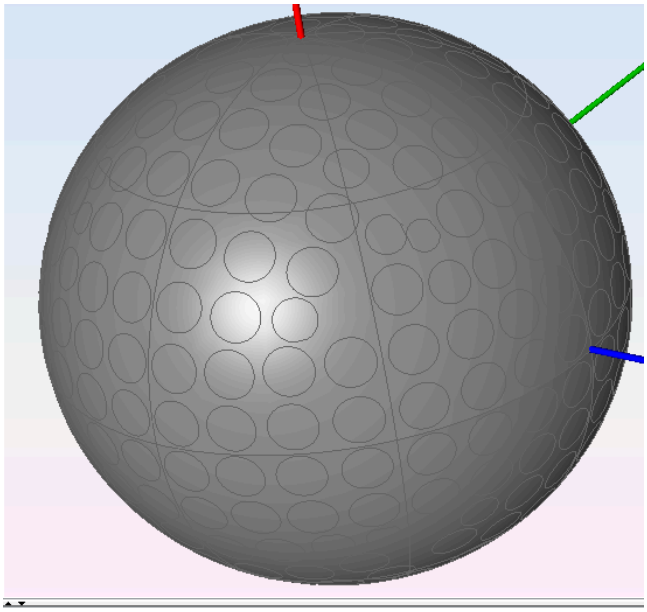


Figure 481: Result of the execution of the arrayOnSurface command with default parameters

The next figure shows the full example of the disk array on the sphere generated with parametric mode (the original objects have not been deleted).

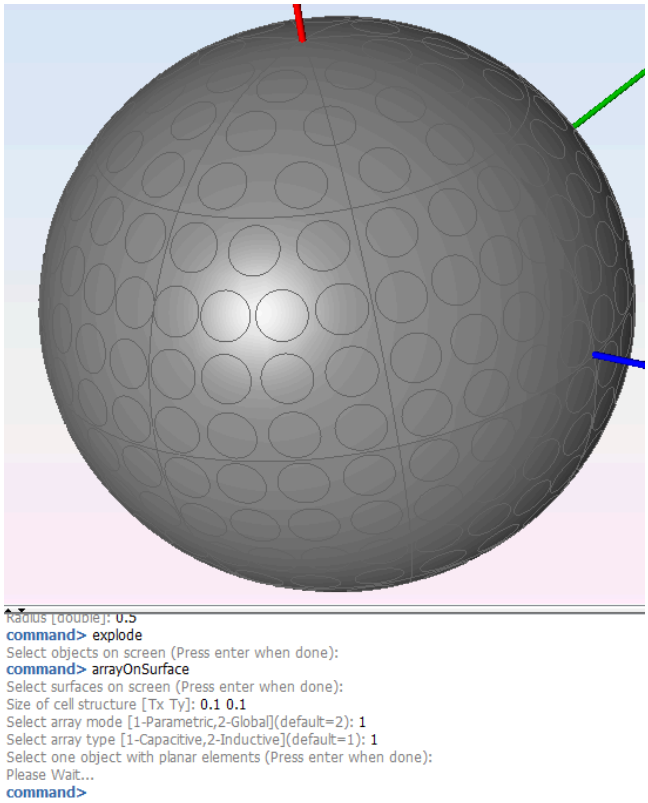


Figure 482: Result of the execution of the arrayOnSurface command using parametric mode

The next figure shows again the full example of the disk array on the sphere generated with the global mode and the inductive type (the original objects have been deleted).

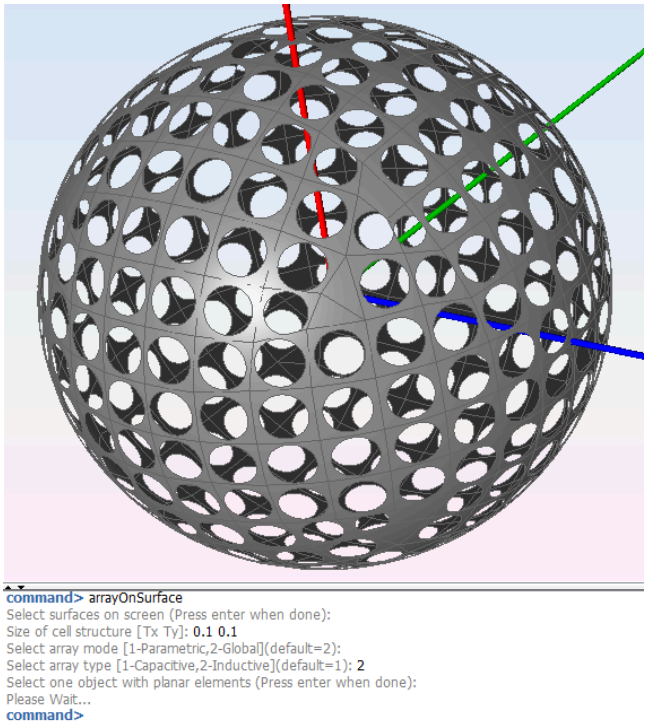


Figure 483: Result of the arrayOnSurface command with global array mode and inductive array type

The next figure shows again the full example of the disk array on the sphere generated with the parametric mode and the inductive type (the original objects have been deleted).

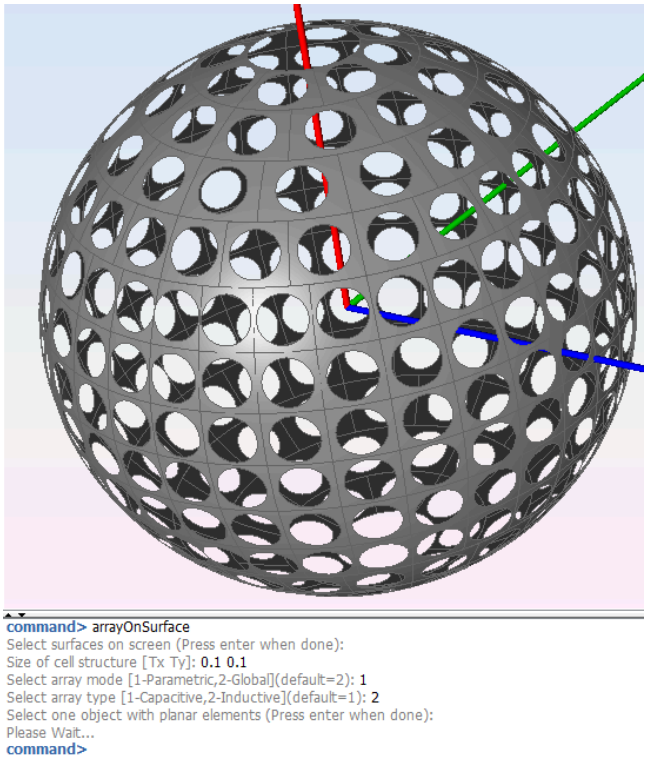


Figure 484: Result of the arrayOnSurface command with parametric array mode and inductive array type

6.3.9 axis

This option toggles the visualization of the three axes in the view area and allows to modify their length.

Inline mode usage:

- axis -h: Displays the help file that summarizes the parameters for this command.
- axis off: Turns off the visualization of the axes.
- axis on: Turns on the visualization of the axes.
- axis -size <value>: Sets the length of each of the axis to the specified value.

6.3.10 bentDipoles

Add new "Bent dipoles" structure to the geometry.

Inline mode usage:

- bentDipoles -h: Display a help message.
- bentDipoles -n name -p x y z separation firstDipoleHeight firstDipoleWidth firstDipoleLength secondDipoleHeight secondDipoleWidth secondDipoleLength: Creates a new pair of bent dipoles.

Interactive mode usage:

The following steps will be needed in order to create a new pair of bent dipoles.

1. Select center [x y z]: Central point of the structure (equidistant to the vertical segment of both dipoles).
2. Separation [double]: Separation between dipoles
3. Height of first dipole [double]: Height of the first dipole.
4. Width of first dipole [double]: Width of the first dipole.
5. Length of first dipole [double]: Length of the first dipole.
6. Height of second dipole [double]: Height of the second dipole.
7. Width of second dipole [double]: Width of the second dipole.
8. Length of second dipole [double]: Length of the second dipole.

Example:

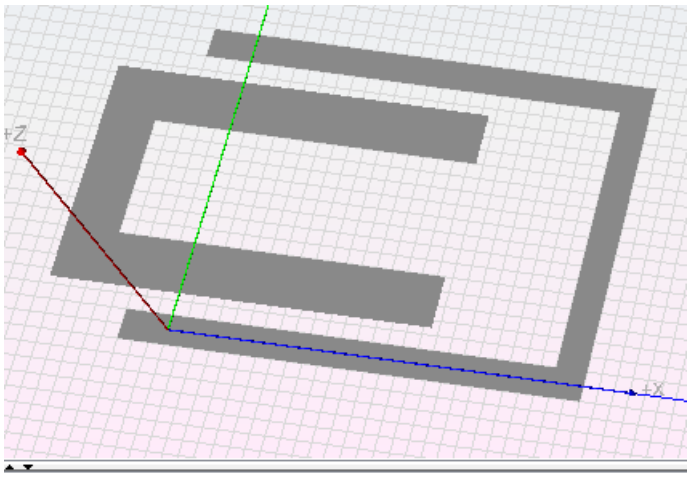


Figure 485: bentDipoles example

6.3.11 booleanDifference

This command computes the resulting surfaces of subtracting the second object to the first one (A minus B operation), that is, the regions of the first object that are not contained in the second object.

Inline mode usage:

booleanDifference -h
Displays the help file that summarizes the parameters for this command.

booleanDifference –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the objects of A minus the objects of B.

Interactive mode usage:

- booleanDifference
- **Parameters**:
- ◦ Surfaces of the 'objectA' selection of surfaces and objects of the first object.
  ◦ Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose we want to create a sphere with a sphere-shaped hole in its surface. To do this, we need to create a whole sphere (using the sphere command) and the sphere that will represent the hole we want to make in the previously created sphere. The following commands will be used for the creation of the two spheres:

```
command> sphere
```

```
Select center [x y z]: 0 0 0
```

```
Radius [double]: 2
```

```
command> sphere
```

```
Select center [x y z]: 1 1 1
```

```
Radius [double]: 0.75
```

```
command>
```

The state of the geometry at this moment is shown in the next figure:

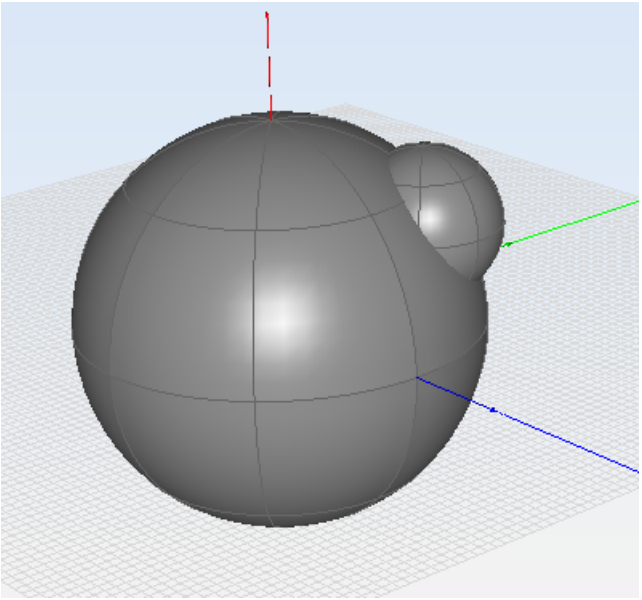


Figure 486: State of the geometry before the booleanDifference operation

To create the desired object, we need to subtract the regions of the small sphere from the regions of the big sphere. This can be done in the following way:

```
command> booleanDifference
```

```
Select the surfaces of the 'objectA' on screen (Press enter when done) (Select the
 big sphere)
```

```
Select the surfaces of the 'objectB' on screen (Press enter when done) (Select the
 small sphere)
```

```
Please Wait...
```

```
command>
```

If the spheres had names, the command could also be run in the following way (assuming the spheres have, as names, "bigSphere" and "smallSphere"):

```
command> booleanDifference -s -objectA bigSphere -objectB smallSphere
```

```
Please Wait...command>
```

The final result is shown in the next figure:

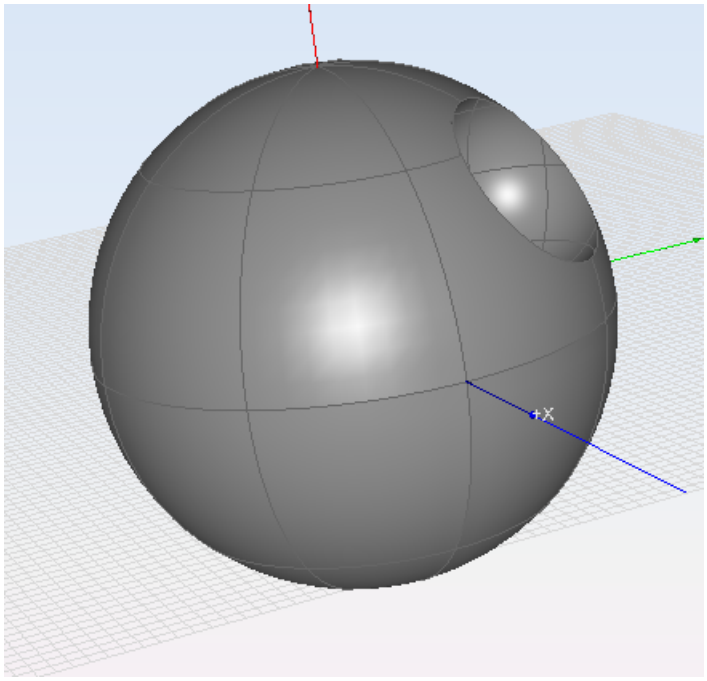


Figure 487: State of the geometry after the booleanDifference operation

6.3.12 booleanInside

This command select the surfaces of the first object that are contained in the second one (A inside B operation).

Inline mode usage:

booleanInside -h
Displays the help file that summarizes the parameters for this command.

booleanInside –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the surfaces listed in A that are contained in B.

Interactive mode usage:

- booleanInside
- **Parameters**:
- ◦ Surfaces of the 'objectA' selection of surfaces and objects of the first object.
  ◦ Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose we have the following geometry, which is the same as the created for example of the booleanDifference command:

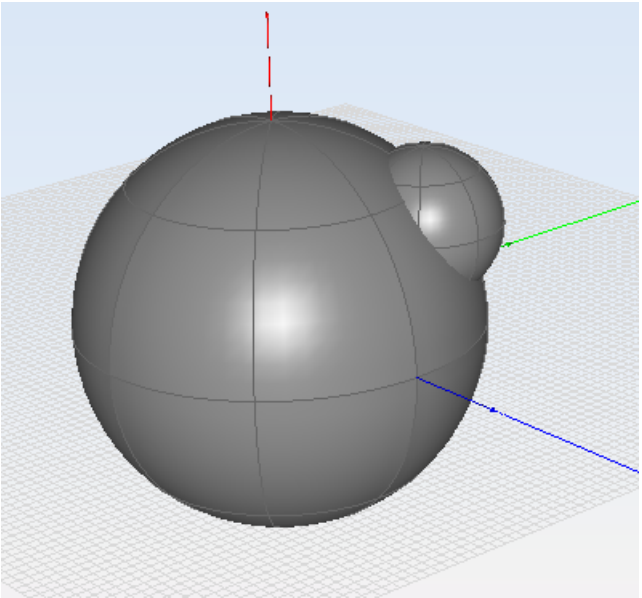


Figure 488: Initial state of the geometry for the booleanInside command

With the booleanInside command it is posisble to obtain the surfaces of the small sphere that are contained inside the big sphere. To do this, we have to run the booleanInside command in the following way:

```
command> booleanInside
```

```
Select the surfaces of the 'objectA' on screen (Press enter when done) (Select the
 small sphere)
```

```
Select the surfaces of the 'objectB' on screen (Press enter when done) (Select the
 big sphere)
```

```
Please Wait...
```

```
command>
```

As usual, the command can be run in non-interactive mode by specifying the names of the objects:

```
command> booleanInside -s -objectA smallSphere -objectB bigSphere
```

```
Please Wait...
```

```
command>
```

The final result is shown in the next figure:

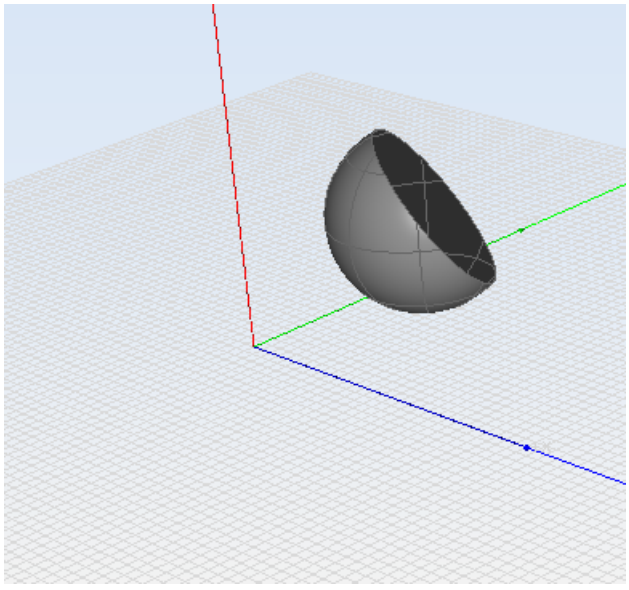


Figure 489: State of the geometry after the booleanInside operation

6.3.13 booleanIntersection

This command creates an object with the regions of the two selected objects that are included in both of them (A AND B operation). Note that the original objects used for the operation will be deleted.

Inline mode usage:

booleanIntersection -h
Displays the help file that summarizes the parameters for this command.

booleanIntersection –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the surfaces listed in A and B that are contained in both objects.

Interactive mode usage:

- booleanIntersection
- **Parameters**:
- ◦ Surfaces of the 'objectA' selection of surfaces and objects of the first object.
  ◦ Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose we have a box and an overlapping sphere, like the shown in the next figure:

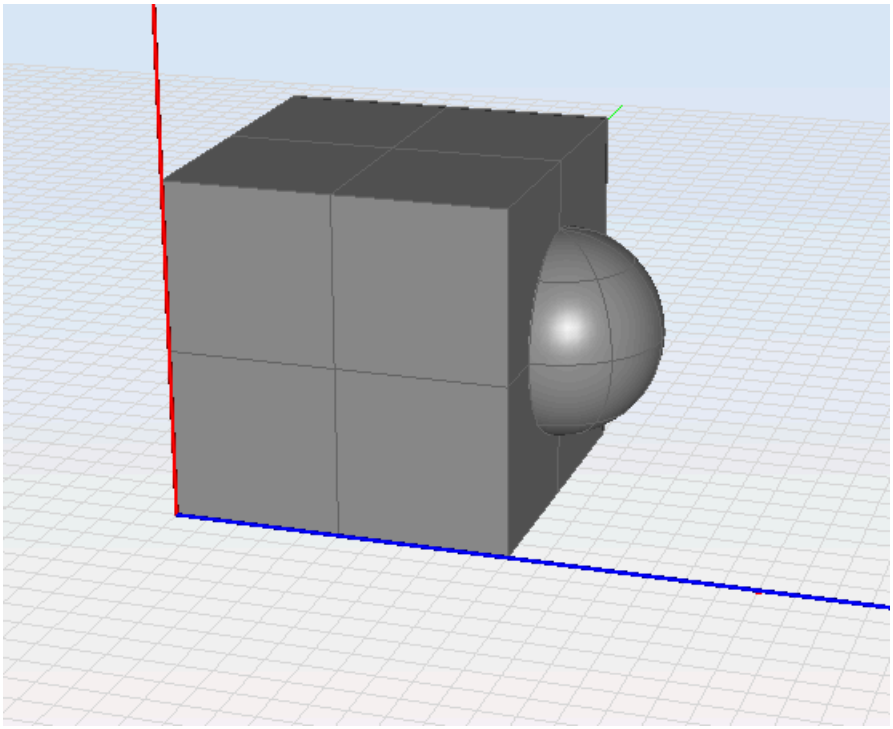


Figure 490: Initial state of the geometry for the booleanIntersection command

This geometry was created by running the following commands:

```
command> boxFirst corner of base [x y z]: 0 0 0
```

```
Base size [width depth]: 1 1
```

```
Height [double]: 1
```

```
command> sphere
```

```
Select center [x y z]: 1 0.5 0.5
```

```
Radius [double]: 0.3
```

```
command>
```

Getting an object of the shared regions between the two objects is possible by running the booleanIntersection command in the following way (note that the boolean intersection operation is commutative, so the order of the selection does not matter):

```
command> booleanIntersection
```

```
Select the surfaces of the 'objectA' on screen (Press enter when done) (Select one of
 the objects)
```

```
Select the surfaces of the 'objectB' on screen (Press enter when done) (Select the
 other object)
```

```
Please Wait...
```

```
command>
```

As usual, it is possible to run the command in non-interactive mode by using the object names. Assuming the names of the objects are "sphere" and "box":

```
command> booleanIntersection -s -objectA sphere -objectB box
```

```
Please Wait...
```

```
command>
```

The final state of the geometry is shown in the following figure:

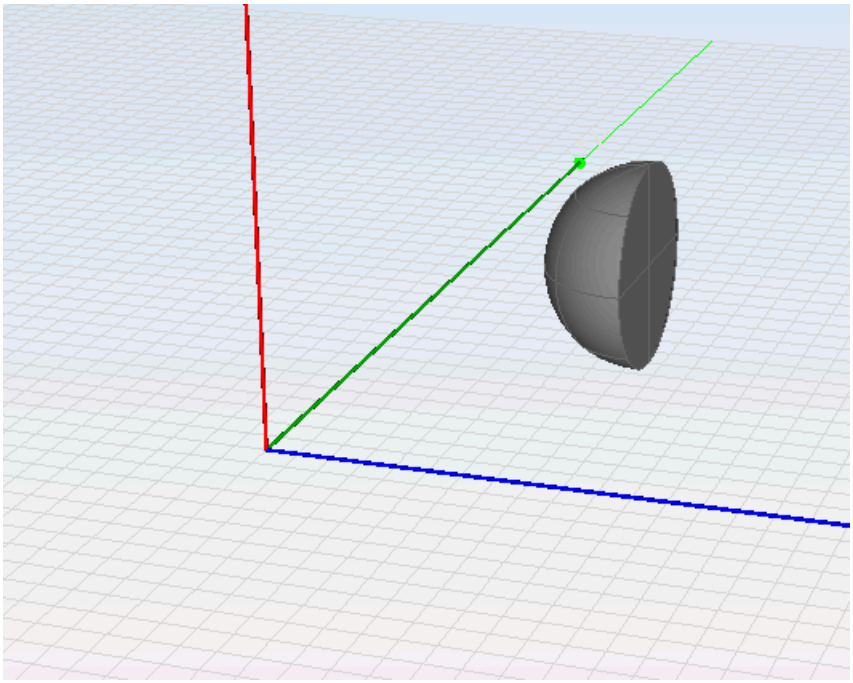


Figure 491: State of the geometry after the booleanIntersection operation

Note that the booleanIntersection operation computes the **regions** that are shared between the objects (therefore, this operation is commutative). On the other hand, the booleanInside operation calculates only the **parts of the surface** of an object that are inside the other object (this operation is not commutative). As a result, the output of the booleanIntersection operation is a closed surface because it encloses a region (assuming the input objects are also closed surfaces, as in this example), while the booleanInside operation would not output a closed surface because its output is only the part of the surface of an object that is enclosed inside another object.

6.3.14 booleanOutside

This command select the surfaces of the first object that are not contained in the second one (A outside B operation).

Inline mode usage:

booleanOutside -h
Displays the help file that summarizes the parameters for this command.

booleanOutside –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the surfaces listed in A that are not contained in B.

Interactive mode usage:

Invocation
booleanOutside

Parameters

- Surfaces of the 'objectA' selection of surfaces and objects of the first object.
- Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose we have the following open cone and box in the geometry:

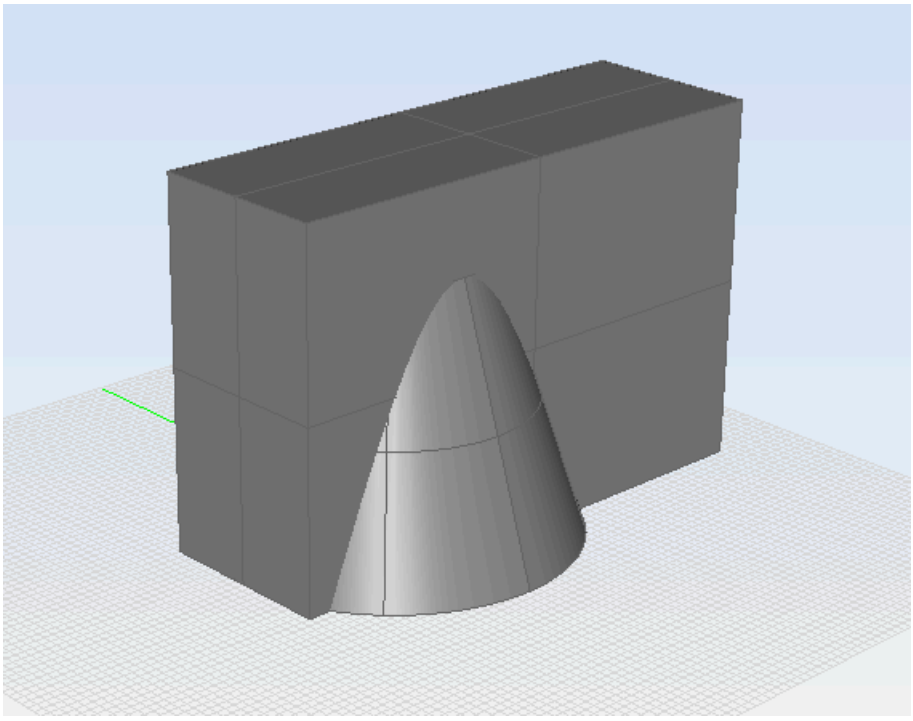


Figure 492: Initial state of the geometry for the booleanOutside command

These objects can be created by running the following commands:

```
command> cone
```

```
Select center [x y z]: 0 0 0
```

```
Bottom radius [double]: 2
```

```
Height [double]: 3
```

```
Top radius [double]: 1
```

```
command> box
```

```
First corner of base [x y z]: -2 -1 0
```

```
Base size [width depth]: 6 2
```

```
Height [double]: 4
```

```
command>
```

Let's assume we want to obtain the part of the surfaces of the cone that are outside the region enclosed by the box. To achieve this, we need to run the booleanOutside command in the following way:

```
command> booleanOutside
```

```
Select the surfaces of the 'objectA' on screen (Press enter when done) (Select the
 cone)
```

```
Select the surfaces of the 'objectB' on screen (Press enter when done) (Select the
 box)
```

```
Please Wait...
```

```
command>
```

As usual, if the cone and the box had the names "myCone" and "myBox" respectively, the command could also be run in the following way:

```
command> booleanOutside -s -objectA myCone -objectB myBoxPlease Wait...command>
```

The final result is shown in the following figure:

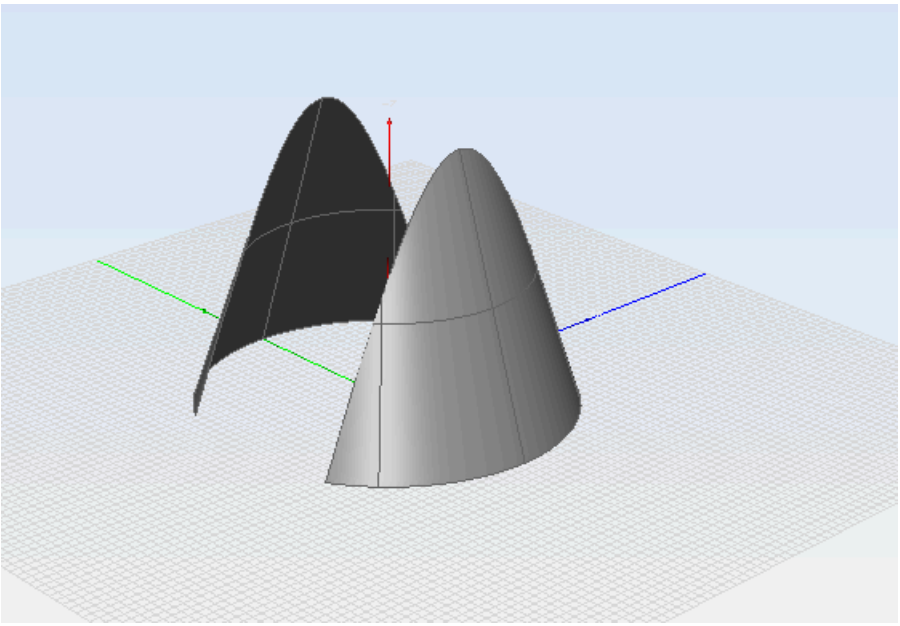


Figure 493: State of the geometry after the booleanOutside operation

6.3.15 booleanSplit

This command introduces the cuts that an object produces in another one.

Remark If the object that this command is called on is a surface, the surface itself will be split into several surfaces. However, if it is an object, this command will only create a cutting loop on the object. In the latter situation, the user may want to explode the object so it is possible to manipulate the surfaces delimited by the cutting loop individually.

Inline mode usage:

booleanSplit -h
Displays the help file that summarizes the parameters for this command.

booleanSplit –s –objectA <list of names A> -objectB <list of names B>
Split mutually the selected objects.

Interactive mode usage

Invocation
booleanSplit

Parameters

- Surfaces of the 'objectA' selection of surfaces and objects of the first object.
- Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Let's assume we have the following geometry, consisting on a sphere and a surface:

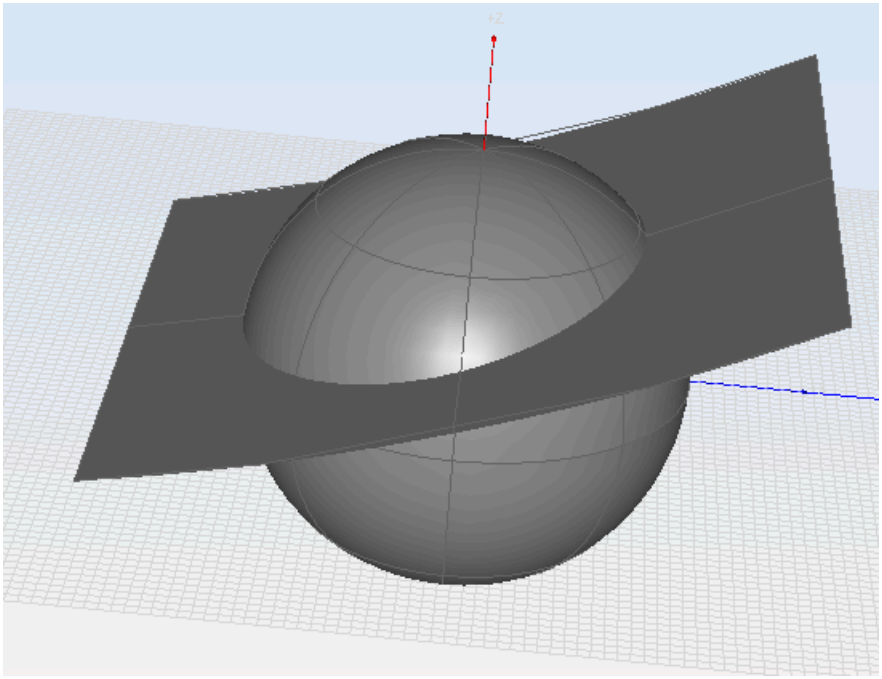


Figure 494: Initial state of the geometry for the booleanSplit command

This geometry was created using the following commands:

```
command> sphere
Select center [x y z]: 0 0 0
Radius [double]: 2
command> surface
Points in u direction [integer](default=3): 3
Points in v direction [integer](default=3): 2
Point [0][0] [x y z w]: -3 -3 0 1
Point [0][1] [x y z w]: -3 3 0 1
Point [1][0] [x y z w]: 0 -3 0.5 1
Point [1][1] [x y z w]: 0 3 0.6 1
Point [2][0] [x y z w]: 3 -3 2 1
Point [2][1] [x y z w]: 3 3 2.1 1
command>
```

Now we will use the booleanSplit command on these two objects to create a cutting loop on the sphere at the intersection between it and the surface. To do this, we run the command in the following way:

```
command> booleanSplit
Select the surfaces of the 'objectA' on screen (Press enter when done): (Select one
 of the objects)
Select the surfaces of the 'objectB' on screen (Press enter when done): (Select the
 other object)
Please Wait...
command>
```

This will create cutting loops on **both** the sphere and the surface. The surface will be split into two surfaces the part that is outside of the sphere, and the part that is inside of the sphere. However, the sphere will not be split because it is considered a 3D object. Nevertheless, it is possible to explode the sphere (so it is divided into individual surfaces) and, because the cutting loop was created, we can select the surfaces of both parts of the sphere (above and below the cutting loop) independently, as shown in the next figure:

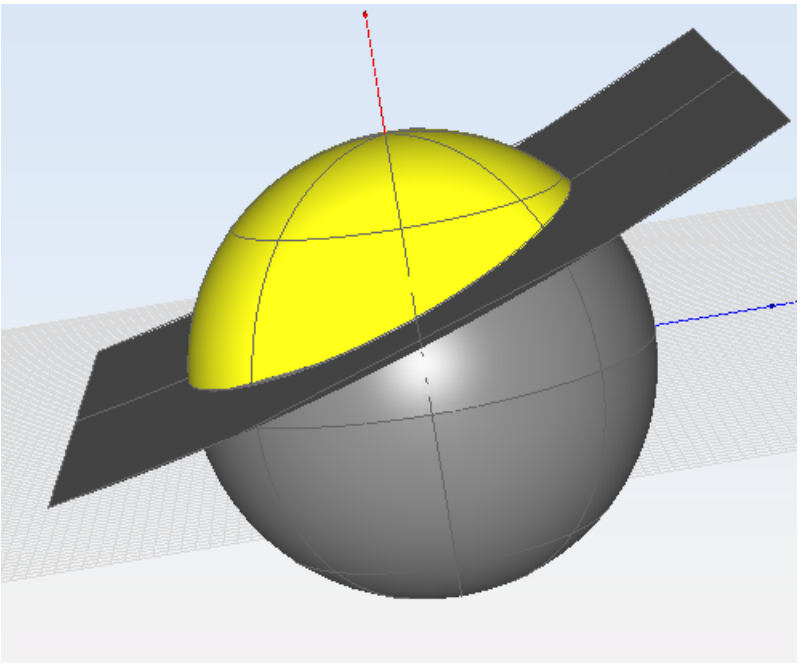


Figure 495: Selecting the part of the surface above the cutting loop

6.3.16 booleanUnion

This command creates a new object that encloses the regions of the two selected objects that are included in any of them (A OR B operation). Note that this command will delete the original objects.

Inline mode usage:

booleanUnion -h
Displays the help file that summarizes the parameters for this command.

booleanUnion –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the surfaces listed in A and B that are contained in any object.

Interactive mode usage

Invocation
booleanUnion

Parameters

- Surfaces of the 'objectA' selection of surfaces and objects of the first object.
- Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose that we have created the two spheres shown in the next figure:

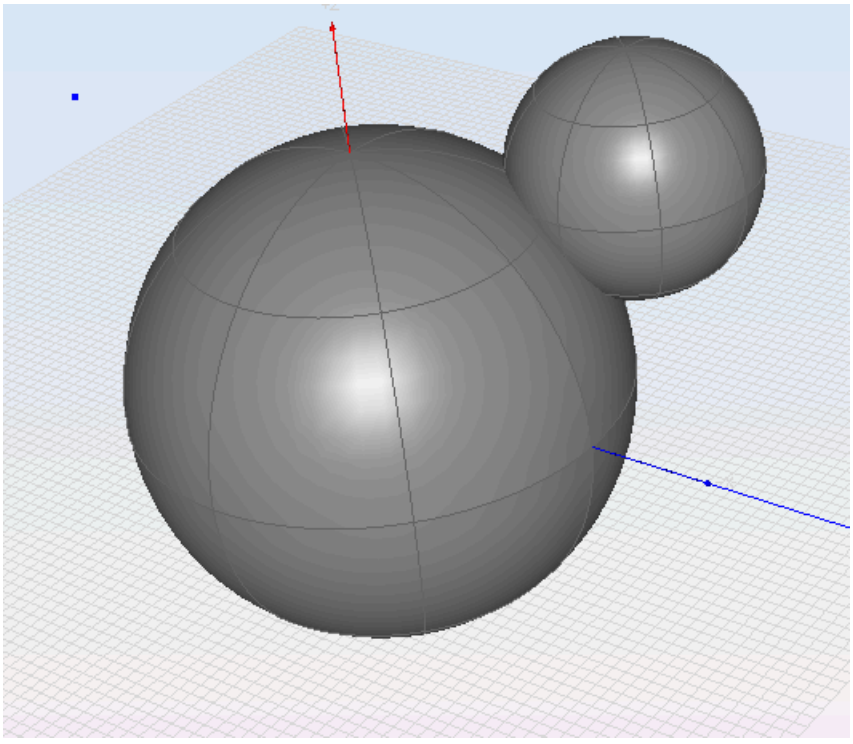


Figure 496: Initial state of the geometry for the booleanUnion command

The geometry was created by using the following commands:

```
command> sphere
Select center [x y z]: 0 0 0
Radius [double]: 2
command> sphere
Select center [x y z]: 1.5 1.5 1.5
Radius [double]: 1
```

We want to create an object that encloses the regions of both spheres. To do this, we need to run the booleanUnion command on both objects:

```
command> booleanUnion
Select the surfaces of the 'objectA' on screen (Press enter when done) (Select one
 sphere)
Select the surfaces of the 'objectB' on screen (Press enter when done) (Select the
 other sphere)
Please Wait...
command>
```

If the spheres were named "sphere1" and "sphere2", the user could run the command in non-interactive mode in the following way:

```
command> booleanUnion -s -objectA sphere1 -objectB sphere2
Please Wait...
command>
```

This command will create the surfaces, grouped as an object, needed to enclose the regions of both objects. Visually, the result is the same as shown in the previous figure.

6.3.17 booleanXOR

This command selects the surfaces of the two selected objects that are included only in one of them (A XOR B operation).

Inline mode usage:

booleanXOR -h
Displays the help file that summarizes the parameters for this command.

booleanXOR –s –objectA <list of names A> -objectB <list of names B>
Draws the parts of the surfaces listed in A and B that are contained only in an object.

Interactive mode usage

Invocation
booleanXOR

Parameters

- Surfaces of the 'objectA' selection of surfaces and objects of the first object.
- Surfaces of the 'objectB' selection of surfaces and objects of the second object.

Example:

Suppose we have the following boxes in the geometry:

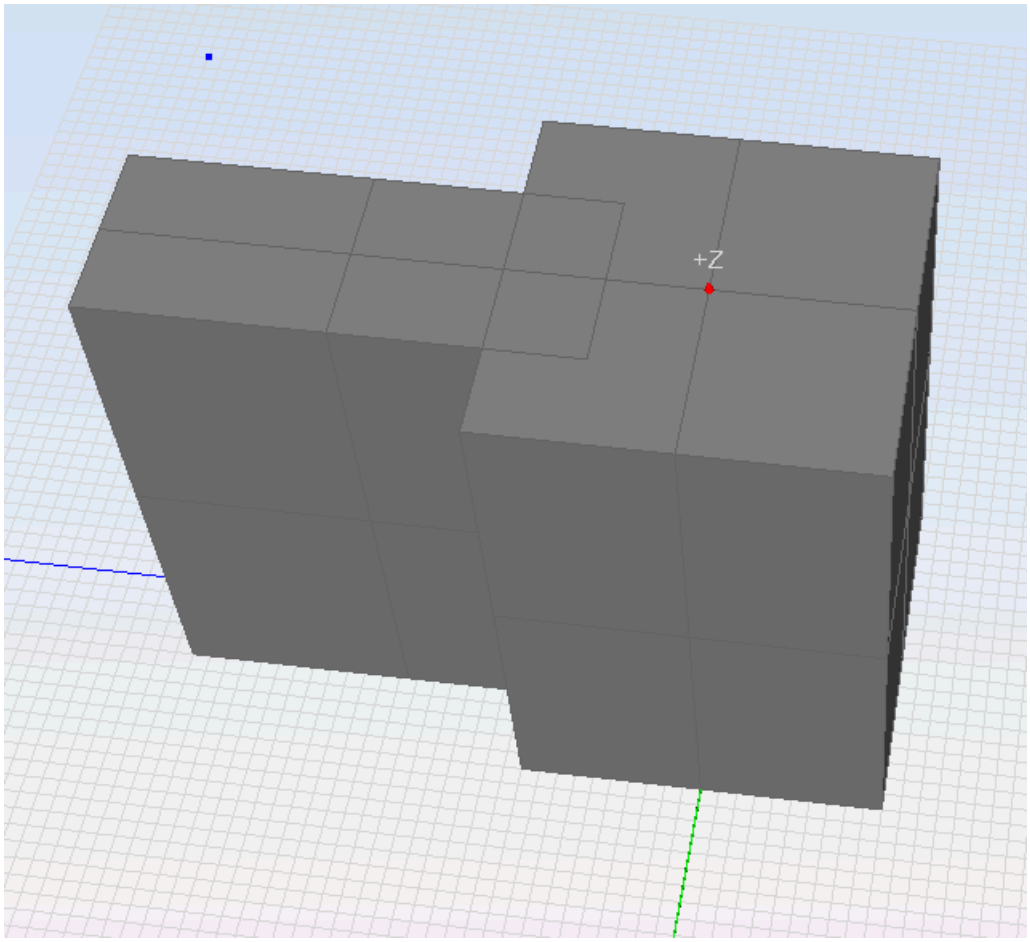


Figure 497: Initial state of the geometry for the booleanXOR command

Note that the box in the left is overlapping the box in the right, so both boxes have a shared region. These two boxes are created with the following commands:

```
command> box
First corner of base [x y z]: -1 -1 0
Base size [width depth]: 2 2
Height [double]: 3
command> box
First corner of base [x y z]: 0.5 -0.5 0
Base size [width depth]: 2.5 1
Height [double]: 3
command>
```

We want to run a boolean XOR operation in these two boxes, in order to end up with an object that encloses the non-shared regions of both boxes. To do this, we run the booleanXOR command in the following way:

```
command> booleanXOR
Select the surfaces of the 'objectA' on screen (Press enter when done): (Select one
 box)
```

```
Select the surfaces of the 'objectB' on screen (Press enter when done): (Select the
 other box)
Please Wait...
command>
```

Or, if the boxes have the names like "box1" and "box2", the command can be run in non-interactive mode:

```
command> booleanXOR -s -objectA box1 -objectB box2
Please Wait...
command>
```

The result of running this command is shown in the next figure:

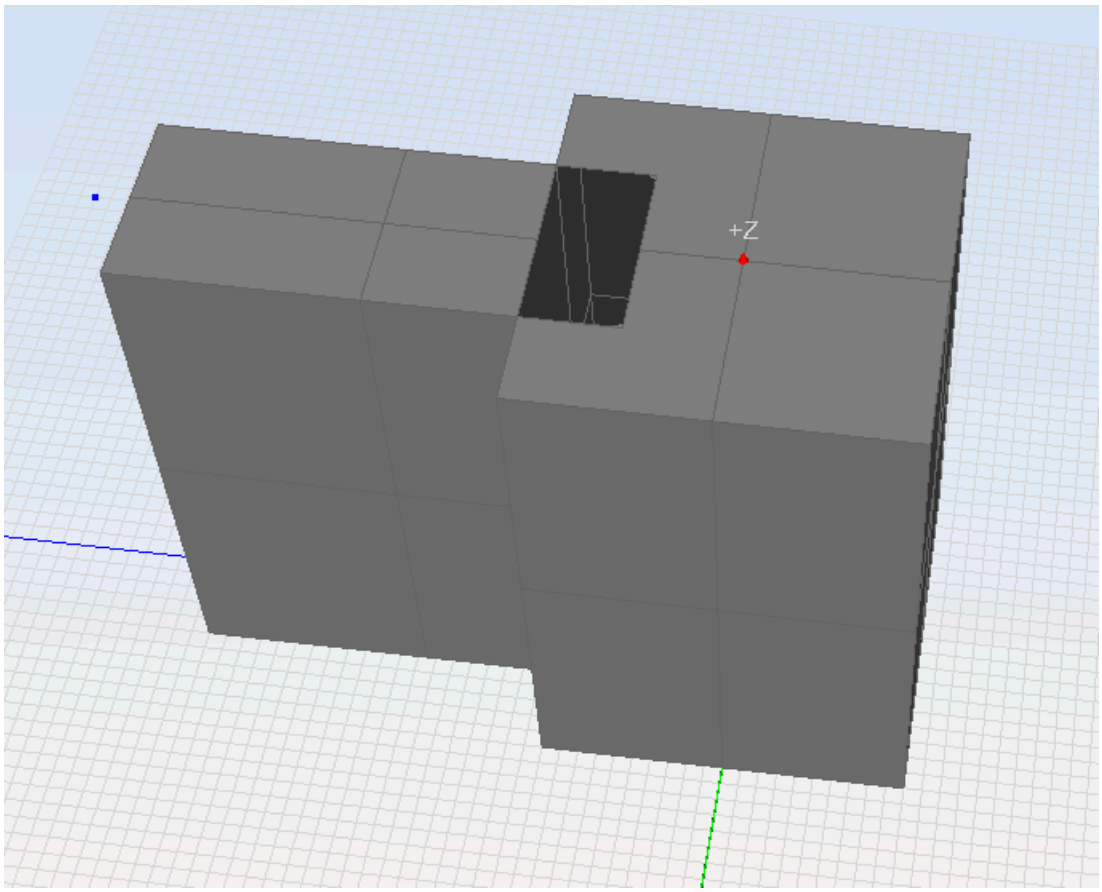


Figure 498: State of the geometry after the booleanXOR command

6.3.18 box

This command draws a box at the specified coordinates taking a square bounded by width, depth and height.

Inline mode usage

box -h
: Displays the help file that summarizes the parameters for this command.

box–n <name> -p <x> <y> <z> <width> <depth> <height>
: Draws a box with the specified parameters. The x, y and z parameters refer to the corner of the base of the box.

Interactive mode usage

Invocation
: box

Parameters

- First corner point of the base of the box: x y z.
- Base size: width depth.
- Height of the box: height.

The point of the first corner is specified by writing its three Cartesian components, while width and depth specify the dimension of the base.

Example

```
>box
```

```
First corner of base [x y z]: 0 -1 0
```

```
Base size [width depth]: 2 1
```

```
Height [double]: 3
```

Result

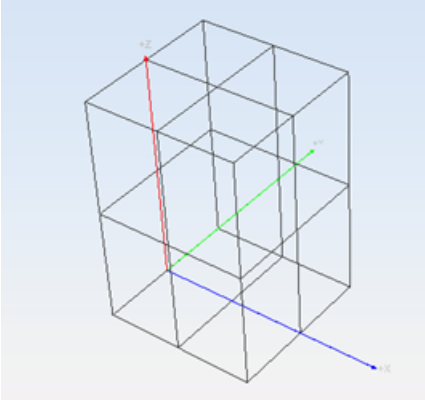


Figure 499: Resulting box

6.3.19 capacitance

Add a new capacitance to the geometry.

Inline mode usage:

- capacitance -h: Display a help message.
- capacitance -n name -p centerX centerY centerZ separation width height length: Create a new capacitance.

Screen usage:

The following steps will be needed in order to create a new capacitance:

1. Select center [x y z]: Central point of the capacitance (equidistant to both plates).
2. Separation between plates [separation]: Separation between the two plates.
3. Size of plate [width height]: Size of both plates.
4. Length [length]: Length of the outer segment.

Remark: All input values (besides the central point components) must be greater than zero.

Example

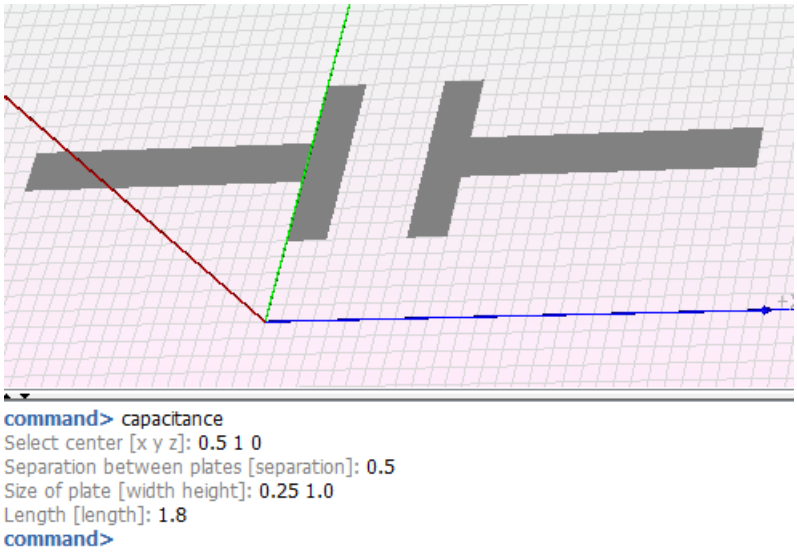


Figure 500: capacitance example

6.3.20 circle

This command draws a circle with a specified radius and center point.

Inline mode usage

circle -h
Displays the help file that summarizes the parameters for this command.

circle –n <name> -p <x_center> <y_center> <z_center> <radius>
Draws a circle named <name>with center coordinates (<x_center>, <y_center>, <z_center>) and specified radius <radius>.

Interactive mode usage:

Invocation
circle

Parameters

- Center point, in "x y z" format: x y z.
- Radius of the circle: radius.

Example:

```
>circle
```

```
Center [x y z] 0 1 0
```

```
Radius [double] 3
```

Result

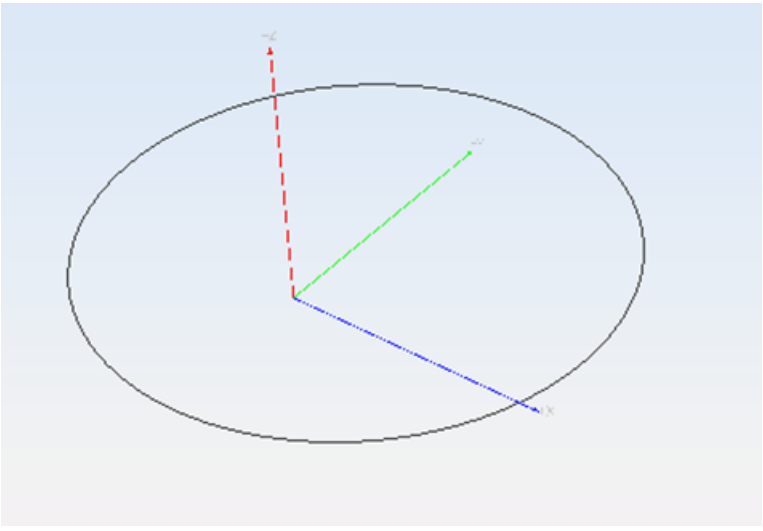


Figure 501: Resulting circle

6.3.21 circularHole

Add a new plane with a circular hole in it to the geometry.

Inline mode usage:

- circularHole -h: Display a help message.
- circularHole -n name -p cornerX cornerY cornerZ width depth radius: Create a new circular hole.

Interactive mode usage:

The following steps will be needed in order to create a new circular hole.

1. First corner of plane [x y z]: Corner point of the plane.
2. Plane size [width depth]: Size of the plane (width and depth).
3. Hole radius [radius]: Radius of the circular hole.

Example:

Figure 502: circularHole example

6.3.22 cone

This command draws a closed or open cone.

Inline mode usage:

cone -h
Displays the help file that summarizes the parameters for this command.

cone –n <name> -p <center_x> <center_y> <center_z> <radius_bottom> <height> <radius_top> <height>
Draws an open cone (no surfaces for the bottom and top bases) with the specified parameters.

cone –c –n <name> -p <center_x> <center_y> <center_z> <radius_bottom> <height> <radius_top> <height>
Draws a closed cone with the specified parameters.

Interactive mode usage:

Invocation
cone (only the shell (lateral surface) is drawn) or cone –c (caps of the cone as a solid model).

Parameters

- Center point of the base, specified in "x y z" format: x y z.
- Bottom radius: radius_bottom.
- Height of the cone: height.
- Top radius: radius_top.

Example:

```
> cone -c
```

```
Select center [x y z]: 0 1 0
```

```
Set bottom radius: 5
```

```
Set height: 4
```

```
Set top radius: 3
```

Result:

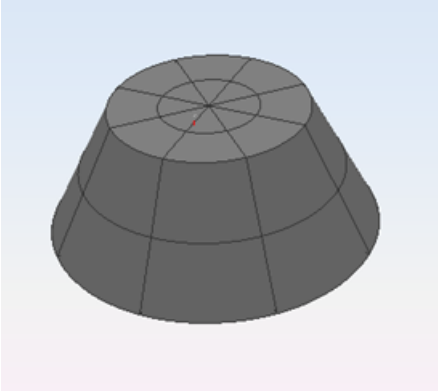


Figure 503: Resulting cone

6.3.23 connect2Surfaces

Generates a new surface to join the two selected ones (if they are not in contact).

Inline mode usage

connect2Surfaces -n name -p nameSurface1 nameSurface2 Connects the two surfaces named "nameSurface1" and "nameSurface2" by creating a new surface that joins them. The name of the joining surfaces will be "name".

Interactive mode usage:

Invocation

connect2Surfaces

Parameters

- -n name: name of the new generated surface
- -p name:Surface1 nameSurface2: name of the surfaces to connect

Example:

Suppose we have created two hemispherical surfaces, as shown in the next figure:

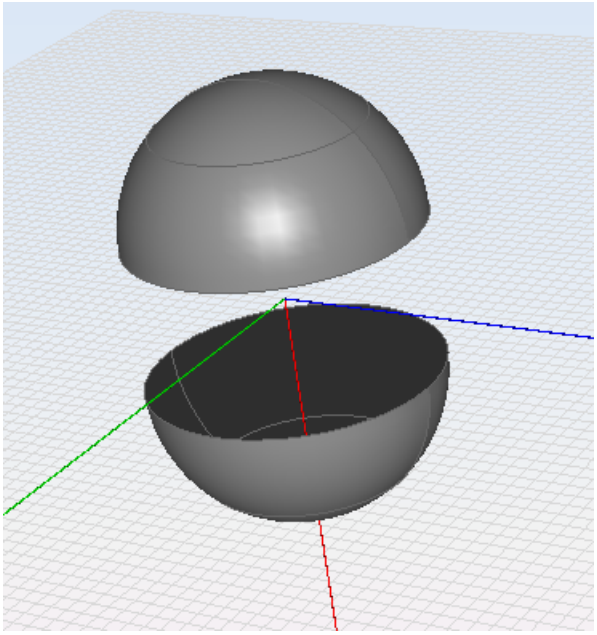


Figure 504: Two separated hemispheres

Let's also suppose we want to connect them to end up with a capsule-shaped object. To achieve this, the connect2Surfaces command can be used the following way:

```
command> connect2Surfaces
```

```
Select the first surface (Press enter when done or Esc to Cancel): (Select one of the
 hemispheres)
```

```
Select the second surface (Press enter when done or Esc to Cancel) (Select the other
 hemisphere)
```

```
Please Wait...
```

```
command>
```

Or even, if we are dealing with named surfaces, and the hemispheres are named "hemisphere1" and "hemisphere2":

```
command> connect2Surfaces
```

```
 -p hemisphere1 hemisphere2
```

```
Please Wait...
```

```
command>
```

The final result after the execution of this command is shown in the next figure:

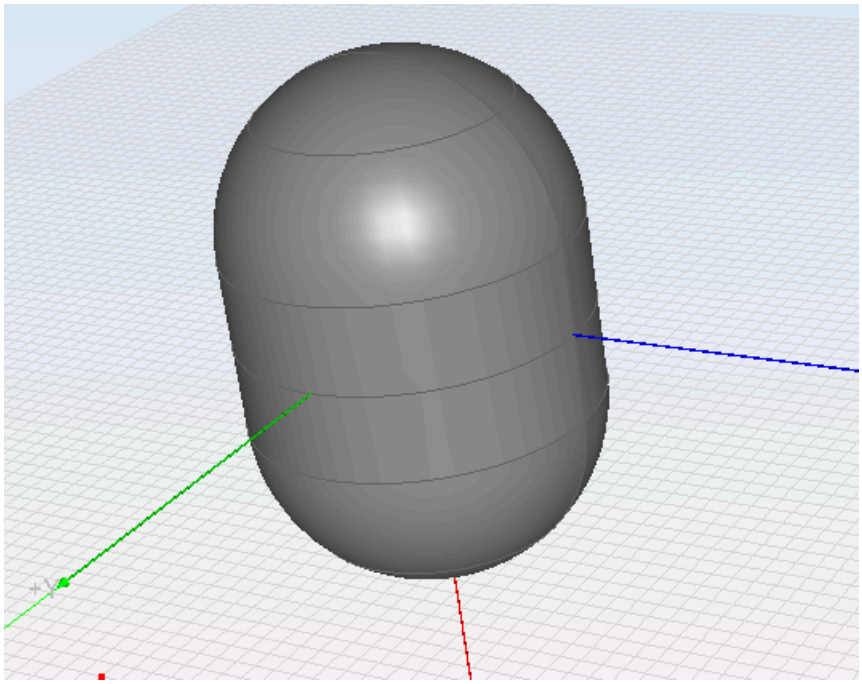


Figure 505: Result of the execution of the connect2Surfaces command

6.3.24 connectCurves

Connects the selected curves.

Inline mode usage

connectCurves -s all|name... -n resul_name

Interactive mode usage:

Invocation

connectCurves

Usage

Step 1: Select curves on screen (Press enter when done)

6.3.25 coons

Creates a coons patch through a selected loop of 2, 3 or 4 curves.

Inline mode usage

coons -h
: Displays the help file that summarizes the parameters for this command.

coons –s all
: Selects all curves of the scene.

coons –s <name>
: Selects <name> curve.

Interactive mode usage

Invocation
: coons

Parameters

- Two, three or four curves used for the creation of the coons patch. The set of selected curves must form a closed loop.

Example:

We are going to create a coons patch from a set of four curves, so the first step is to create the curves. For this, we will use the **`curveInterpolation`** command:

```
>curveInterpolation
Degree [3] 3
Select curve points [Press Enter when done]
Point 0 [x y z] 0 0 0
Point 1 [x y z] 1 0 2
Point 2 [x y z] 2 0 1
Point 3 [x y z] 4 0 0
Point 4 [x y z]:
>curveInterpolation
Degree [3] 3
Select curve points [Press Enter when done]
Point 0 [x y z] 4 0 0
Point 1 [x y z] 4 1 0.5
Point 2 [x y z] 4 3 1
Point 3 [x y z] 4 4 0
Point 4 [x y z]:
>curveInterpolation
Degree [3] 3
Select curve points [Press Enter when done]
Point 0 [x y z] 4 4 0
Point 1 [x y z] 3 4 0.5
Point 2 [x y z] 1 4 0
Point 3 [x y z] 0 4 2
```

Remember that you need to press **Enter** when all the points of a curve have been introduced. The result of these curves are shown in the following figure:

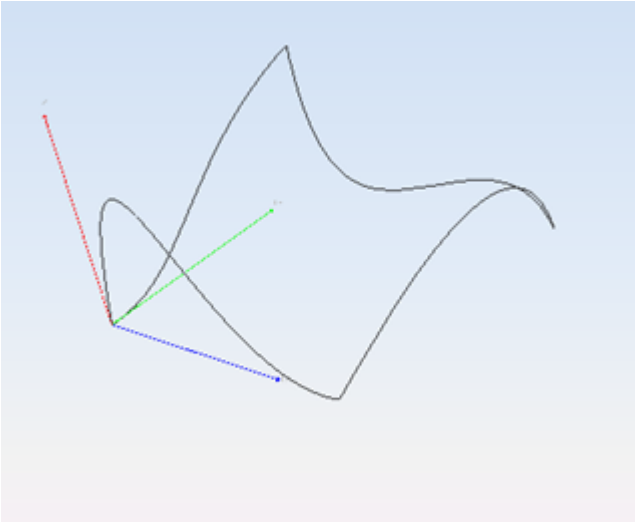


Figure 506: Curves used for the coons patch

Now we are going to create the coons surface. To do this, use the **coons** command:

```
>coons
```

```
Select a closed set of 2, 3 or 4 curves (press Enter when done)
```

```
>
```

The result is shown in the following figure:

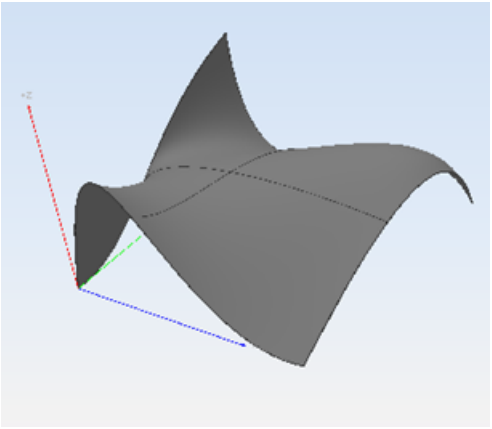


Figure 507: Resulting coons patch

6.3.26 copy

Copies an object or set of objects. The difference between this command and the array command is that the copy command copies objects by applying a displacement vector with an arbitrary direction.

Inline mode usage

copy -h
Displays the help file that summarizes the parameters for this command.

copy –s all –p <x1 y1 z1> <x2 y2 z2>
Copies all the objects from start point <x1 y1 z1> to end point <x2 y2 z2>.

copy –s <name> –p <x1 y1 z1> <x2 y2 z2>
Copies object with name <name> from start point <x1 y1 z1> to end point <x2 y2 z2>.

Interactive mode usage

Invocation

copy-all
Copy all objects.

Copy
Copy only the selected objects.

Parameters

- Objects to copy (only used when not specifying the -all parameter). These objects must be selected by left-clicking on them (hold Ctrl to select several objects) or selecting them on the tree view.
- Start point: x1 y1 z1.
- Copy point: x2 y2 z2.

The difference between the two points determines the displacement vector applied to the original object to create the copy.

Example

A 2x2x2 box was created prior to executing this command:

```
>copy
```

```
Select objects on screen (Press enter when done): // Select the box.
```

```
Select start point [x y z]: 0 0 0
```

```
Select copy point [x y z]: 2 2 0
```

Result

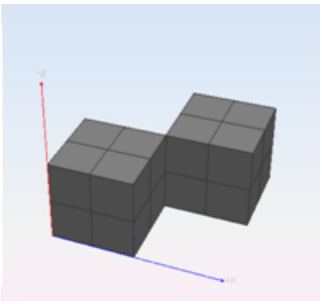


Figure 508: Result of applying the copy operation

6.3.27 cross

This command draws a cross-shaped surface.

Inline mode usage

cross -h
Displays the help file that summarizes the parameters for this command.

cross –n <name> -p <x> <y> <z> <sizeA> <sizeB>
Draws a cross named <name> and centered at the specified point <x> <y> <z>. <sizeA> determines the side length of the central square of the cross, while <sizeB> determines one of the sides (i.e. the one not shared with the central square) of each of the external rectangles.

Interactive mode usage

Invocation
cross

Parameters

- Center point, in "x y z" format: x y z.
- Size A, length: sizeA.
- Size B, width: sizeB.

The meanings of both sizes are specified above.

Example

```
> cross
```

```
Select center [x y z]: 0 0 0
```

```
Set Size A: 2
```

```
Set Size B: 3
```

Result:

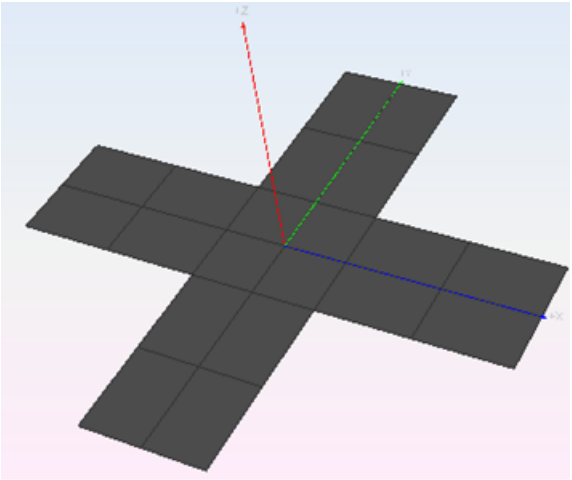


Figure 509: Resulting cross

6.3.28 crossHole

Add a new plane with a cross shaped hole in it.

Inline mode usage:

- crossHole -h: Display a help message.
- crossHole -n name -p cornerX cornerY cornerZ width height holeThickness holeSideLenght: Create a new cross hole.

Interactive mode usage:

The following steps will be needed in order to create a new cross hole:

1. First corner of plane [x y z] Corner point of the plane.
2. Width and height of plane [width height] Size of the plane (width and depth).
3. Thickness of cross-shaped hole [thickness] Thickness of the hole.
4. Length of sides of the cross-shaped hole [length] Length of the blades of the cross-shaped hole.

Example:

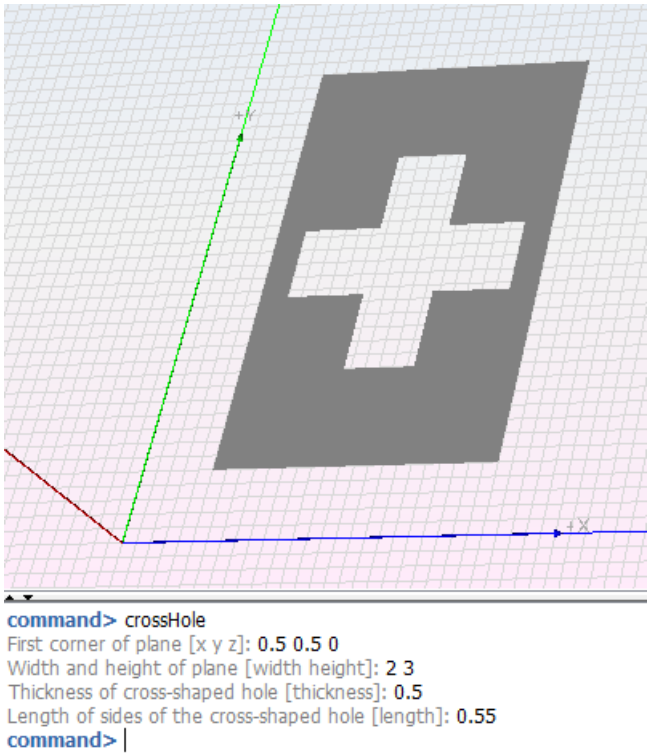


Figure 510: crossHole example

6.3.29 curve

This command creates a NURBS (Non-Uniform Rational B-Spline) curve by specifying its control points (as well as the weight of each one). Note that the number of points must be at least one more than the degree of the curve.

Inline mode usage

curve -h
Displays the help file that summarizes the parameters for this command.

curve –n <name> -d <degree> -p <x y z w>
Creates a NURBS curve named <name> with degree <degree> and specified points of control.

curve –n <name> -p <x y z w>
Creates a NURBS curve named <name> with degree 3 and specified points of control.

Interactive mode usage

Invocation
curve

Parameters

- The degree of the curve degree.
- The control points, in "x y z weight" format. The user has to press Enter after specifying each control point. A minimum of **degree + 1** points are required. To finish the insertion of points, press Enter.

Example

```
>curve
```

```
Degree [3]: 2
```

```
Select control points [Press Enter when done]
```

```
Point 0 [x y z w]: 0 0 0 1
```

```
Point 1 [x y z w]: 3 3 0 0.7
```

```
Point 2 [x y z w]: 6 3 0 1
```

```
Point 3 [x y z w]:
```

Result

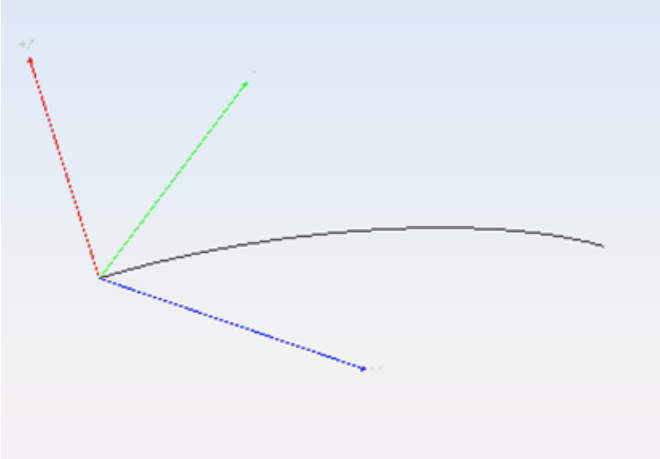


Figure 511: Resulting curve

6.3.30 curveFunction

This command draws a curve with three functions that define points in Cartesian coordinates.

Inline mode usage

curveFunction -h
Displays the help file that summarizes the parameters for this command.

curveFunction –n <name> -p <u_start> <u_end> <u_samples> <x=f($u)> <y=f($u)> <z=f($u)>
Creates a curve named <name> that starts in <u_start>, ends in <u_end> and has <u_samples>, defined by functions <x=f($u)>, <y=f($u)> and <z=f($u)>.

Interactive mode usage

Invocation
curveFunction

Parameters

- Parameter 'u', define range of variable 'u' start end samples.
- 'x' coordinate function: x=f($u).
- 'y' coordinate function: y=f($u).
- 'z' coordinate function: z=f($u).

Remark
The parameter is bound to the $u variable.

Example:

```
> curveFunction
```

```
Parameter '$u' (start end samples): 0 24 48
```

```
x = f($u) = $u
```

```
y = f($u) = sin($u)
```

```
z = f($u) = 0
```

Result:

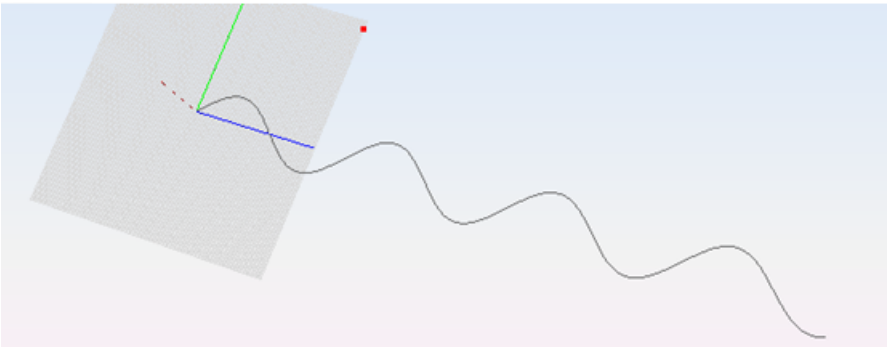


Figure 512: Resulting curve function

6.3.31 curveInterpolation

This option allows the user to draw a curve that goes through a set of points by applying polynomial interpolation.

Inline mode usage

curveInterpolation -h
Displays the help file that summarizes the parameters for this command.

curveInterpolation–n <name> -d <degree> -p <x y z>
Creates a curve named <name> with degree of the polynomial of <degree> and with points <x y z> used for the interpolation.

curveInterpolation–n <name> -p <x y z>
Creates a curve named <name> with degree of the polynomial of 3 and with points <x y z> used for the interpolation.

Interactive mode usage:

Invocation
curveInterpolation

Parameters

- Degree of the curve degree.
- The points used for defining the points of interpolation to form a curve. The number of points specified must minimally be (degree + 1) in order for the interpolation algorithm to work. To finish the insertion of points, press Enter.

Example:

```
> curveInterpolation
```

```
Degree [3]: 3 // (3+1) = 4 points required for interpolation to work.
```

```
Select curve points [Press Enter when done]
```

```
Point 0 [x y z]: 0 0 0
```

```
Point 1 [x y z]: 3 3 0
```

```
Point 2 [x y z]: 2 5 1
```

```
Point 3 [x y z]: 3 1 3
```

```
Point 4 [x y z]:
```

Result:

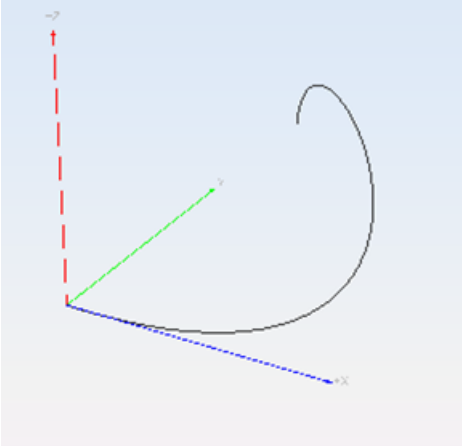


Figure 513: Resulting curve

6.3.32 cylinder

Creates a closed or open (i.e. without taps) cylinder.

Inline mode usage

- cylinder -h: Displays the help file that summarizes the parameters for this command.
- cylinder <center_x> <center_y> <center_z> <radius> <height>: Draws an open cylinder with the specified parameters.
- cylinder -c <center_x> <center_y> <center_z> <radius> <height>: Draws a closed cylinder with the specified parameters.

Interactive mode usage

Invocation

cylinder

Parameters

- Center point of the base, in "x y z" format.
- Radius of the circular section of the cylinder.
- Height of the cylinder.

Example

```
>cylinder -c
```

```
Select center [x y z] 0 2 0
```

```
Set radius 4
```

```
Set height 3
```

```
>
```

Result:

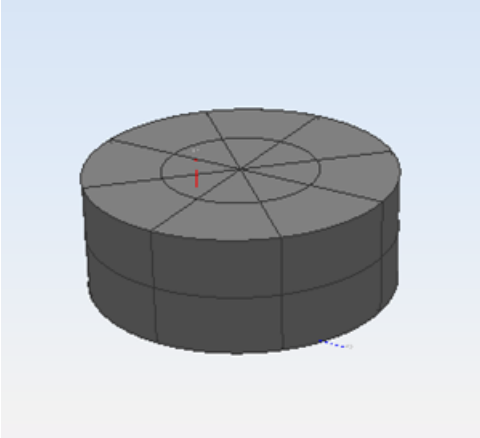


Figure 514: Resulting cylinder

6.3.33 delete

This command deletes an object or set of objects from the geometry. It is also possible to delete all the command history.

Inline mode usage

delete -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

delete -all
Delete all objects of the geometry and the command history.

delete
Delete the selected objects of the geometry.

Parameters

- Objects to delete (only when not specifying -h or all parameters). These objects must be selected by left-clicking on them (hold Ctrl to select several objects) or selecting them on the tree view.

When invoking "delete -all", the command will instead ask for confirmation to delete all objects of the geometry and command history. Note that this operation cannot be undone.

6.3.34 disk

This command creates a disk by specifying its center point and the radius.

Inline mode usage

disk -h
Displays the help file that summarizes the parameters for this command.

disk –n <name> -p <center_x> <center_y> <center_z> <radius>
Creates a disk named <name>, center at the specified point and with a given radius.

Interactive mode usage

Invocation
disk

Parameters

- Center point of the disk.
- Radius of the disk.

Example

```
>disk
```

```
Select center [x y z] 0 0 0
```

```
Set radius 4
```

```
>
```

Result

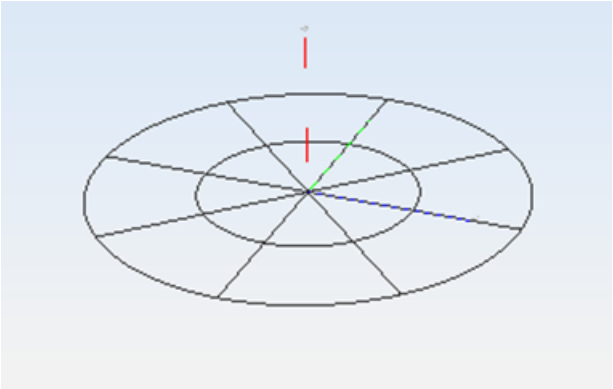


Figure 515: Resulting disk

6.3.35 distance

This option allows the user to calculate the Euclidean distance between two given points.

Inline mode usage:

distance -h
: Displays the help file that summarizes the parameters for this command.

distance <point1> <point2>
: Calculates and displays the distance between the two specified points. Each point is represented by its three components, separated by a blank space.

Interactive mode usage:

Invocation
: **distance**

Parameters

- First point
- Second point

Example:

```
> distance
```

```
Select first point [x y z] 3 0 0
```

```
Select second point [x y z] 0 4 0
```

```
The distance between points is 5.0
```

```
>
```

6.3.36 duplicate

Creates a copy of an existing object at the same position as the original one.

Inline mode usage:

- duplicate -h: Display a help message.
- duplicate -s all: Creates a copy of every object in the geometry.
- duplicate -s <objectName1> <objectName2> ... <objectNameN>: Creates a copy of the objects with names "<objectName1>", "<objectName2>", ..., "<objectNameN>".

Interactive mode usage:

If the command is called without any parameters, it will be run in interactive mode. In this mode, the user only needs to select the objects they want to duplicate and press Enter to perform the copy.

Example:

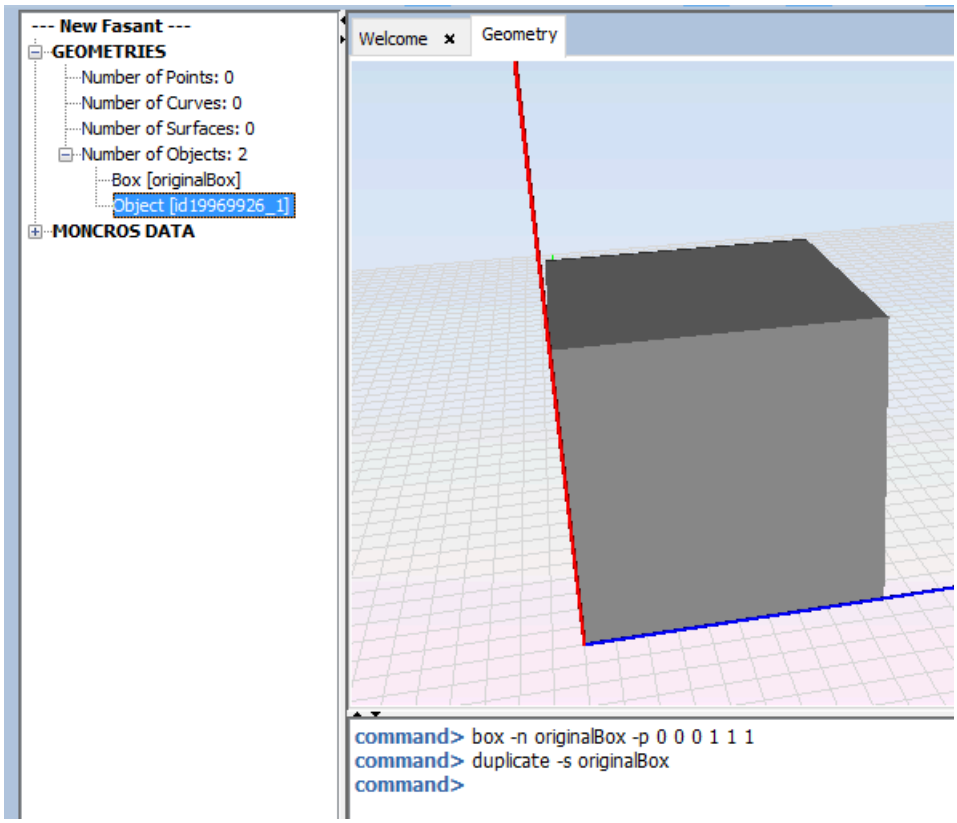


Figure 516: "duplicate" command example

The object with name "id19969926_1" is created by the "duplicate" command.

6.3.37 ellipse

This command draws an ellipse.

Inline mode usage

ellipse -h
Displays the help file that summarizes the parameters for this command.

ellipse –n <name> -p <center_x> <center_y> <center_z> <radius_x> <radius_y>
Draws an ellipse centered at the specified point and with the defined radii.

Interactive mode usage

Invocation
ellipse

Parameters

- Center point of the ellipse.
- Radius along the X axis.
- Radius along the Y axis.

Example:

```
>ellipse
```

```
Center [x y z]: 0 0 0
```

```
First radius (double): 1
```

```
Second radius (double): 5
```

```
>
```

Result:

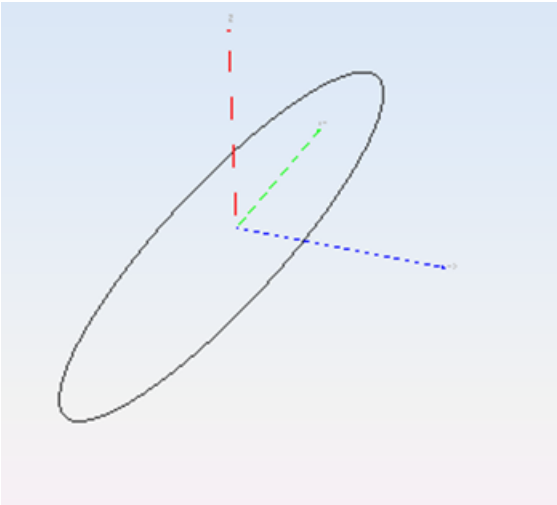


Figure 517: Resulting ellipse

6.3.38 ellipsoid

This command draws an ellipsoid in the geometry.

Inline mode usage

ellipsoid -h
Displays the help file that summarizes the parameters for this command.

ellipsoid –n <name> -p <center_x> <center_y> <center_z> <radius_x> <radius_yz>
Draws an ellipse centered at the given point, and with the specified radii.

Interactive mode usage

Invocation
ellipsoid

Parameters

- Center of the ellipse
- Radius along the Y and Z axes
- Radius along the X axis.

Example

```
>ellipsoid
```

```
Select center [x y z]: 0 0 0
```

```
Set major radius: 3
```

```
Set minor radius: 1
```

```
>
```

Result

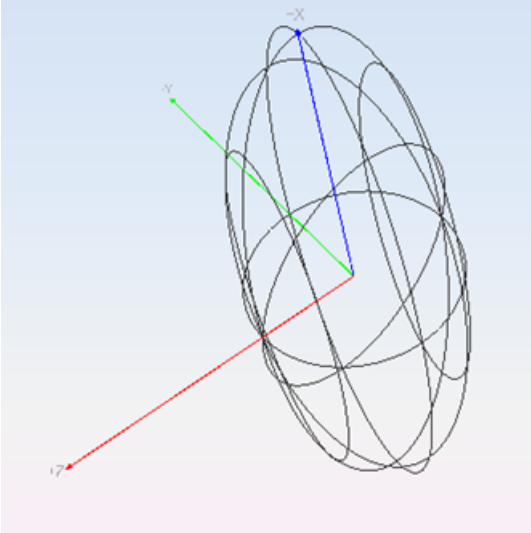


Figure 518: Resulting ellipsoid

6.3.39 exit

This command closes the application.

Inline mode usage

exit -h
: Displays the help file that summarizes the parameters for this command.

exit
: Closes the application.

When this command is issued, a dialog will require a confirmation from the user to exit newFASANT. If a project is already opened, it will give the possibility to save the project before exiting.

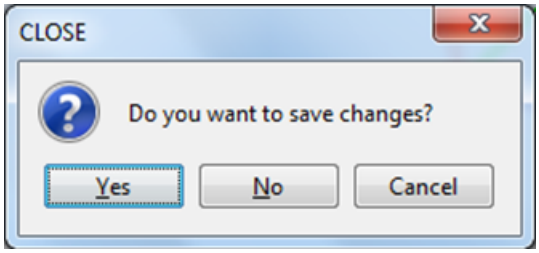


Figure 519: Dialog asking to save changes to the current project

6.3.40 explode

Divides an object into a specific number of surfaces.

Inline mode usage

explode -h
: Displays the help file that summarizes the parameters for this command.

explode -all
: Explode all geometries in the screen.

Interactive mode usage

Invocation
: explode or explode -all

Parameters

- Objects to explode (when not using the all option).

Example

A box was created prior to the following operation:

```
> explode
```

```
Select objects on screen (Press enter when done) // Select the box
```

```
>
```

Result

We can see the object representing the box was split into 6 different surfaces:

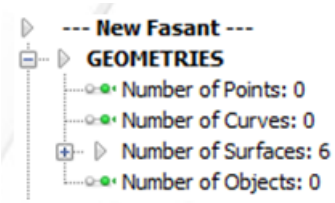


Figure 520: Resulting tree view after the explode operation

6.3.41 export

This command lets the user export the geometry contained in the current project into a file (NURBS, IGES, DXF, MSH, NAS…)

Inline mode usage:

export -h
Displays the help file that summarizes the parameters for this command.

export
Exports all selected objects. Search the geometry file with a file chooser window.

export <filepath>
Exports all objects into a file specified in the parameters.

export –s
Exports the selected objects. Search the geometry file with a file chooser window.

export –s <filepath>
Exports the selected objects into a file specified in the parameters.

Interactive mode usage

Invocation
export

Parameters
Path of the geometry file.

6.3.42 extendHole

This option allows the user to trim a surface with a loop and extrude the resulting cut off part of the surface.

Inline mode usage:

extendHole -h
Displays the help file that summarizes the parameters for this command.

extendHole –s <name> –c <curve> –p <height>
Extends a hole created by a surface <name> and a loop of curves <curve> with specified height.

Interactive mode usage:

Invocation
extendHole

Parameters

- The surface where the hole will be created.
- The curve used to create the hole.
- The length of the extrusion to be applied to the hole.

Example:

We are going to create a circular hole in a box, using the extendHole command. To do this, create the box and a circle using the following commands:

```
>box
```

```
First corner of base [x y z]: 0 0 0
```

```
Other corner of base [x y]: 2 2
```

```
Height: 2
```

```
>circle
```

```
Center [x y z]: 1 1 2
```

```
Radius [double]: 0.5
```

```
>
```

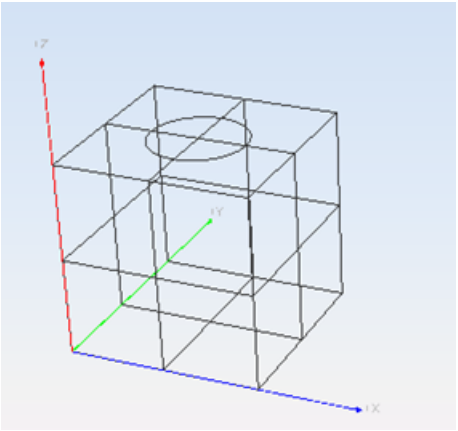


Figure 521: Created box and circle

Then we need to separate the box into different surfaces so we can work with only the top surface instead of the whole object. To do this, select the box and enter the explode command into the command line.

Next, we are going to invert the normal vectors of the curve. Initially, the normals of the circle are pointing counter-clockwise so the trim will cut off the part of the surface that is **outside** the circle. This is why we need to invert the normals, so select the circle and enter the invertNormals command into the command line.

Now, we can call the extendHole command the following way:

```
>extendHole
```

```
Select one surface to prolong, then press Enter. // Select the top face of the box
```

```
Select a closed set of curves, then press Enter. // Select the circle
```

```
Height [double]: -1
```

```
>
```

We pass a negative length to the command so we create a hole (if you need to extend the hole outwards, enter a positive length). The result is the following:

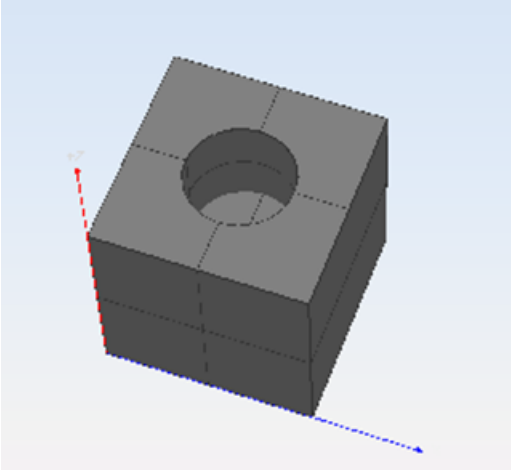


Figure 522: Result of the extendHole command

6.3.43 extrude

This command creates a surface by sweeping a curve along a translation vector, or by joining all of its points to a central point.

Inline mode usage

extrude -h
Displays the help file that summarizes the parameters for this command.

extrude –point –s all –p <x y z>
Extrudes all curves to a point.

extrude –point –s name –p <x y z>
Extrudes one curve <name> to a point.

extrude –s all –p <x1 y1 z1> <x2 y2 z2>
Extrudes all curves from a start point to an end point.

extrude –s name –p <x1 y1 z1> <x2 y2 z2>
Extrudes one curve <name> from a start point to an end point.

Interactive mode usage

Sweeping method:

Invocation
extrude

Parameters

- Curves to extrude (selecting them on the screen or in the tree view).
- Start point.
- End point.

The vector that joins the start point and the end point will be considered the translation vector used for the extruding.

Point method:

Invocation
extrude -p

Parameters

- Curves to extrude (selecting them on the screen or in the tree view).
- The coordinates of the extrusion point.

Example

```
A circle (centered at the origin, radius = 2) was created prior to the following
 operations.
```

```
> extrude
```

```
Select curves on screen (Press enter when done) // Select the circle
```

```
Start Point [x y z]: 0 0 0
```

```
End Point [x y z]: 0 0 3
```

```
>
```

Result

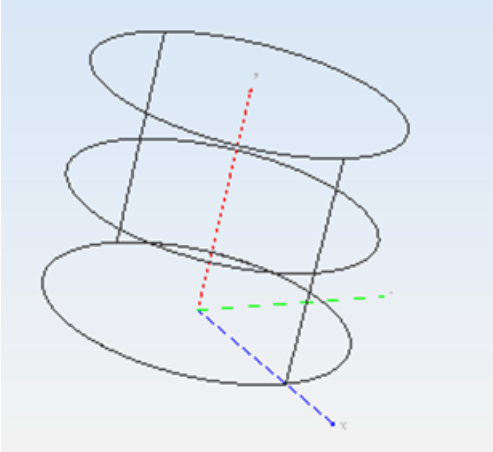


Figure 523: Result after extruding the circle

6.3.44 extrudeNormal

Creates a closed object by sweeping the chosen surfaces in the direction of their normal vectors. This command is a simplified version of the extrudeSurface command as the extrusion direction is set to the normal vector of the surfaces.

Inline mode usage:

- extrudeNormal -h: Display a help message.
- extrudeNormal -n <name> -s <surface1> <surface2> ... -closeMode <mode> -d <distance>: Creates a object by sweeping the surfaces with names <surface1>, <surface2>, the close mode <mode> and the given distance <distance> along their corresponding normal vectors.

Interactive mode usage:

If the command is called with no arguments, the steps required to properly execute this command in interactive mode are the following:

1. Select the surfaces to be extruded on screen (Press enter when done) It is possible to select more than one surface by holding the Ctrl key.
2. Insert the extrusion distance (Press enter when done) The distance the surfaces will be swept.
3. Select the close mode [1-None, 2-All, 3-Inner, 4-Outer, 5-Auto].

In the third parameter one of the following modes must be selected:

1. **None**: Only the parallel layer is returned.
2. **All**: Connects all edges of the surfaces. If the surfaces are connected it generates duplicate surfaces on the outer edges.
3. **Inner**: Only connects the inner edges (holes) of the surfaces.
4. **Outer**: Only connects the outer edges (bounds) of the surfaces.
5. **Auto**: Study the topology and connect the naked edges. It is not recommended to use on curved surfaces with long extrusion distances because the parallel layer may not be well connected.

Example:

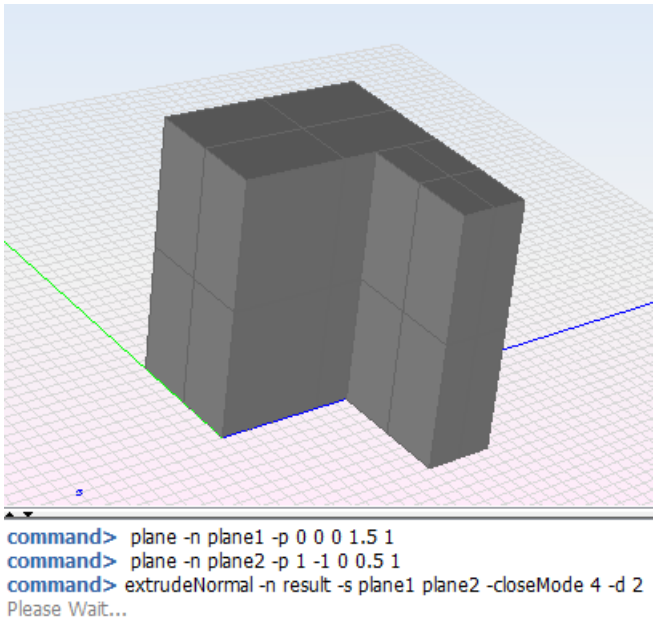


Figure 524: "extrudeNormal" command example

6.3.45 extrudeSurface

This option allows the user to create a solid volume from a surface. The original surface is not removed from the view area after the extrusion has been performed.

Creates a solid object by sweeping a surface along a translation vector, or by connecting all points on the perimeter of the surface to an extrusion point.

Inline mode usage:

extrudeSurface -h
Displays the help file that summarizes the parameters for this command.

extrudeSurface –point –s <name> –p <x y z>
Creates a solid object by connecting all points of the surface <name> to a single point.

extrudeSurface –s <name> –p <x1 y1 z1> <x2 y2 z2>
Creates a solid object by sweeping a surface along a direction vector.

Interactive mode usage:

Sweeping method:

Invocation
extrudeSurface

Parameters

- The surface or surfaces to extrude.
- Start point.
- End point.

The vector that joins the start point and the end point will be considered the vector used for the extrusion.

Point method:

Invocation
extrudeSurface -p

Parameters

- The surface or surfaces to extrude.
- The coordinates of the extrusion point.

Example:

```
A square surface was created prior to executing the following command.
```

```
>extrudeSurface
```

```
Select surface on screen (Press enter when done) // Select the surface
```

```
Start Point [x y z] 0 0 0
```

```
End Point [x y z] 0 0 3
```

```
>
```

Result:

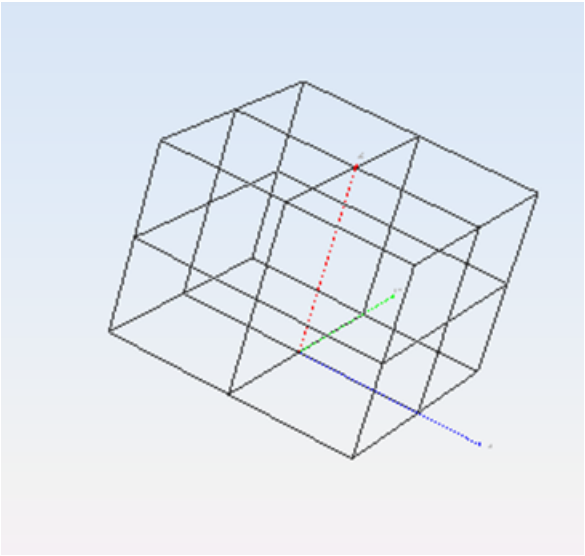


Figure 525: Result of extruding a square surface

6.3.46 flare

This command creates a trapezoidal prism.

Inline mode usage

flare -h
Displays the help file that summarizes the parameters for this command.

flare –n <name> -p <center_x> <center_y> <center_z> <bottomWidth> <bottomDepth> <height> <top_x> <top_y> <topWidth> <topDepth>
Draws a flare with the specified parameters with no caps (only the lateral surface).

flare –c –n <name> -p <center_x> <center_y> <center_z> <bottomWidth> <bottomDepth> <height> <top_x> <top_y> <topWidth> <topDepth>
Draws a flare with the specified parameters with caps.

Interactive mode usage

Invocation
flare or flare -c

Parameters

- First corner of bottom base.
- Second corner of bottom base.
- Height of the flare.
- First corner of top base.
- Second corner of top base.

The corners of both the bottom base and the top base must be two opposite ones (i.e. in the same diagonal).

Example

```
> flare
```

```
First corner of base [x y z] 0 0 0
```

```
Other corner of base [x y] 2 2
```

```
Set height 4
```

```
First corner of top [x y] 0.5 0.5
```

```
Other corner of top [x y] 1 1
```

```
>
```

Result

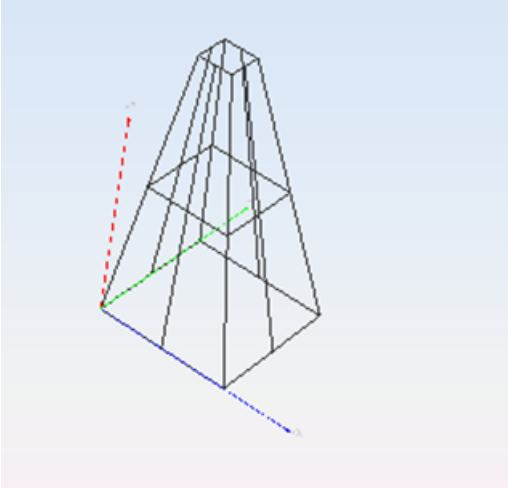


Figure 526: Resulting flare

6.3.47 group

This command groups the selected surfaces into an object.

Inline mode usage

group -h
Displays the help file that summarizes the parameters for this command.

group –s all
Groups all surfaces.

group –s <name>
Groups selected surfaces in <name>.

Interactive mode usage

Invocation
group or group -all

Parameters

- The surfaces to group into a single object (if the "all" flag is not specified).

6.3.48 help

This command displays the help file that summarizes the parameters for the console or the selected command.

Interactive mode usage: help [commandName | no_param]

- comandName displace the help that describes the command specified by **commandName**.
- no_param display the help that explain the most important tips for using the console with interactive commands.

Example

The next information is displayed when the **help -box** command is used:

```
Description: Adds new box in geometry panel taking a square bounded by width, depth
 and height
Screen usage: box [-h]
  -h: Print this help
  no_param: Follow these steps:
    Step 1: First corner of base [x y z]: x y z
    Step 2: Base size [width depth] width depth
    Step 3: Height [double] height
```

The next information is displayed when the **help** command is used:

```
Help Usage Type 'help <command>' or '<command> -h' to show help about a specific
 command.
Commands:
 - Type a command and press ENTER to execute it.
 - Press ESC to abort a command.
 - Press TAB to complete the written pattern name with the coincident commands.
 - Use the mouse to pick points or draw the geometry directly.
 - Press Ctrl+Left in a command step for coming back to the previous one.
History:
 - Use Up/Down arrows to show the commands used recently.
 - Use Ctrl+Z or Ctrl+Y for Undo/Redo.
 - Go to Edit->History to modify previous operations or delete specific commands.
Ctrl+L Clear the console panel.
```

6.3.49 hexagon

Adds a new hexagon to the geometry.

Inline mode usage:

- hexagon -h: print help guide
- hexagon -n name -p x y z radius: create hexagon

Interactive mode usage:

- Invocation hexagon
- Parameters:
- - Center [x y z]: point marked as center.
  - Radius [double]: hexagon radius.

Example:

```
> hexagonCenter [x y z]: 0.4 0.6 0.0
Radius [double]: 0.92
```

Result:

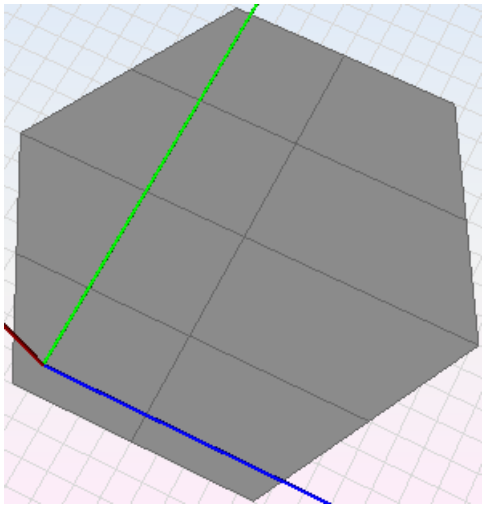


Figure 527: hexagon example

6.3.50 hyperbola

This command creates a hyperbolic curve on the geometry.

Inline mode usage

hyperbola -h
Displays the help file that summarizes the parameters for this command.

hyperbola
-n <name> -p <centerX> <centerY> <centerZ> <focusX> <focusY> <focusZ> <pX> <pY> <pZ>: Creates a hyperbola with the directrix center, focus and corner point.

Interactive mode usage

Invocation
hyperbola

Parameters

- Directrix center point [x y z].
- Focus point [x y z].
- Corner point [x y z].

Example

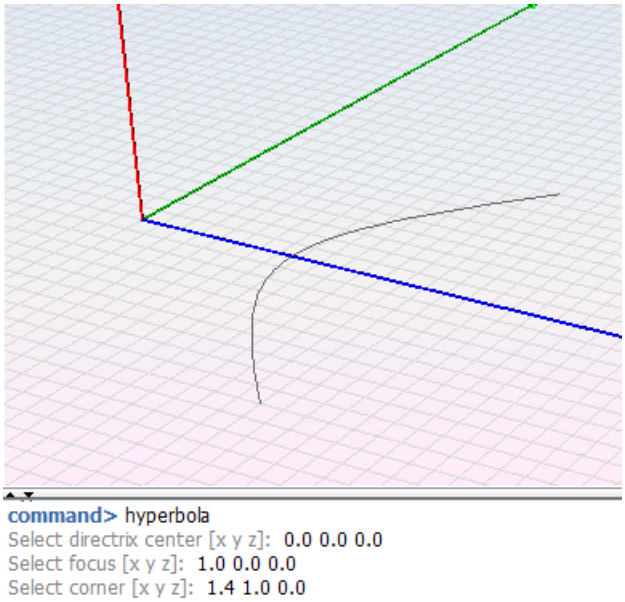


Figure 528: Hyperbola.

6.3.51 hyperboloid

This command draws a hyperboloid on the geometry.

Inline mode usage

hyperboloid -h
Displays the help file that summarizes the parameters for this command.

hyperboloid -n <name> -p <centerX> <centerY> <centerZ> <focus> <height> <radius> <holeRadius>
Draws a hyperboloid on the screen with the vertex point, focus distance, height of hyperboloid and radius. Hole radius is used to create a hole around the vertex.

Interactive mode usage

Invocation
hyperboloid

Parameters

- Center point: Describes the vertex of the hyperboloid.
- Focus distance: It is defined as the distance between the vertex and focus point.
- Radius: It is the distance from the vertex to the corner point in XY plane.
- Height: It is the distance from the vertex to the corner point in Z.
- Hole radius: It is the size of the hole in the center of hyperboloid.

Example

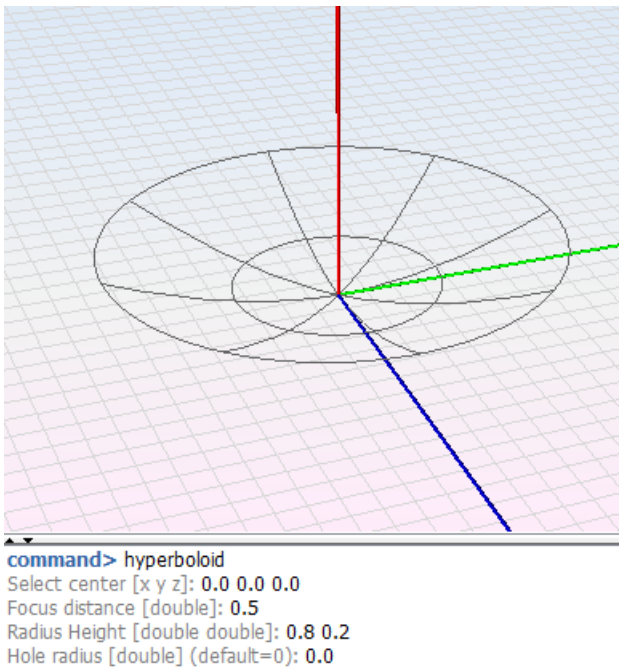


Figure 529: Hyperboloid.

6.3.52 import

Description:This option allows the user to import an external geometry in a file (NUR, IGES, DXF, MSH, NAS…)

Inline mode usage:

import -h
Displays the help file that summarizes the parameters for this command.

import
Import and search the geometry file with a file chooser window.

import <file_path>
Import from <file_path> directly.

Interactive mode usage:

Invocation
import

Parameters
Path of the geometry file.

6.3.53 interdigitalLines

Creates a stripline interdigital filter.

Inline mode usage:

- interdigitalLines -h: show help message
- interdigitalLines -n name -p x y z width height gap separation: create one geometry instance

Interactive mode usage:

- Center [x y z]: center for the interdigital filter.
- Gap [double]: separation between each one of the five parallel segments.
- Separation [double]: separation between each one of the two segment groups.
- Height [double]: height for the segments
- Width [double]: thickness for every segment in the filter.

Example:

```
> interdigitalLines
Center [x y z]: 0 0 0
Gap [double]: 1
Separation [double]: 0.4
Height [double]: 1.7
Width [double]: 0.44
```

Result:

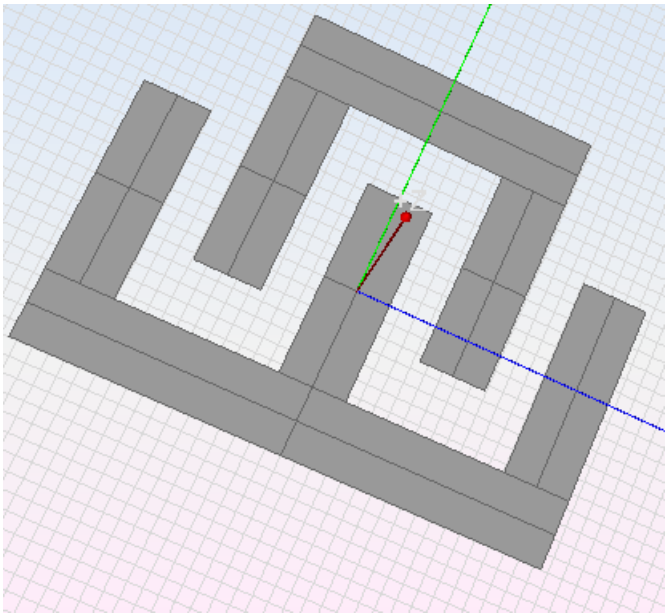


Figure 530: interdigitalLines example

6.3.54 invertNormals

This command inverts the direction of the normal vectors of a curve, surface or solid object.

Inline mode usage:

invertNormals -h
Displays the help file that summarizes the parameters for this command.

invertNormals -all
Invert the normal vector of all objects.

invertNormals
Invert normals by selecting objects on screen.

Interactive mode usage:

Invocation
invertNormals

Parameters

- The curve, surface or object whose normal surface vectors are to be inverted.

Example:

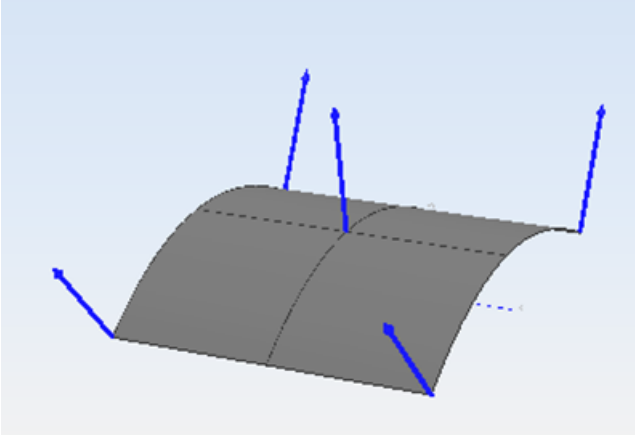


Figure 531: Visualizing normals

We’re going to invert the normal vectors of an existing surface. To help us visualize this, select the surface and select the *View Normals* option by clicking the button. The normal vectors are now visible on the geometry view:

Now we want to invert the normals, so they point downwards. To do this, use the invertNormals command:

```
>invertNormals
Select objects on screen (Press enter when done): // Select the surface and press
 Enter
>
```

Result:

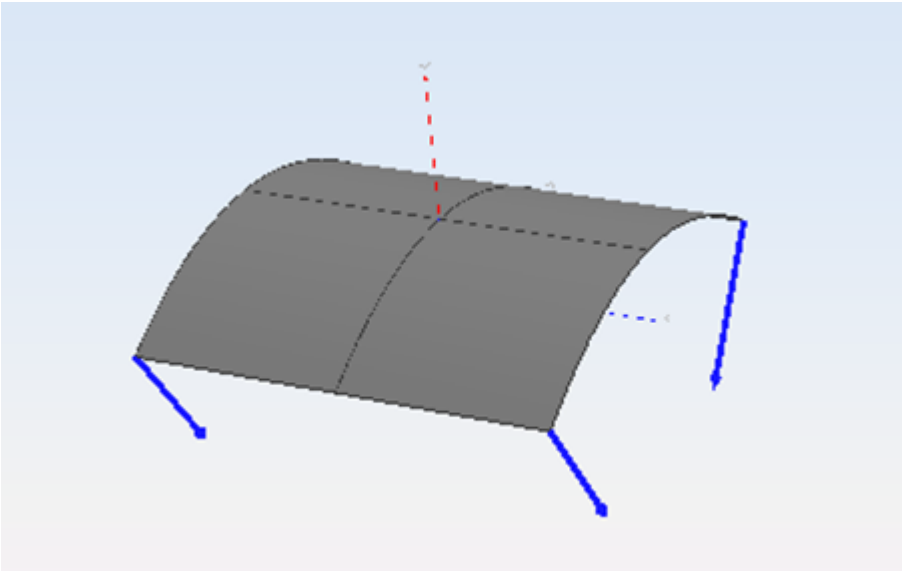


Figure 532: Result of inverting the normals

6.3.55 length

This command returns the total length of the selected curves.

Inline mode usage

length -h
Displays the help file that summarizes the parameters for this command.

length -all
The total length of the whole geometry is computed.

Interactive mode usage

Invocation
length

Parameters

- Selection of curves to be analyzed.

Example

```
command> spiral
```

```
Center [x y z] {default (0,0,0)}:
```

```
Radius [double] {default 1.0}:
```

```
Number of turns [double] {default 2.0}:
```

```
command> length
```

```
Select the curves to be analyzed (Press enter when done)
```

```
The length of the selected curves is 6.431392544068673 meters
```

```
command>
```

6.3.56 line

This command generates a straight line in the geometry.

Inline mode usage

line -h
Displays the help file that summarizes the parameters for this command.

line –n <name> -p <x1> <y1> <z1> <x2> <y2> <z2>
Draws a straight line between the two points specified by the user.

Interactive mode usage

Invocation
line

Parameters

- Initial point of the line.
- Final point of the line.

Example

```
>line
```

```
First point [x y z]: 0.5 1 0
```

```
Second point [x y z]: 1 0 2
```

```
>
```

Result

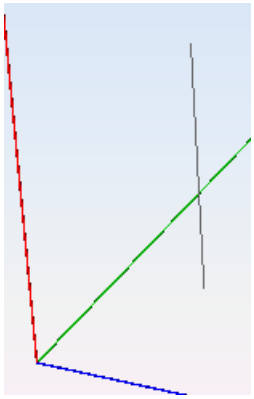


Figure 533: Resulting straight line

6.3.57 localTransform

This command allows the user to transform the selected objects, which are assumed to be positioned in the default Cartesian coordinate system, to the current coordinate system (determined by the reference plane).

Inline mode usage:

- localTransform -h: Display the help message that summarizes the parameters for this command.
- localTransform -all: Transforms all the objects in the geometry to the current reference plane.
- localTransform -s <objectName1> <objectName2> ... <objectNameN>: Transforms the objects with names <objectName1>, <objectName2>, ... and <objectNameN> to the current reference plane.

Interactive mode usage

Invocation

localTransform

Parameters

- Step 1: Select objects on screen that need to be transformed.

Interactive mode usage:

Example:

A typical use case for this command is the positioning of an antenna feed at the focal point of a single reflector or reflector system. In the following example, we will put an antenna feed (designed to be at the origin point of the absolute coordinate system) at the focal point of a Gregorian reflector system. Note that the following example can only be done in the MOM module.

Firstly, we will create the antenna feed. The chosen antenna feed for this example will be a pyramidal horn:

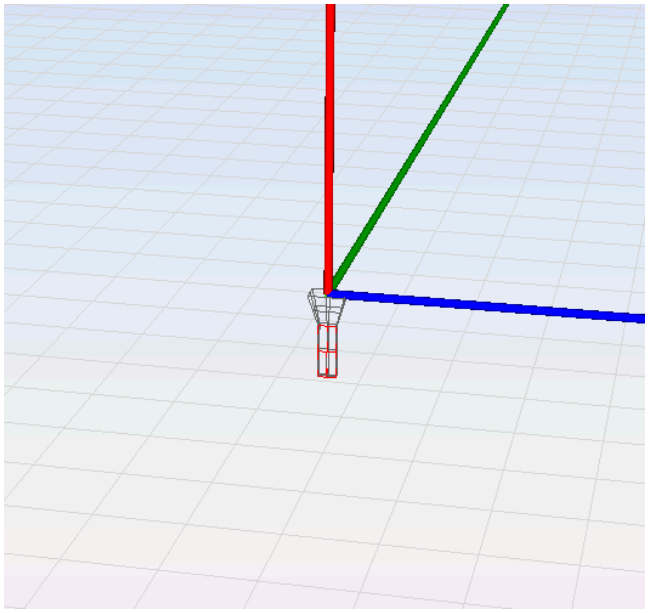


Figure 534: Created pyramidal horn

Next, we will create a offset-fed Gregorian reflector system. The feed point for this reflector system is chosen to be at (0.0, 0.0, 0.2):

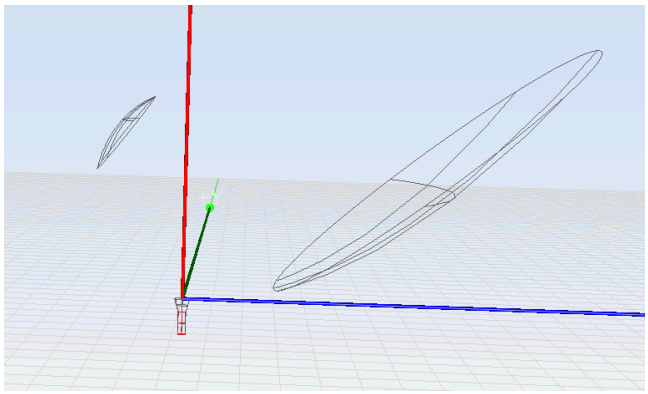


Figure 535: Created horn and Gregorian reflector system

Now, we will use the localTransform command in order to locate the pyramidal horn at the feed point of the Gregorian reflector system. However, prior to this operation we will need to set the reference plane so the origin point is at the feed point of the Gregorian system, and the positive Z axis is pointing to the center of the subreflector.

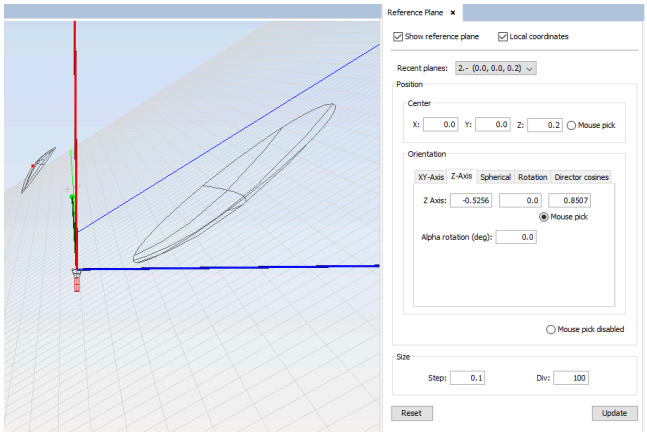


Figure 536: Reference plane for the localTransform operation

Once we have the reference plane (and therefore, our local coordinate system) configured adequately, it is possible to use the localTransform command to locate the horn at the feed point of the Gregorian system. We can do so in two different ways:

- Running the localTransform command in inline mode "localTransform -s <name of the horn object>".
- Running the localTransform command in interactive mode by writing "localTransform" in the command line, then selecting the horn object in the geometry panel and pressing Enter.

The result of the execution of this command is shown in the next figure:

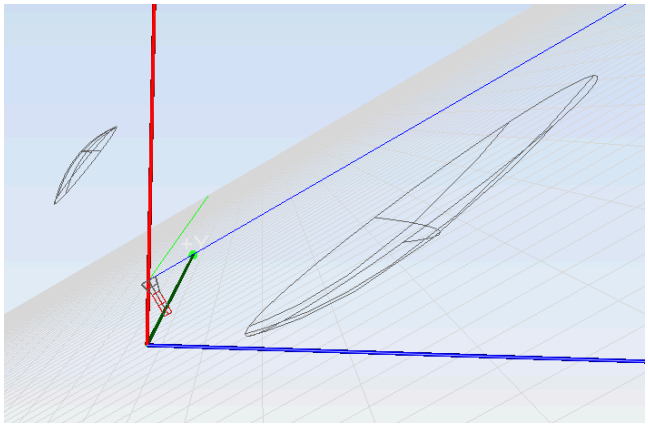


Figure 537: Result of the execution of the localTransform command on the horn

6.3.58 meshFromSurfaces

This command creates a mesh from a set of surfaces.

Inline mode usage:

- meshFromSurfaces -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

meshFromSurfaces

Parameters

- Surfaces to join to create a mesh.

6.3.59 meshToSurfaces

This command converts a mesh to a set of (open) surfaces.

Inline mode usage:

- meshToSurfaces -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

meshToSurfaces

Parameters

- Mesh to convert to surfaces.

6.3.60 move

This command moves an object by applying a translation vector to it.

Inline mode usage

move -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

move or move -all

Parameters

- The objects or objects to move (only when not specifying the all parameter). The objects must be selected by clicking on them in the geometry view (hold Ctrl to select several objects) or by selecting them on the tree view.
- Start point.
- End point.

The vector that joins the start point and the end point will be considered the translation vector considered for the move operation. For example, if the start point is (1,0,0) and the end point is (2,0,2), the object will be moved 1 unit in the X direction and 2 units in the Z direction.

Example

Assume a box was created prior to the move operation with the "box 0 0 0 2 2 2" command:

```
> move
```

```
Select objects on screen (Press enter when done): // Select the box and press Enter.
```

```
Start Point [x y z]: 1 1 1
```

```
End Point [x y z]: 3 3 1
```

Result

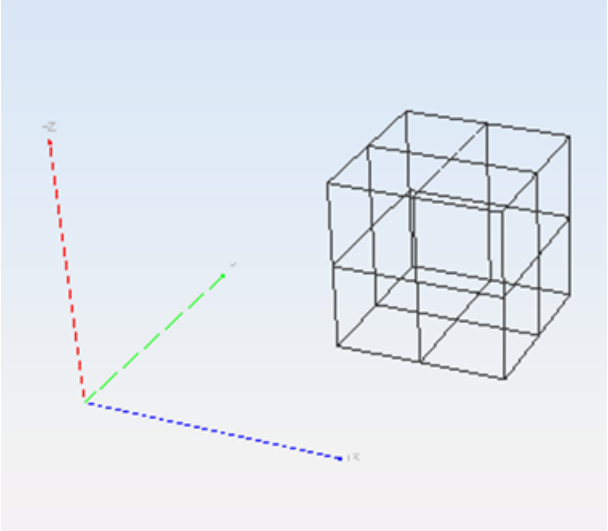


Figure 538: Result of the move operation

6.3.61 ogive

This command create a tangent ogive in the geometry.

Inline mode usage

- ogive-h: Displays the help file that summarizes the parameters for this command.
- ogive -n <name> -p <center_x> <center_y> <center_z> <radius_bottom> <height> <radius_top>: Creates a tangent-ogive named **name** whose base is centered at (**center_x**, **center_y**, **center_z**) and has a radius of **radius_bottom**, and the given **height** and **radius_top** at the peak. The tangent ogive is opened both at the base and peak.
- ogive -c -n <name> -p <center_x> <center_y> <center_z> <radius_bottom> <height> <radius_top>: Creates a spherically blunted-tangent ogive named **name** whose base is centered at (**center_x**, **center_y**, **center_z**) and has a radius of **radius_bottom**, and the given **height** and **radius_top** at the peak. The tangent ogive is closed both at the base and peak.

Interactive mode usage

Interactive mode usage

Invocation

ogive

Parameters

- -c :Creates a spherically blunted-tangent ogive that is closed both at the base and peak.
- no_param: Creates a tangent ogive that is open both at the base and peak.
- [-rightTurned | no_param]: The following parameters are required:
- - Step 1: Base center, given by its **x**, **y** and **z** coordinates.
  - Step 2: Radius at the bottom.
  - Step 3: Height of the tangent ogive.
  - Step 4: Radius at the top of the tangent ogive.

Examples

- A spherically blunted-tangent ogive is shown in next figure.

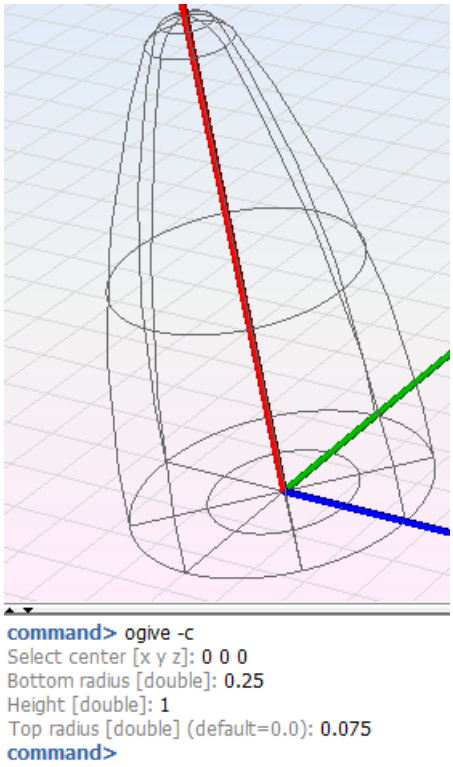
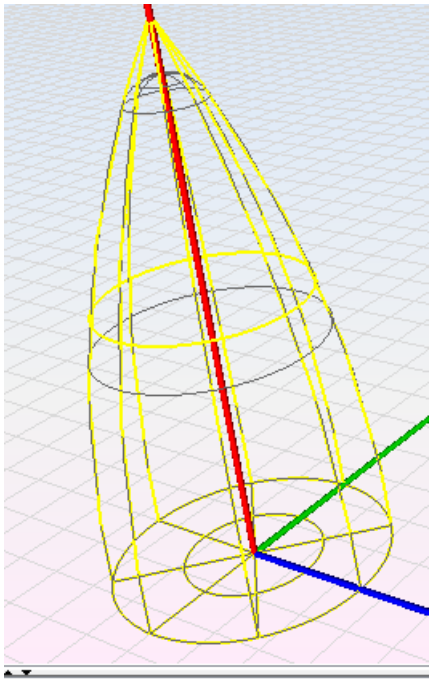


Figure 539: Spherically blunted-tangent ogive

- A tangent ogive with the same parameters than the spherically blunted is shown in figure below. Note that the **height** of the generated ogive only agrees with the specified one when the **top radius** is set to 0 (tangent ogive), otherwise the ogive is shorter (spherically blunted-tangent ogive).



```
command> ogive -c
Select center [x y z]: 0 0 0
Bottom radius [double]: 0.25
Height [double]: 1
Top radius [double] (default=0.0): 0.075
command> ogive -c
Select center [x y z]: 0 0 0
Bottom radius [double]: 0.25
Height [double]: 1
Top radius [double] (default=0.0):
command>
```

Figure 540: Tangent ogive compared to the spherically blunted-tangent ogive

6.3.62 openRing

Add a new open ring to the geometry.

Inline mode usage:

- openRing -h: Display a help message.
- openRing -n name -p centerX centerY centerZ minorRadius majorRadius openingAngle openingDirection: Creates a new open ring.

Interactive mode usage:

The following steps will be needed in order to create a new open ring:

1. Select center [x y z]: Center point of the ring.
2. Minor radius [double]: Inner radius.
3. Major radius [double]: Outer radius.
4. Opening angle [degrees]: Angle, in degrees, that determines how large is the opening.
5. Opening direction [degrees]: Angle, in degrees, that determines the direction the opening is facing in. An angle of zero degrees means that the opening is facing in the direction of the positive Y axis, and larger angles are interpreted as counter-clockwise rotations.

Remark The opening angle MUST be between 0 and 360.

Example:

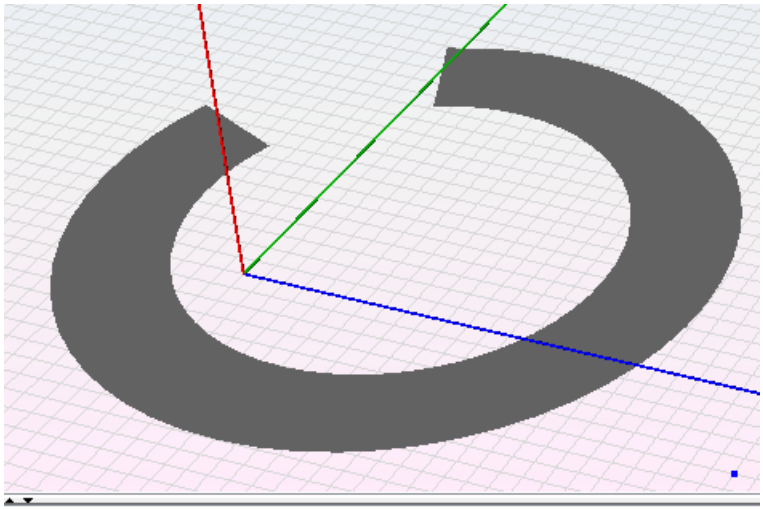


Figure 541: openRing example

6.3.63 parabola

This command draws a parabola.

Inline mode usage

parabola -h
Displays the help file that summarizes the parameters for this command.

parabola -n <name> -p <centerX> <centerY> <centerZ> <focusX> <focusY> <focusZ> <pX> <pY> <pZ>
Draws a parabola with the specified directrix center, focus and corner point.

Interactive mode usage:

Invocation
parabola

Parameters

- Directrix center: Center point of the directrix.
- Focus point: Parabola focus.
- Corner point: It is the end of the parabola.

Example:

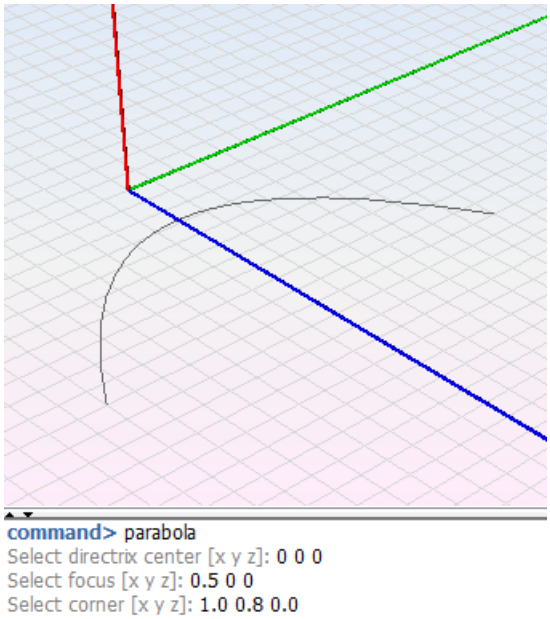


Figure 542: Parabola

6.3.64 paraboloid

This command creates a paraboloid in the geometry.

Inline mode usage

paraboloid -h
Displays the help file that summarizes the parameters for this command.

paraboloid -n <name> -p <centerX> <centerY> <centerZ> <focus> <radius> <holeRadius>
Draws a paraboloid on the screen with the vertex point, focus distance and radius. Hole radius is used to create a hole around the vertex.

Interactive mode usage

Invocation
paraboloid

Parameters

- Center point: Describes the vertex of the paraboloid.
- Focus distance: It is defined as the distance between the vertex and focus point.
- Radius: It is the distance from the vertex to the corner point in XY plane.
- Hole radius: It is the size of the hole in the center of paraboloid.

Example

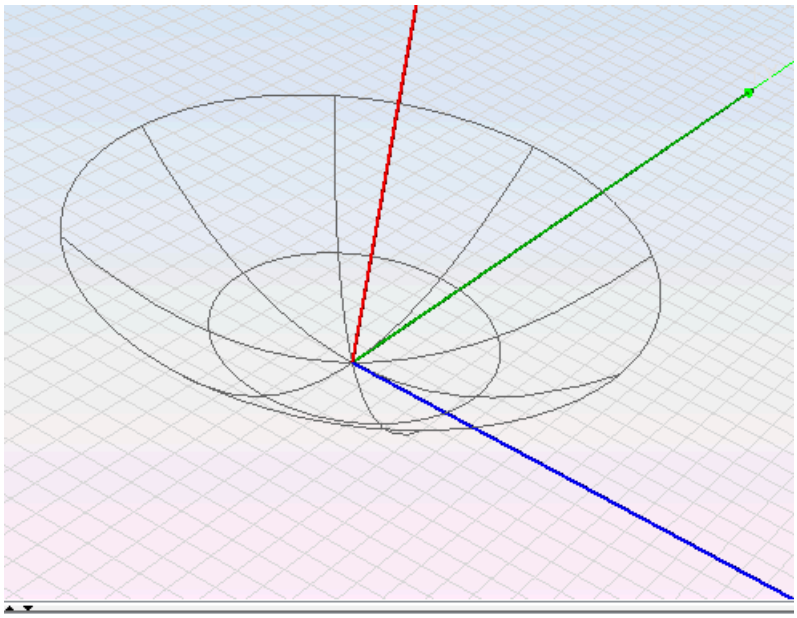


Figure 543: Paraboloid.

6.3.65 perpendicular

This command generates a straight line in the geometry perpendicular to the selected object that cross the introduced point.

Inline mode usage

perpendicular -h
Displays the help file that summarizes the parameters for this command.

perpendicular –n <resulName> -s <selectionName> -p <pX> <pY> <pZ> <length>
Generate a straight line normal to the selectionName curve/surface and crossing the point <pX> <pY> <pZ> with the specific length.

Interactive mode usage

Invocation
perpendicular

Parameters

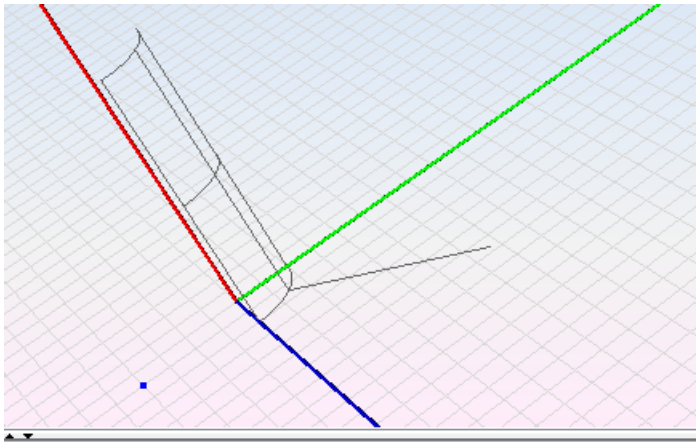
- Point in the perpendicular line.
- Length of the line.

Example

The following example shows how to insert a perpendicular line to a surface generated by revolving a line.

```
command> line
First point [x y z] 0.2 0 0
Second point [x y z] 0.2 0 1
command> revolve
Select curves on screen (Press enter when done)
Axis point 1 [x y z] 0 0 0
Axis point 2 [x y z] 0 0 1
Initial Angle [degrees] {default 0.0}
Final Angle [degrees] {default 360.0} 90
command> perpendicular
Insert the coordinates of a point in the perpendicular line [x y z] 0.5 0.5 0.0
Insert the length of the perpendicular line 0.75
command>
```

Result



```
command> line
First point [x y z]: 0.2 0 0
Second point [x y z]: 0.2 0 1
command> revolve
Select curves on screen (Press enter when done):
Axis point 1 [x y z]: 0 0 0
Axis point 2 [x y z]: 0 0 1
Initial Angle [degrees] {default 0.0}:
Final Angle [degrees] {default 360.0}: 90
command> perpendicular
Insert the coordinates of a point in the perpendicular line [x y z]: 0.5 0.5 0.0
Insert the length of the perpendicular line: 0.75
command>
```

Figure 544: Resulting perpendicular line

6.3.66 pipe

This command create a circular cross-section tube that surrounds the selected curve with the specified radius.

Inline mode usage

pipe -h
: Displays the help file that summarizes the parameters for this command.

pipe -s <nameCurve> -p <radius>
: Create the pipe along the given curve with the specified radius.

Interactive mode usage

Invocation
: pipe

Parameters

- Rail curve. Axis-curve where the pipe is centered in.
- Radius. Radius of the circular-cross section pipe. Corresponding distance from the rail curve to the closest point in the pipe.

Example

We're going to create a torus from a circle by using the pipe command.

```
command> circle
Center [x y z]: 0 0 0
Radius [double]: 1
command> pipe
Select one curve on screen (Press enter when done)
Radius [double]: 0.2
command>
```

Results

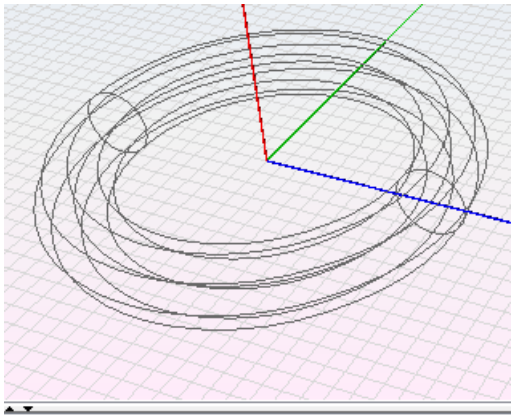


Figure 545: Pipe of a circle

6.3.67 planarSurface

This option allows the generation of planar surfaces from multiple coplanar lines.

Inline mode usage

planarSurface -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage

Invocation

planarSurface

Parameters

- Selected coplanar curves on the screen.

Example

First of all, a circle and a rectangle will be created in order to generate a planar surface from them.

```
> circle
```

```
Center [x y z] 0 0 0
```

```
Radius (double) 1
```

```
> rectangle
```

```
First corner of the rectangle [x y z] 0.5 0.5 0
```

```
Size [width depth] 1 1.5
```

```
>
```

Then, planar surface can be created.

```
> planarSurface
```

```
Select coplanar curves on the screen (press enter when done)
```

```
>
```

Result:

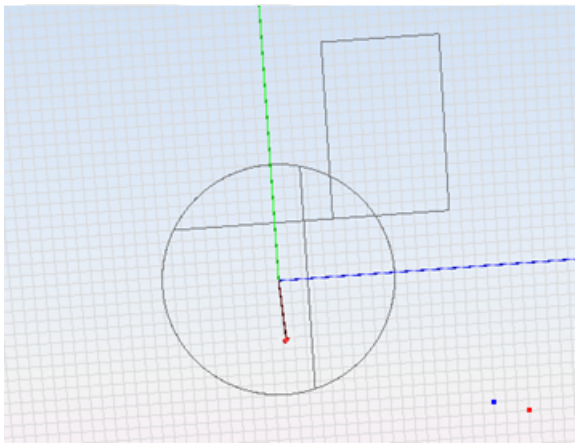


Figure 546: Resulting planar surface

6.3.68 plane

This command creates a plane in the geometry.

Inline mode usage

plane -h
Displays the help file that summarizes the parameters for this command.

plane –n <name> -p <point1_x> <point1_y> <point1_z> <width> <depth>
Creates a new plane determined by a start point, width and depth.

Interactive mode usage

Invocation
plane

Parameters

- First point.
- Width.
- Depth.

Example

```
> plane
```

```
Select first point [x y z] 0 0 0
```

```
Plane size [width depth] 2 2
```

```
>
```

Result

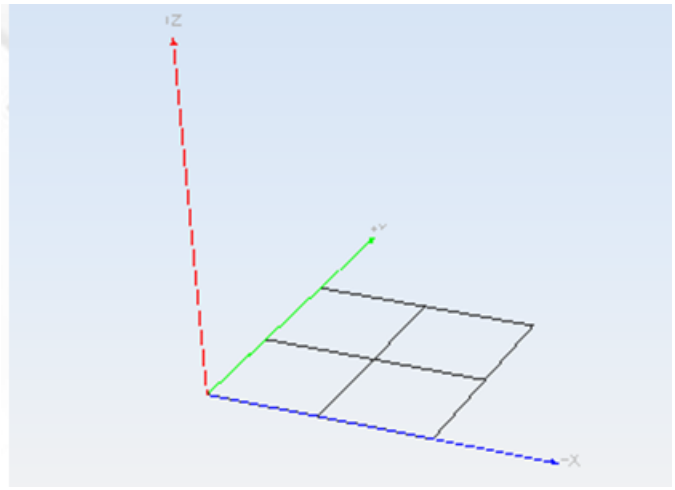


Figure 547: Resulting plane

6.3.69 point

This command defines a point in the geometry.

Inline mode usage

point -h
Displays the help file that summarizes the parameters for this command.

point –n <name> -p<x><y><z>
Draws a point at the specified coordinates.

Interactive mode usage

Invocation
point

Parameters

- Components of the point. Each component must be separated by a blank space. The point can also be selected by left-clicking points in the geometry view.

Example

In this example, we will create two points by using the two different interactive methods mentioned above.

```
> point
```

```
Select point on screen [x y z] 2 2 0
```

```
>
```

Result

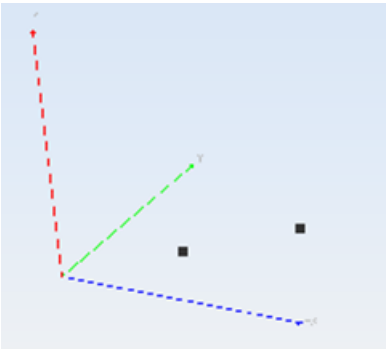


Figure 548: Resulting points

6.3.70 pointCloud

This command allows the user to create several points on the geometry.

Inline mode usage

pointCloud -h
Prints command usage.

pointCloud–n <name> -p <x y z>
Adds any arbitrary points given by <x y z> in geometry.

Interactive mode usage

Invocation
pointCloud

Parameters

- Points to draw. The points are specified by entering their coordinates in the command line or by selecting them on the geometry view. In both cases, the user will press Enter to confirm the selection. To finish entering points, press Enter without specifying an additional point.

Example

```
> pointCloud
```

```
Select point on screen [x y z]: 1 1 0
```

```
Select point on screen [x y z]: 1 1 1
```

```
Select point on screen [x y z]: 2 2 3
```

```
Select point on screen [x y z]:
```

```
>
```

Result

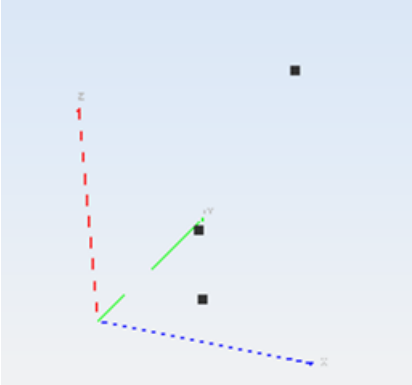


Figure 549: Resulting points

6.3.71 pointsFromCurve

The user can draw several points along a selected curve. The points are generated along a curve as a result of its division in several subcurves (the curve is not split) of identical size.

Inline mode usage

pointsFromCurve -h
Prints command usage.

pointsFromCurve –s <name_curve> -p <n_points>
Extract <n_points> points from the curve <name_curve>.

Interactive mode usage

Invocation
pointsFromCurve

Parameters

- Curve
- Number of points

Example

For this example, one curve must be defined.

```
> pointsFromCurve
```

```
Select one curve (Press enter when done):
```

```
Number of points [integer]: 4
```

```
>
```

Result

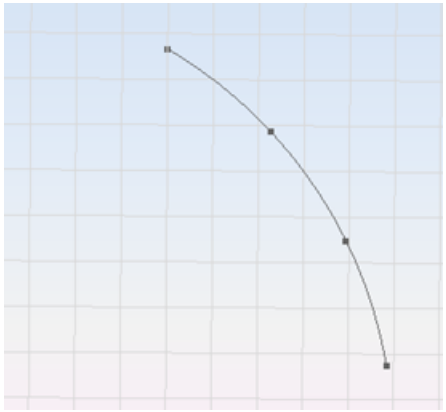


Figure 550: Extracted points from a curve

6.3.72 pointsFromCurveByDistance

The user can draw several points along a selected curve. The points are generated along a curve when the desired length of a subcurve (the input curve is not split) is reached.

Inline mode usage

pointsFromCurveByDistance -h
Prints command usage.

pointsFromCurveByDistance –s <name_curve> -p <distance>
Extract points from the curve <name_curve> separated <distance>.

Interactive mode usage

Invocation
pointsFromCurveByDistance

Parameters

- Curve
- Distance

Example

For this example, one curve must be defined.

```
> pointsFromCurveByDistance
```

```
Select one curve (Press enter when done):
```

```
Distance between points [double]:0.1
```

```
>
```

Result

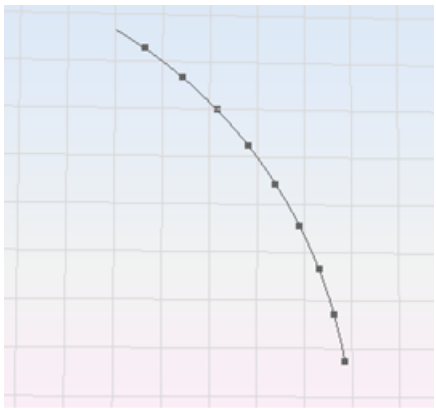


Figure 551: Extracted points from a curve by distance

6.3.73 pointsFromSurface

The user can draw several points along the two parametric dimensions of a selected surface.

Inline mode usage

pointsFromSurface -h
: Prints command usage.

pointsFromSurface –noTrim <name_surface> -p <n_points_U> <n_points_V>
: Extract points from along a surface <name_surface>. The user needs to introduce the maximum number of points along the U-dimension and the V-dimension of the surface, not considering trimmed points.

pointsFromSurface <name_surface> -p <n_points_U> <n_points_V>
: Extract points from along a surface <name_surface>. The user needs to introduce the maximum number of points along the U-dimension and the V-dimension in the valid region of the surface.

Interactive mode usage

Invocation
: pointsFromSurface

Parameters

- Surface
- Number of points along the U-dimension
- Number of points along the V-dimension

Example

For this example, one surface must be defined.

```
> pointsFromSurface
```

```
Select objects on screen (Press enter when done):
```

```
Number of points in U-dimension [integer] 20
```

```
Number of points in V-dimension [integer] 40
```

```
>
```

Result

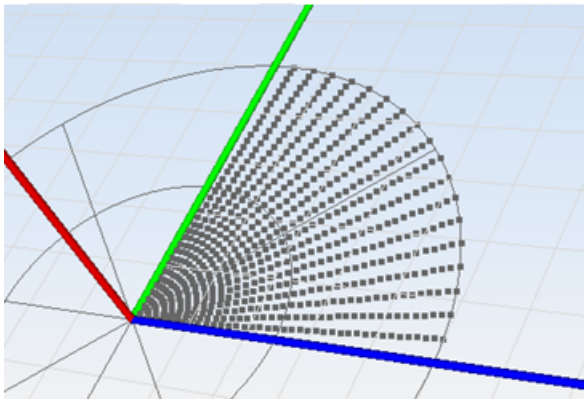


Figure 552: Extract points from a surface

6.3.74 polygon

This command draws a polygon in the geometry.

Inline mode usage

polygon -h
Displays the help file that summarizes the parameters for this command.

polygon –n <name> -p <x1> <y1> <z1> <x2> <y2> <x3> <y3> ...
Draws a N-gon with the specified vertices.

polygon –n <name> -r -p <sides> <x y z> <radius>
Draws a regular polygon with <sides> number of sides.

polygon –n <name> -s -p <sides> <x y z> <radius_1> <radius_2>
Draws a regular star with <sides> number of sides.

Interactive mode usage

Vertices method:

Invocation
polygon

Parameters

- The coordinates of each vertex of the polygon, in “x y z” format. When the desired points have been entered, the user needs to press Enter (without specifying any coordinates) to draw the polygon. A minimum of three vertices are required.

Regular polygon method:

Invocation
polygon -r

Parameters

- The number of sides of the polygon.
- The center point of the polygon.
- The radius of the circle that the polygon is circumscribed in.

Star method:

Invocation
polygon -s

Parameters:

- Number of spikes of the star.
- Center point of the star.
- Minor radius of the star.
- Major radius of the star.

Example

```
> polygon
```

```
Point 0 [x y z] 1 1 0
```

```
Point 1 [x y] 1 3
```

```
Point 2 [x y] 3 3
```

```
Point 3 [x y] 4 1
```

```
Point 4 [x y]:
```

```
>
```

Result

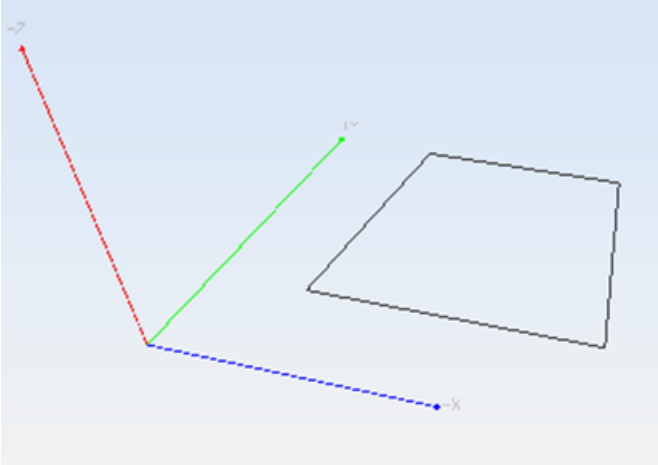


Figure 553: Resulting polygon

6.3.75 project

This command projects points and curves onto surfaces or objects. The original points and curves are not modified.

Inline mode usage

project -h
Displays the help file that summarizes the parameters for this command.

project -s <nameToProject1> <nameToProject2> ... –targets <nameSurface1> <nameSurface2> ...
Project the selected points and curves onto the closest points of the target surfaces selection.

project -s <nameToProject1> <nameToProject2> ... –targets <nameSurface1> <nameSurface2> ... -dir <startX> <startY> <start> <endX> <endY> <endZ>
Project the selected points and curves onto the target surfaces selection according to the vector defined by the differences <endX-startX> <endY-startY> <endZ-startZ>.

Interactive mode usage

Invocation
project or project -dir

Parameters

Objects to project
Selection of points and curves to be projected. They are not modified.

Target surfaces
Selection of surfaces and objects where the points and curves are projected.

Start point
Origin of the projection direction. This parameter is only required when the option -dir is included in the invocation.

End point
End of the projection direction. This parameter is only required when the option -dir is included in the invocation.

Example

We’re going to project a circle onto a sphere by using the two different modes.

The next commands create the objects required for the projection.

```
command> circle
```

```
Center [x y z]: 0 0 0
```

```
Radius [double]: 0.1
```

```
command> sphere
```

```
Select center [x y z]: 0 0 -1
```

```
Radius [double]: 2
```

```
command>
```

If no options is included in the invocation, the project to the closest point is done. This projection may modify the dimensions of the resulting curves.

If the option -dir is included in the invocation, two points are required to define the projection direction.

Results

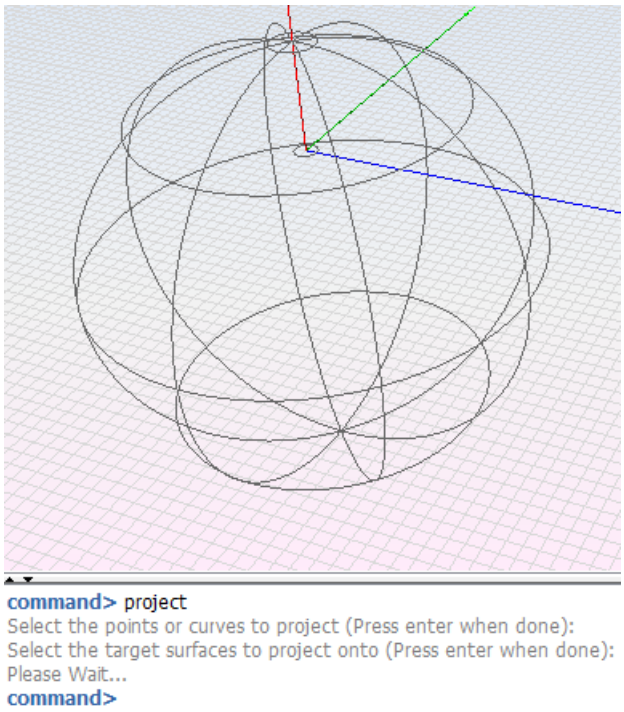


Figure 554: Project a circle onto a sphere

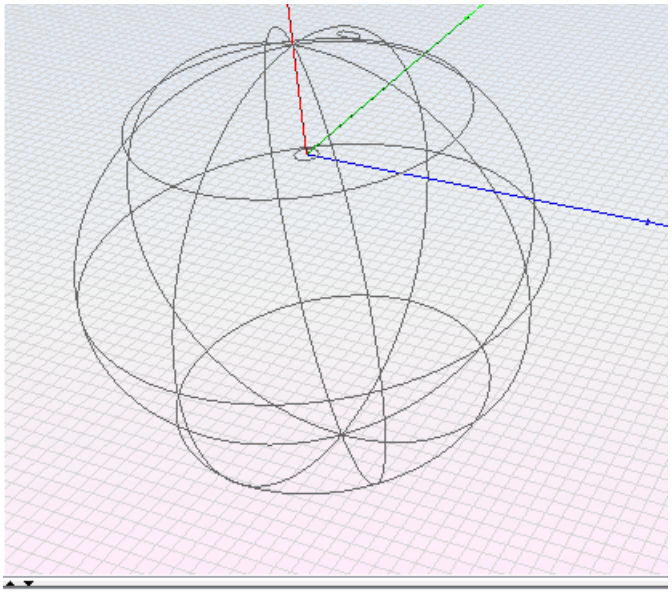


Figure 555: Project a circle onto a sphere by direction

6.3.76 rebuildCurve

This option allows rebuilding any given curve by sampling and interpolating the resulting points.

Inline mode usage:

rebuildCurve -h
Displays the help file that summarizes the parameters for this command.

rebuildCurve –s <curve> -p <degree> <points>
Rebuild the selected curve.

rebuildCurve –c –s <curve> -p <degree> <points>
Rebuild the selected curve, keeping the original surface.

Interactive mode usage:

Invocation
`rebuildCurve`

Parameters

- Selected curve.
- Degree of the new curve.
- Points of the new curve.

Example:

For this example one curve must be defined.

```
>rebuildCurve
```

```
Select one curve (Press enter when done):
```

```
Degree [integer](default=2): 3
```

```
Points [integer](default=3): 11
```

```
>
```

6.3.77 rebuildSurface

This option allows rebuilding any given surface by sampling and interpolating the resulting points.

Inline mode usage:

rebuildSurface -h
Displays the help file that summarizes the parameters for this command.

rebuildSurface –s <surface> -p <uDegree> <uPoints> <vDegree> <vPoints>
Rebuild the selected surface

rebuildSurface –c –s <surface> -p <uDegree> <uPoints> <vDegree> <vPoints>
Rebuild the selected surface, keeping the original surface.

rebuildSurface -noTrim –s<surface> -p <uDegree> <uPoints> <vDegree> <vPoints>
Rebuild the selected surface not trimmed.

rebuildSurface –c -noTrim –s <surface> -p <uDegree> <uPoints> <vDegree> <vPoints>
Rebuild the selected surface not trimmed, keeping the original surface.

Interactive mode usage:

Invocation
rebuildSurface

Parameters

- Selected surface.
- Degree along the U-dimension.
- Degree along the V-dimension.
- Points along the U-dimension.
- Points along the V-dimension.

Example:

For this example, one surface must be defined.

```
>rebuildSurface
```

```
U-Degree [integer](default=2): 1
```

```
V-Degree [integer](default=2): 1
```

```
Points in u direction [integer](default=3): 2
```

```
Points in v direction [integer](default=3): 2
```

```
>
```

Original surface:

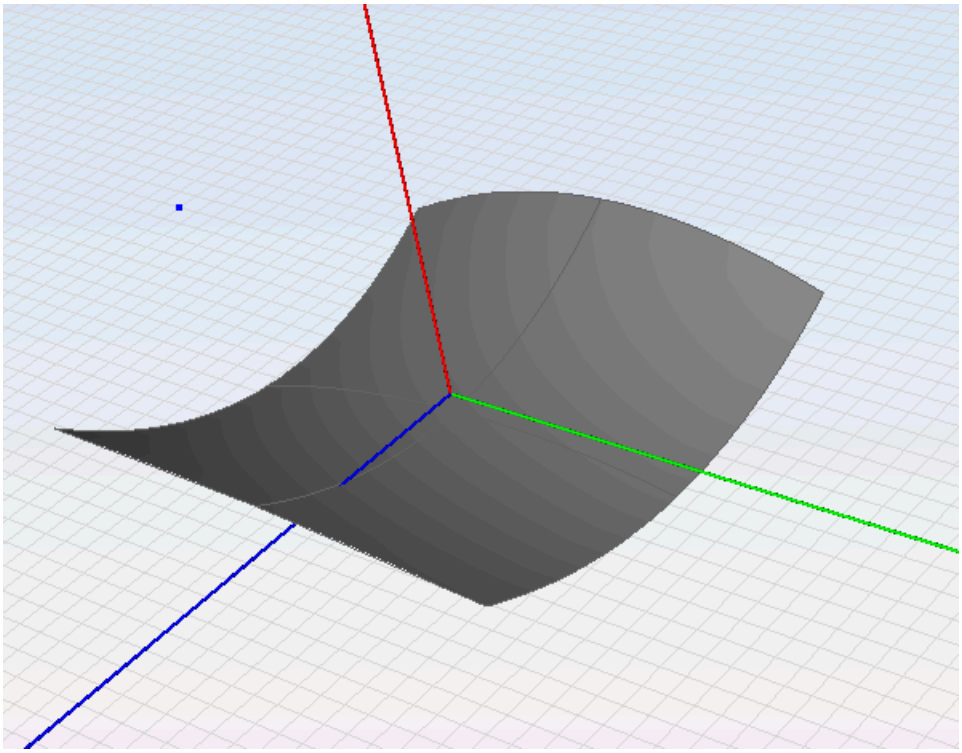


Figure 556: Original surface used for the rebuildSurface operation

Resulting surface:

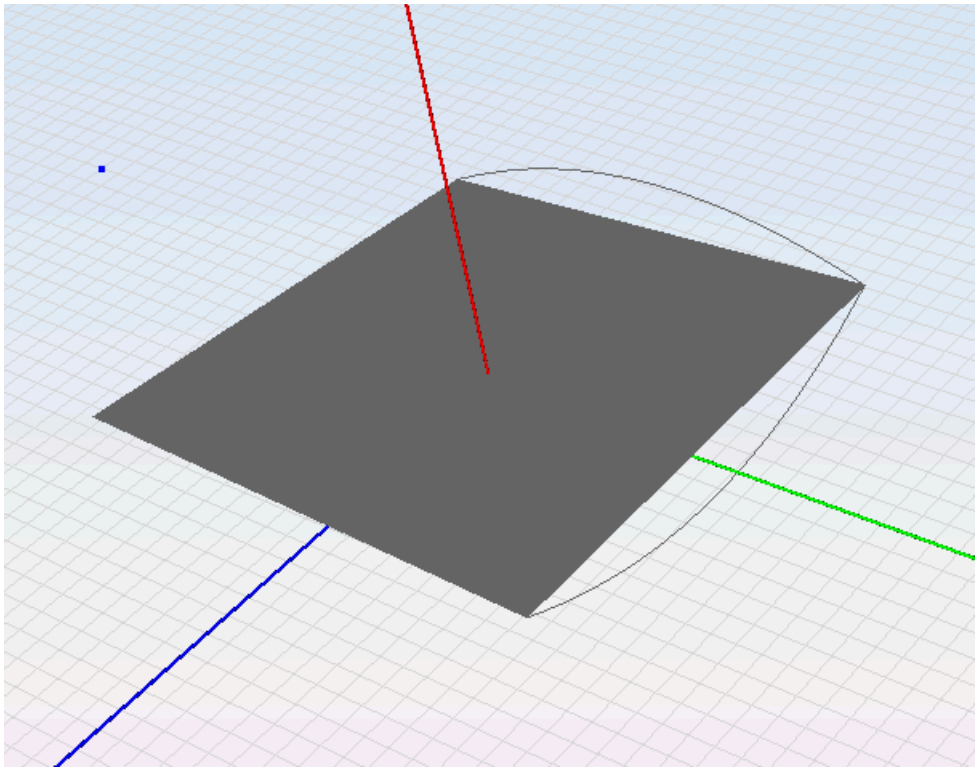


Figure 557: Resulting surface after the rebuildSurface operation

6.3.78 rectangle

This command draws a rectangle in the geometry.

Inline mode usage

rectangle -h
Displays the help file that summarizes the parameters for this command.

rectangle –n <name> -p <x1> <y1> <z1> <width> <depth>
Draws a rectangle with a start point on the corner of the geometry, width and depth.

Interactive mode usage:

Invocation
rectangle

Parameters

- Coordinate of first corner.
- Width
- Depth.

Example

```
> rectangle
```

```
First point [x y z] 0 0 0
```

```
Size [width depth] 2 4
```

```
>
```

Result

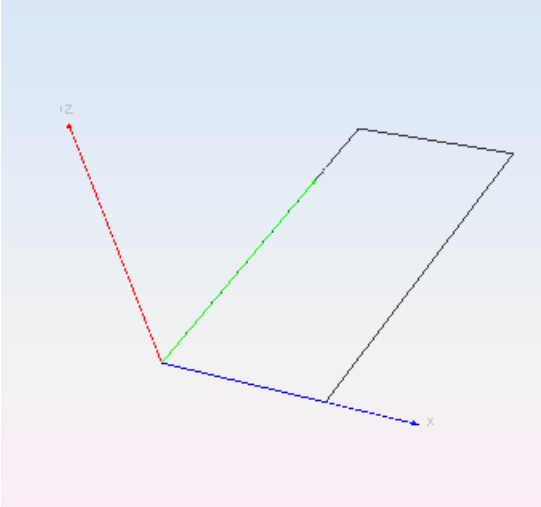


Figure 558: Resulting rectangle

6.3.79 rectangularHole

Adds new hole rectangle in geometry.

Inline mode usage:

- rectangularHole -h: shows the help guide for this command
- rectangularHole -n name -p x y z width height holeWidth holeHeight: creates a rectangular hole

Interactive mode usage:

- Invocation holeRectangle
- Parameters:
- - Select corner [x y z]: x y z - x, y, z corner for this holeRectangle
  - Major [width height]: width and height for the rectangle.
  - Minor [width height]: width and height for the inner hole.

Example:

```
> rectangular
```

```
HoleFirst corner of plane [x y z]: -1 -1 0
```

```
Size of plane [width height]: 2 2
```

```
Size of hole [width height]: 1 1
```

Result:

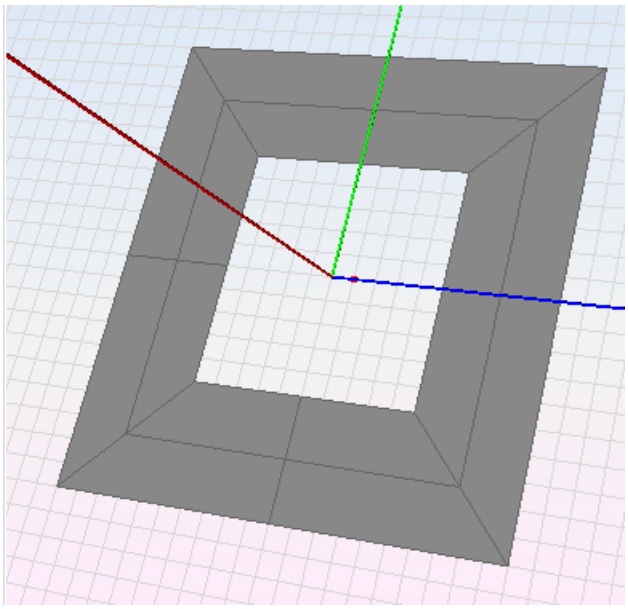


Figure 559: rectangularHole example

6.3.80 rename

This command renames an object to a given new name

Inline mode usage

rename -h: Displays the help file that summarizes the parameters for this command.

Interactive mode usage:

Invocation

rename <oldname> <newname>

Parameters

- Old name of the object.
- New name of the object.

6.3.81 revolve

This command creates a surface by revolving a curve around a rotational axis.

Inline mode usage

revolve -h
: Displays the help file that summarizes the parameters for this command.

revolve –s <name> –p <x1 y1 z1> <x2 y2 z2> <angle>
: Create a surface by rotating a curve by a number of degrees.

Interactive mode usage

Invocation
: revolve

Parameters

- Curves to revolve.
- First axis point.
- Second axis point.
- Rotation angle, in degrees.

The line that joins both axis points will be the axis considered for the rotation.

Example

We’re going to create a torus by revolving a circle around a rotational axis. In order to do this, we have to create the circle to revolve:

```
>circle
```

```
Center [x y z]: 6 0 0
```

```
Radius [double]: 2
```

```
>
```

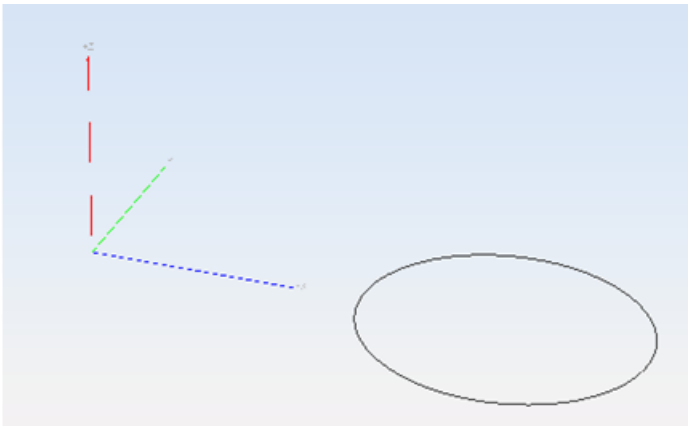


Figure 560: Base Circle

Revolving this circle around the Y axis we will create a torus with an inner radius of 4 and an outer radius of 8, as we will see below:

```
> revolve
```

```
Select curves on screen (Press enter when done): // Select the circle
```

```
Axis point 1 [x y z]: 0 0 0
```

```
Axis point 2 [x y z]: 0 1 0
```

```
Angle [degrees]: 360
```

```
>
```

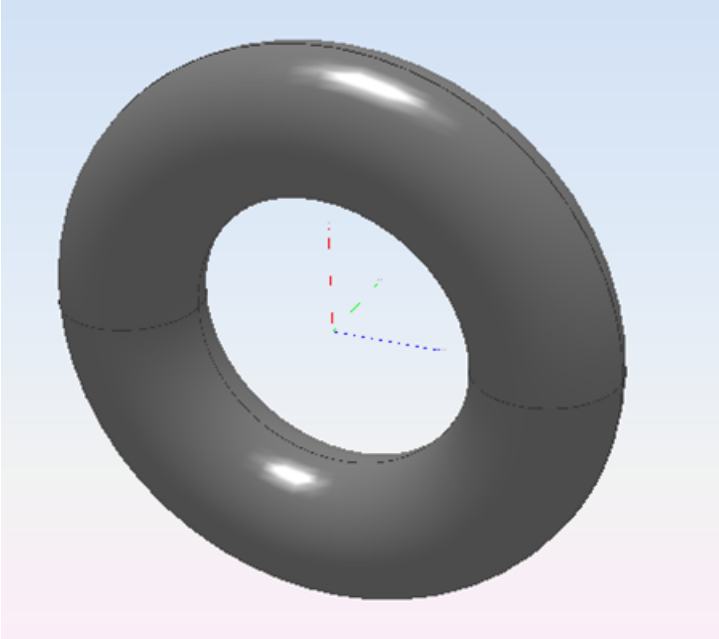


Figure 561: Resulting torus

6.3.82 rhomboid

This command draws a rhomboid in the geometry.

Inline mode usage

rhomboid -h
Displays the help file that summarizes the parameters for this command.

rhomboid –n <name> -p <x1> <y1> <z1> <x2> <y2> <x3> <y3>
Draws a rhomboid in the geometry given three of its corner points are defined by the user.

Interactive mode usage

Invocation
rhomboid

Parameters

- First point.
- Second point.
- Third point.

The fourth point of the rhomboid will be calculated by adding the difference between the second point and the first point to the third point.

Example

```
> rhomboid
```

```
First point [x y z]: 1 1 0
```

```
Second point [x y]: 3 2
```

```
Third point [x y]: 2 5
```

```
>
```

Result

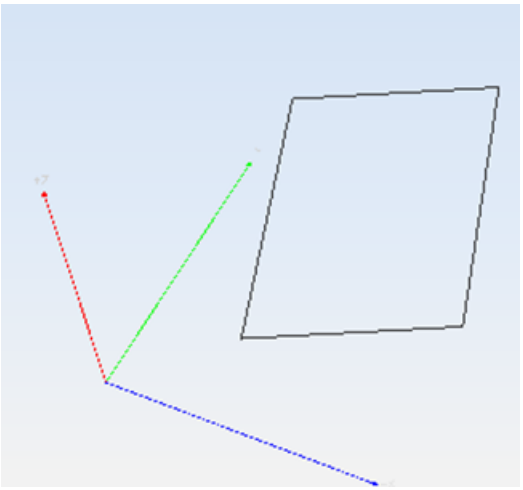


Figure 562: Resulting rhomboid

6.3.83 ring

Adds new ring in geometry panel.

Inline mode usage:

- ring -h: Display a help message.
- ring -n name -p x y z minorRadius majorRadius: Generates a new ring surface.

Interactive mode usage:

The following steps will be needed in order to create a new ring:

1. First corner of base [x y z]: Center point of the ring.
2. Minor radius [double]: Inner radius of the ring.
3. Major radius [double]: Outer radius of the ring.

Example:

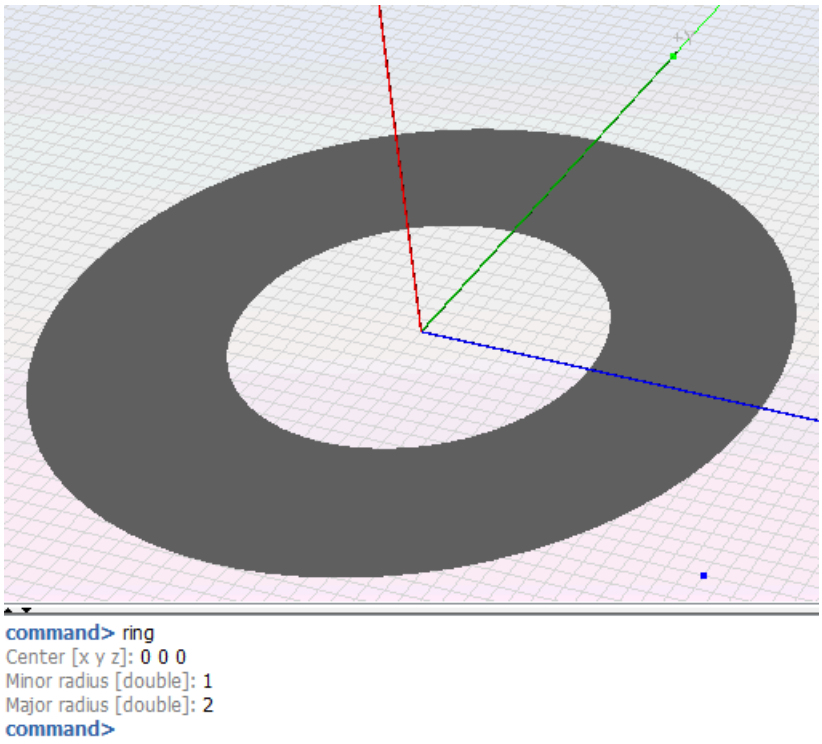


Figure 563: ring example

6.3.84 rotate

This command rotates an object a given number of degrees around a specified axis.

Inline mode usage

rotate -h
Displays the help file that summarizes the parameters for this command.

rotate –all –p <x1 y1 z1> <x2 y2 z2> <angle>
Rotates all objects from start point <x1 y1 z1> to final point <x2 y2 z2> with angle <angle>.

Interactive mode usage

Invocation
rotate or rotate -all

Parameters

- Objects to rotate (when not specifying the all modifier).
- First point of rotational axis.
- Second point of rotational axis.
- Angle to rotate, in degrees.

Example

We want to create a circle lying on the XZ plane. To do this, we create a circle using the circle command. This circle will lie in the XY plane:

```
>circle
```

```
Center [x y z] 0 0 0
```

```
Radius [double] 2
```

```
>
```

Then, we will rotate this circle by 90 degrees around the Z axis. For that operation, we will use the rotate command:

```
> rotate
```

```
Select objects on screen (Press enter when done) // Select the circle
```

```
Select first point [x y z] 0 0 0
```

```
Select second point [x y z] 1 0 0
```

```
Select angle (double) 90
```

```
>
```

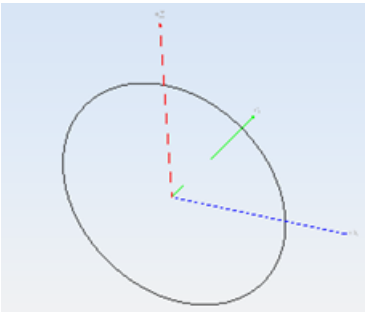


Figure 564: Circle lying in the XZ plane

6.3.85 scale

This command scales the size of an object or objects by a factor specified by the user.

Inline mode usage

scale -h
Displays the help file that summarizes the parameters for this command.

scale –all –p <scaleFactor>
Scales all objects with a scale factor.

scale <name> –p <scaleFactor>
Scales one selected object with a scale factor.

scale –nolocal -all –p <scaleFactor>
Scales all objects with a scale factor, ignoring the reference plane to apply it.

scale –nolocal <name> –p <scaleFactor>
Scales one selected object with a scale factor, ignoring the reference plane to apply it.

Interactive mode usage

Invocation
scale or scale -all

Parameters

- Objects to scale (when not using the all modifier). The objects can be selected by left-clicking on them in the geometry view (hold Ctrl for multiple selection) or by left-clicking on them in the tree view.
- Scale factor. Factors smaller than 1 decrease the size of the selected objects while factors greater than 1 magnify the objects.
- By default, the selected objects will be scaled using the reference plane as reference. Optionally, the user can specify the-noLocalargument to ignore the reference plane when scaling.

Example

We’re going to create a 1x1x1 cube and then scale it so we convert it into a 10x10x10 cube. First, we need to create the original cube:

```
>box
```

```
First corner of base [x y z]: 0 0 0
```

```
Other corner of base [x y]: 1 1
```

```
Height: 1
```

```
>
```

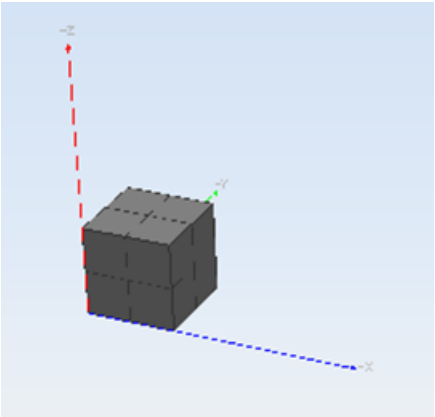


Figure 565: Original 1x1x1 cube

Now, we just have to scale this cube by a factor of 10:

```
>scale
```

```
Select objects on screen (Press enter when done):
```

```
Scale factor (double): 10
```

```
>
```

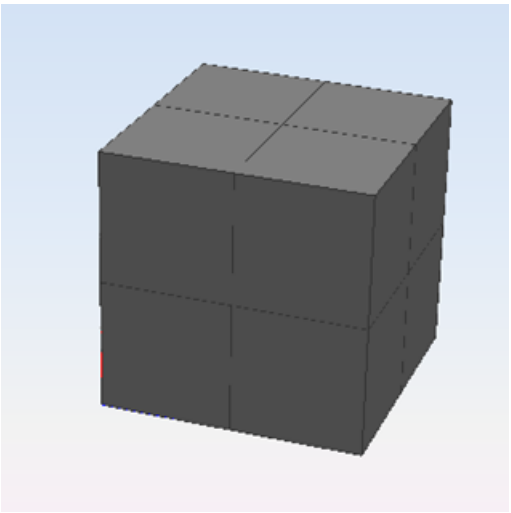


Figure 566: Scaled cube

6.3.86 scale1D

This command scales only the specified dimension (X, Y, Z) of an object or objects by the factor specified by the user.

Inline mode usage

scale1D -h
Displays the help file that summarizes the parameters for this command.

scale1D –all –p <scaleFactor>
Scales all objects with a scale factor.

scale1D <name> –p <scaleFactor>
Scales one selected object with a scale factor.

scale1D –nolocal -all –p <scaleFactor>
Scales all objects with a scale factor, ignoring the reference plane to apply it.

scale1D –nolocal <name> –p <scaleFactor>
Scales one selected object with a scale factor, ignoring the reference plane to apply it.

Interactive mode usage

Invocation
scale1D or scale1D -all

Parameters

- Objects to scale (when not using the all modifier). The objects can be selected by left-clicking on them in the geometry view (hold Ctrl for multiple selection) or by left-clicking on them in the tree view.
- Coordinate. The dimension to be scaled. Only one of the values "x", "y" or "z" are allowed.
- Scale factor. Factors smaller than 1 decrease the size of the selected objects while factors greater than 1 magnify the objects.
- By default, the selected objects will be scaled using the reference plane as reference. Optionally, the user can specify the -noLocal argument to ignore the reference plane when scaling.

Example

We’re going to scale the height of a cone by using the scale1D command.

```
command> cone -c
```

```
Select center [x y z]: 0 0 0
```

```
Bottom radius [double]: 1
```

```
Height [double]: 1
```

```
Top radius [double]: 0
```

```
command> scale1D
```

```
Coordinate {x|y|z}: z
```

```
Scale factor [double]: 2
```

```
command>
```

Result

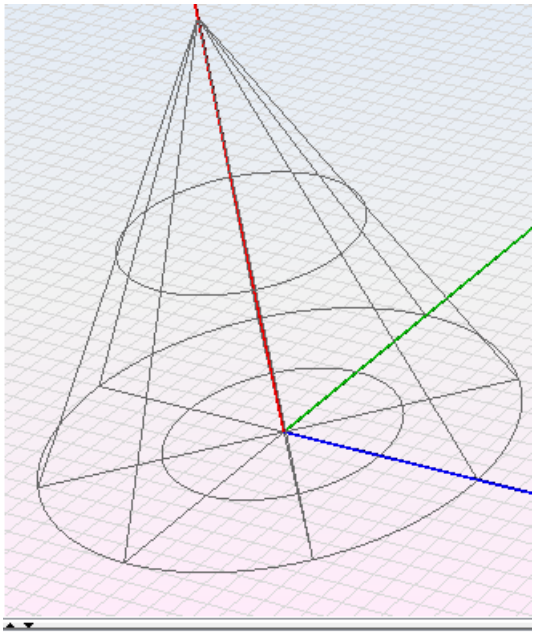


Figure 567: Cone scaled in the Z coordinate

6.3.87 scale2D

This command scales only the two specified dimensions (X, Y, Z) of an object or objects by the same factor specified by the user.

Inline mode usage

scale2D -h
Displays the help file that summarizes the parameters for this command.

scale2D–all –p <scaleFactor>
Scales all objects with a scale factor.

scale2D<name> –p <scaleFactor>
Scales one selected object with a scale factor.

scale2D–nolocal -all –p <scaleFactor>
Scales all objects with a scale factor, ignoring the reference plane to apply it.

scale2D–nolocal <name> –p <scaleFactor>
Scales one selected object with a scale factor, ignoring the reference plane to apply it.

Interactive mode usage

Invocation
scale2D or scale2D -all

Parameters

- Objects to scale (when not using the all modifier). The objects can be selected by left-clicking on them in the geometry view (hold Ctrl for multiple selection) or by left-clicking on them in the tree view.
- First Coordinate. One of the dimensions to be scaled. Only one of the values "x", "y" or "z" are allowed.
- Second Coordinate. The other dimension to be scaled. Only one of the values "x", "y" or "z" (different of the first coordinate) are allowed.
- Scale factor. Factors smaller than 1 decrease the size of the selected objects while factors greater than 1 magnify the objects.
- By default, the selected objects will be scaled using the reference plane as reference. Optionally, the user can specify the -noLocal argument to ignore the reference plane when scaling.

Example

We’re going to convert an ellipsoid into a sphere by using the scale2D command.

```
command> ellipsoid
```

```
Select center [x y z] 0 0 0
```

```
X radius [double] 1
```

```
Y radius [double] 0.5
```

```
Z radius [double] 0.5
```

```
command> scale2D
```

```
Select objects on screen (Press enter when done)
```

```
Coordinate 1 {x|y|z} y
```

```
Coordinate 2 {x|y|z} z
```

```
Scale factor [double] 2
```

```
command>
```

Result

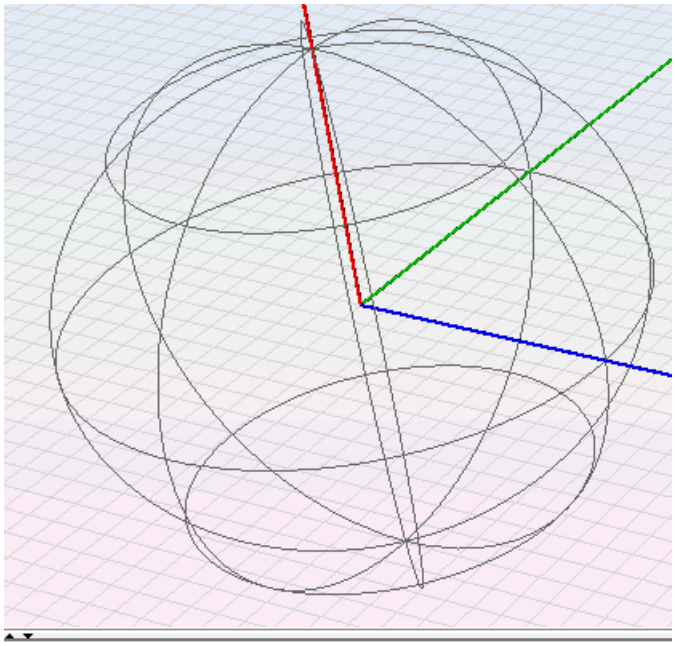


Figure 568: Scaled ellipsoid converted into a sphere by scaling it in two dimensions

6.3.88 scale3D

This command scales the three dimensions (X, Y, Z) of an object or objects by the same factor specified by the user.

Inline mode usage

scale3D -h
: Displays the help file that summarizes the parameters for this command.

scale3D–all –p <scaleFactor>
: Scales all objects with a scale factor.

scale3D<name> –p <scaleFactor>
: Scales one selected object with a scale factor.

scale3D–nolocal -all –p <scaleFactor>
: Scales all objects with a scale factor, ignoring the reference plane to apply it.

scale3D–nolocal <name> –p <scaleFactor>
: Scales one selected object with a scale factor, ignoring the reference plane to apply it.

Interactive mode usage

Invocation
: scale3D or scale3D -all

Parameters

- Objects to scale (when not using the all modifier). The objects can be selected by left-clicking on them in the geometry view (hold Ctrl for multiple selection) or by left-clicking on them in the tree view.
- Scale factor. Factors smaller than 1 decrease the size of the selected objects while factors greater than 1 magnify the objects.
- By default, the selected objects will be scaled using the reference plane as reference. Optionally, the user can specify the -noLocal argument to ignore the reference plane when scaling.

Example

We’re going to create an ellipsoid and then scale it by a factor of 1.5.

```
command> ellipsoid
```

```
Select center [x y z]: 0 0 0
```

```
X radius [double]: 0.5
```

```
Y radius [double]: 1
```

```
Z radius [double]: 0.8
```

```
command> scale3d
```

```
Select objects on screen (Press enter when done)
```

```
Scale factor [double]: 1.5
```

```
command>
```

Result

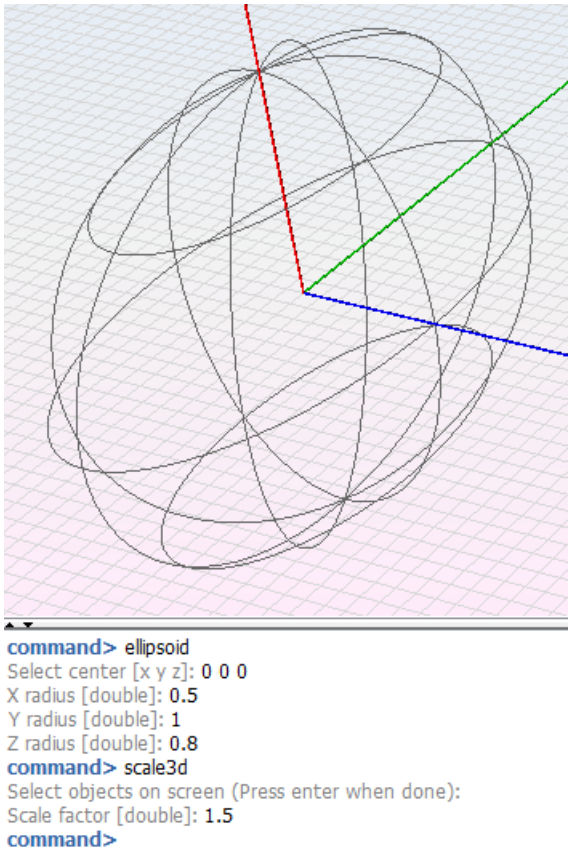


Figure 569: Scaled ellipsoid in the three dimensions

6.3.89 scaleNonUniform

This command scales the three dimensions (X, Y, Z) of an object or objects by the three different scale factors for each dimension specified by the user.

Inline mode usage

scaleNonUniform -h
Displays the help file that summarizes the parameters for this command.

scaleNonUniform –all –p <scaleFactorX>
<scaleFactorY>

scaleNonUniform <name> –p
<scaleFactorX> <scaleFactorY> <scaleFactorZ> Scales the dimensions of the selected object with the corresponding scale factors.

scaleNonUniform –nolocal -all –p
<scaleFactorX> <scaleFactorY> <scaleFactorZ> Scales the dimensions of all objects with the corresponding scale factor, ignoring the reference plane to apply it.

scaleNonUniform –nolocal <name> –p
<scaleFactorX> <scaleFactorY> <scaleFactorZ> Scales the dimensions of the selected object with the corresponding scale factors, ignoring the reference plane to apply it.

Interactive mode usage

Invocation
scaleNonUniform or scaleNonUniform -all

Parameters

- Objects to scale (when not using the all modifier). The objects can be selected by left-clicking on them in the geometry view (hold Ctrl for multiple selection) or by left-clicking on them in the tree view.
- X, Y and Z Scale factors. Factors smaller than 1 decrease the size of the selected objects while factors greater than 1 magnify the objects.
- By default, the selected objects will be scaled using the reference plane as reference. Optionally, the user can specify the -noLocal argument to ignore the reference plane when scaling.

Example

We’re going to convert an ellipsoid into a sphere by scaling each coordinate with a scale factor.

```
command> ellipsoid
```

```
Select center [x y z]: 0 0 0
```

```
X radius [double]: 5
```

```
Y radius [double]: 4
```

```
Z radius [double]: 2
```

```
command> scaleNonUniform
```

```
Select objects on screen (Press enter when done)
```

```
Scale factor for 'X' [double]: 0.2
```

```
Scale factor for 'Y' [double]: 0.25
```

```
Scale factor for 'Z' [double]: 0.5
```

```
command>
```

Result

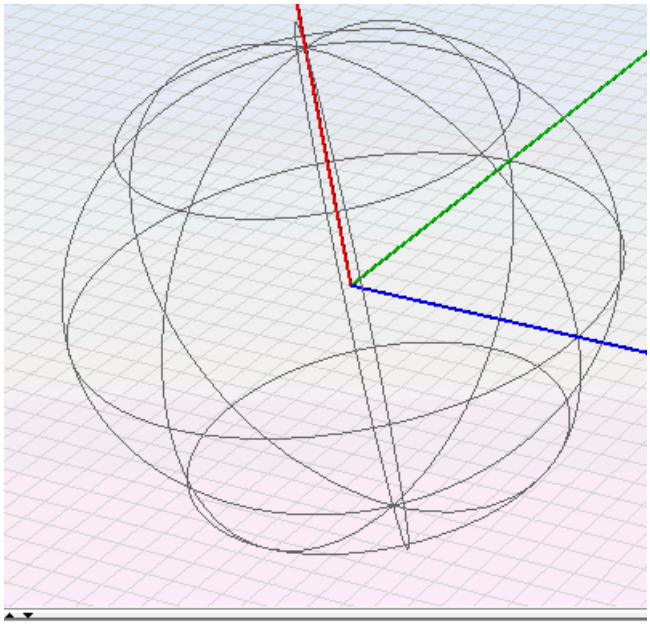


Figure 570: Ellipsoid converted into a sphere by using a non-uniform scale

6.3.90 script

This command loads and executes a script or saves the history to script.

Inline mode usage

script -h
Displays the help file that summarizes the parameters for this command.

script –load <text_plane_script.nfs>
Loads a script.

script –save <text_plane_script.nfs>
Saves a script.

6.3.91 segment

This command draws a straight segment in the geometry.

Inline mode usage

segment -h
Displays the help file that summarizes the parameters for this command.

segment –n <name> -p <x1> <y1> <z1> <x2> <y2> <z2>
Draws a linear segment between the two points specified by the user.

Interactive mode usage

Invocation
segment

Parameters

- Initial point of the segment.
- Final point of the segment.

Example

```
>segment
```

```
First point [x y z]: 0 0 0
```

```
Second point [x y z]: 3 3 2
```

```
>
```

Result

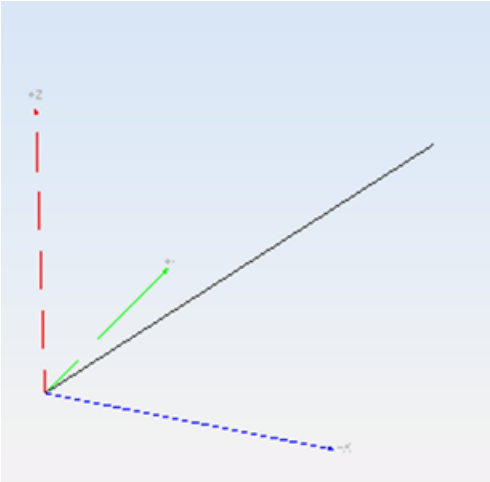


Figure 571: Resulting segment

6.3.92 semicircle

This command draws a semicircle in the geometry.

Inline mode usage

semicircle -h
Displays the help file that summarizes the parameters for this command.

semicircle –n <name> -p <center_x> <center_y> <center_z> <radius> <initial_angle>
Draws a semicircle beginning at the initial angle specified (in degrees) and with the given radius.

Interactive mode usage

Invocation
semicircle

Parameters

- Center point of the semicircle.
- Radius of the semicircle.
- Starting angle (counted counter-clockwise from the +X axis) of the semicircle, in degrees.

Example

```
>semicircle
```

```
Center [x y z] 0 0 0
```

```
Radius [double] 4
```

```
Angle [degrees] 45
```

```
>
```

Result

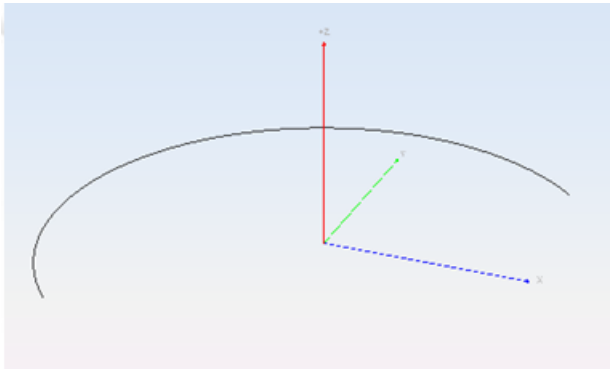


Figure 572: Resulting semicircle

6.3.93 set

This command allows visualizing and/or changing the value of a parameter.

Inline mode usage

set -h
: Displays the help file that summarizes the parameters for this command.

set -v
: Prints the names and values of all parameters.

set -v <varname>
: Displays the values of the parameter named <varname>.

set <varname> {<value1>, <value2>, ..., <valueN>}
: Sets the values <value1>, <value2> … <valueN> to the parameter <varname>.

set <varname> [<initial_value>, <final_value>] <number_samples>
: Sets a linear variation to the parameter named <varname>. If *n* is the number of samples, this command will create a parameter with *n* values: *initial*, *initial + 1*(final - initial)/(n-1)*, *initial + 2*(final - initial)/(n-1)* ... *initial + n*(final - initial)/(n-1)*.

set <varname> <expression>
: Sets the result of the expression <expression> to the values of the parameter <varname>. This expression can be an expression involving constant values or other parameters. Refer to Appendix I for the list of functions that can be used in an expression.

Example

Let's create a new parameter named "myPar" and assign to it the values 1.0, 2.0 and 3.0. To do this, we use the fourth form of the command:

```
> set myPar {1.0, 2.0, 3.0}
```

```
>
```

We can check that the parameter has been created by opening the **Define Parameters** panel or by executing the following command:

```
> set -v myPar
```

```
myPar = {1.0,2.0,3.0}
```

```
>
```

We can see that, effectively, the parameter has been created with the desired values.

Now, let's create another parameter named "myParDouble" whose values are the values of "myPar" doubled. To do this, the user has to enter the following command:

```
> set myParDouble 2*myPar
```

```
>
```

We can check the values of both parameters using the following command:

```
> set -v
```

```
myPar = {1.0,2.0,3.0}
```

```
myParDouble = 2*myPar
```

```
>
```

6.3.94 skinned

This command creates a surface by interpolating it through several curves.

Inline mode usage

skinned -h
Displays the help file that summarizes the parameters for this command.

skinned–s <curve1><curve2>…
Interpolates a surface through the given curves.

Interactive mode usage

Invocation
skinned

Parameters

- Curves used for the interpolation of the surface. The user can select them by left-clicking on them in the geometry view (hold Ctrl and click for multiple selection).

Example

In this example we will create a surface that interpolates through several circles of different radius. To do this, we first create the circles used for the interpolation:

```
> circle 0 0 0 3
```

```
> circle 0 0 2 1
```

```
> circle 0 0 4 2
```

```
>
```

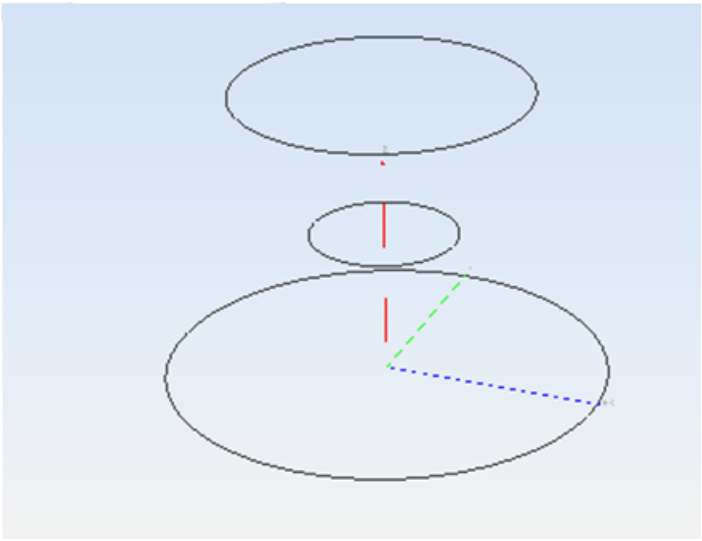


Figure 573: Circles used for the interpolation

To create the surface, we invoke the skinned command and pass to it the three circles as parameters:

```
> skinned
```

```
Select curves on screen (Press enter when done) // Select the three circles
```

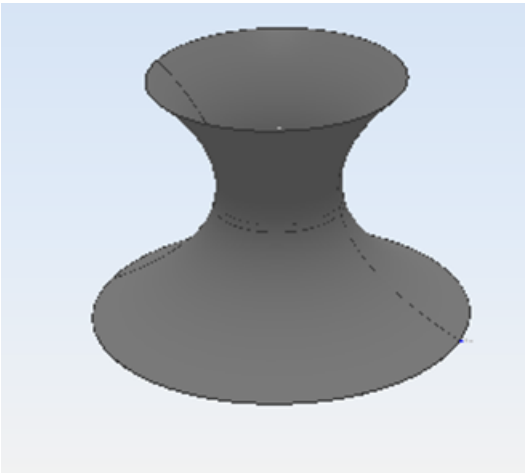


Figure 574: Resulting surface

6.3.95 sphere

This command creates a new sphere in the geometry.

Inline mode usage

sphere -h
Displays the help file that summarizes the parameters for this command.

sphere –n <name> -p <center_x> <center_y> <center_z> <radius>
Creates a sphere centered at the given point and with a given radius.

Interactive mode usage

Invocation
sphere

Parameters

- Center point of the sphere.
- Radius of the sphere.

Example

```
>sphere
```

```
Select center [x y z] 0 1 0
```

```
Select radius 3
```

```
>
```

Result

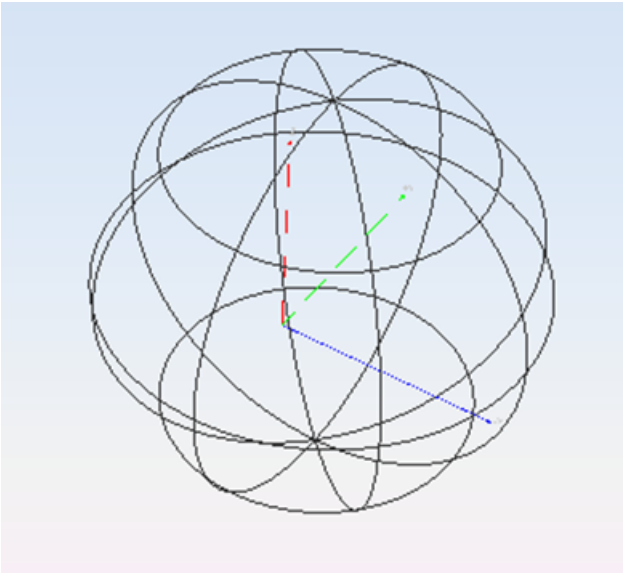


Figure 575: Resulting sphere

6.3.96 spiral

Create a spiral curve with a given center, radius and number of turns.

Inline mode usage:

- spiral -h: Displays the help file that summarizes the parameters for this command.
- spiral -n <name> -p <center_x> <center_y> <center_z> <radius> <numberTurns>: Creates left-turned spiral named <name>, with center at (**center_x**, **center_y**, **center_z**), and the given **radius** and **numberTurns** turns.
- spiral - rightTurned -n <name> -p <center_x> <center_y> <center_z> <radius> <numberTurns>: Creates right-turned spiral named <name>, with center at (**center_x**, **center_y**, **center_z**), and the given **radius** and **numberTurns** turns.

Interactive mode usage: spiral [-rightTurned]

- -rightTurned: Spiral with right-handed turn
- no_param: Spiral with left-handed turn
- [-rightTurned | no_param]: The following parameters are required:
- - Step 1: Center [x y z] x y z
  - Step 2: Radius [double] radius
  - Step 3: Number of turns [double] numberTurns

Example:

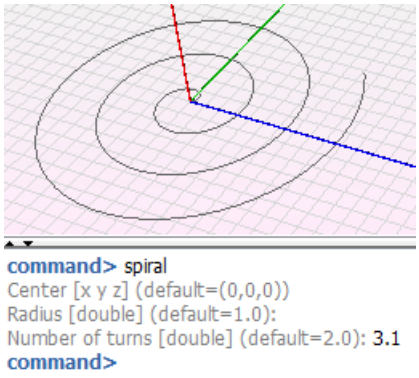


Figure 576: spiral example

6.3.97 spiralCross

Adds a new spiral cross to the geometry.

Inline mode usage:

- spiralCross -h: Display a help message.
- spiralCross -n name -p x y z thickness firstSegmentLength secondSegmentLength thirdSegmentLength: Create a new spiral cross.

Interactive mode usage:

The following steps will be needed in order to create a new spiral cross:

1. Select center [x y z]: Central point of the cross.
2. Thickness [double]: Thickness of the segments of the cross.
3. Length of first segment [double]: Length of the first segment of each blade.
4. Length of second segment [double]: Length of the second segment of each blade.
5. Length of third segment [double]: Length of the third segment of each blade.

Example:

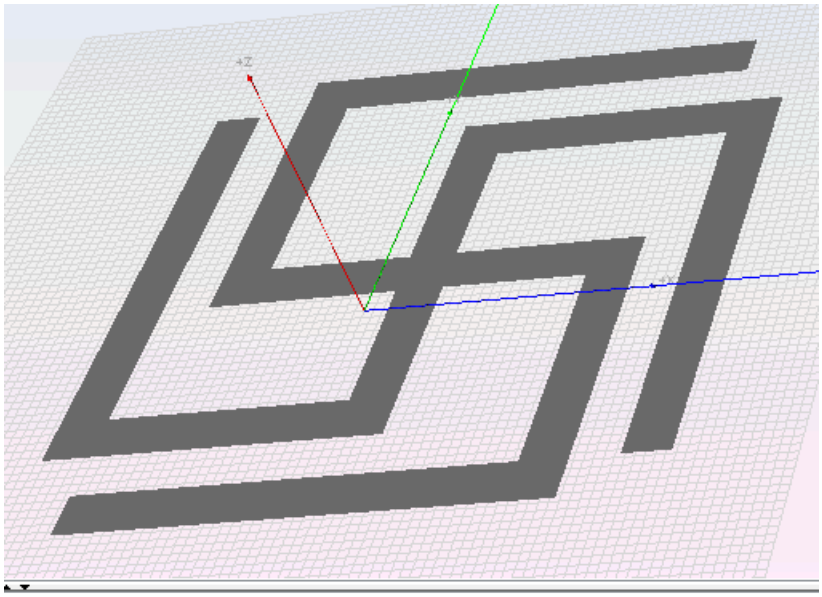


Figure 577: spiralCross example

6.3.98 splitByProjection

This command projects curves onto surfaces or objects and split them with the resulting projections. The projected curves are only used to split the target surfaces and they are not included to the geometry.

Inline mode usage

splitByProjection -h
Displays the help file that summarizes the parameters for this command.

splitByProjection -s <nameCurve1> <nameCurve2> ... –targets <nameSurface1> <nameSurface2> ...
Project the selected curves onto the closest points of the target surfaces selection and then the surfaces are split.

splitByProjection -s <nameCurve1> <nameCurve2> ... –targets <nameSurface1> <nameSurface2> ... -dir <startX> <startY> <start> <endX> <endY> <endZ>
Project the curves onto the target surfaces selection according to the vector defined by the differences <endX-startX> <endY-startY> <endZ-startZ> and then the surfaces are split.

Interactive mode usage

Invocation
splitByProjection or splitByProjection -dir

Parameters

- Curves to project. Selection of curves to be projected. They are not modified and the projected curves are not generated in the geometry.
- Target surfaces. Selection of surfaces and objects to be split by the projected curves.
- Start point. Origin of the projection direction. This parameter is only required when the option -dir is included in the invocation.
- End point. End of the projection direction. This parameter is only required when the option -dir is included in the invocation.

Example

We’re going to split a sphere with the projection of a circle by using the two different modes.

The next commands create the objects required for the projection.

```
command> circle
```

```
Center [x y z]: 0 0 0
```

```
Radius [double]: 0.1
```

```
command> sphere
```

```
Select center [x y z]: 0 0 -1
```

```
Radius [double]: 2
```

```
command>
```

If no options is included in the invocation, the project to the closest point is done. This projection may modify the dimensions of the resulting curves.

If the option -dir is included in the invocation, two points are required to define the projection direction.

Results

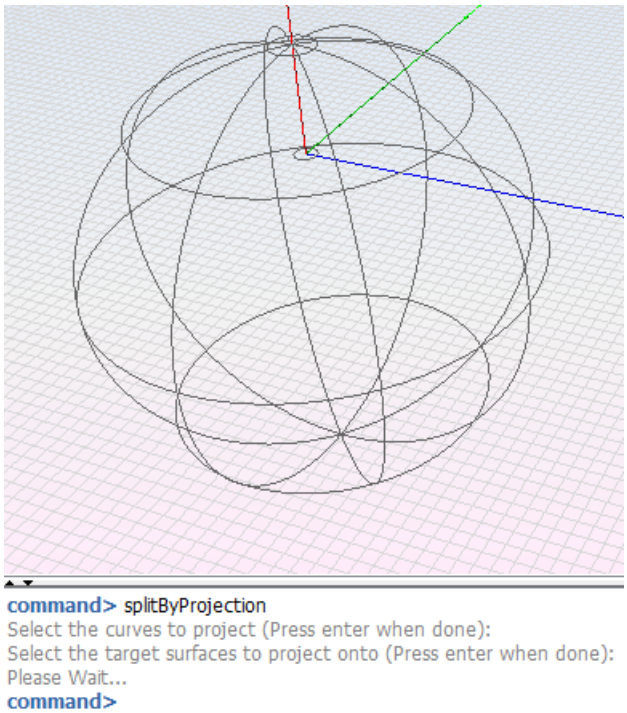


Figure 578: Project a circle onto a sphere and split it

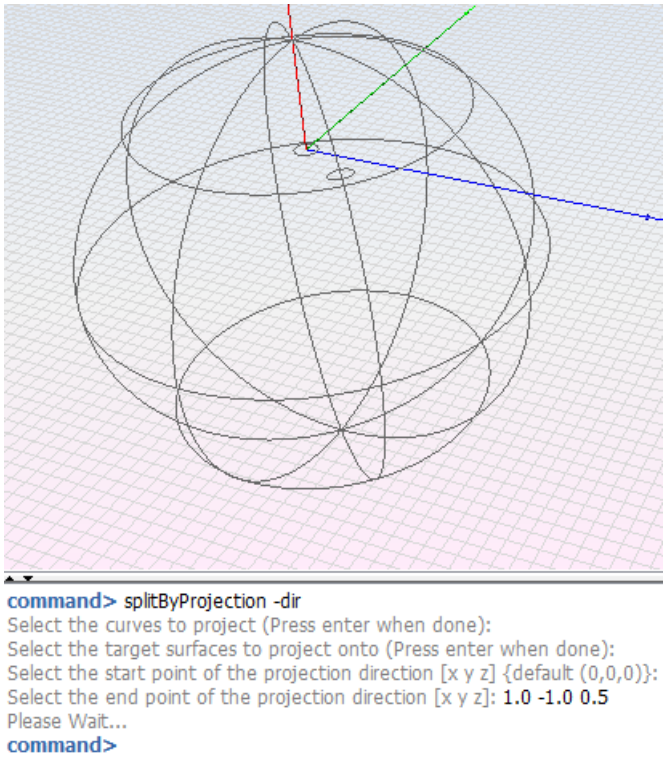


Figure 579: Project a circle onto a sphere by a direction and split it

6.3.99 splitCurves

Split operation applied to curves.

Inline mode usage

splitCurves -h
Displays the help file that summarizes the parameters for this command.

splitCurves–s <curve1><curve2>
Splits between selected curves.

splitCurves–c <object1> –s <curve1>
Selected curves are split by the cutting objects.

splitCurves–p <parametric_value><curve1>
Selected curve is split on parametric space.

Interactive mode usage

Invocation
splitCurves

Parameters

- Selected curves.
- Cutting objects (with –c option).
- Parametric value (with –p option).

Example

Two curves are necessary for this example.

```
>
      splitCurves
```

```
Select the curves to be split on screen (Press enter when done):
```

```
>
```

Result:

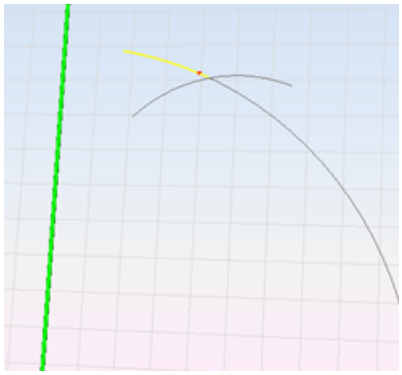


Figure 580: Resulting split curves

6.3.100 splitRing

Add a new split ring to geometry.

Inline mode usage:

- splitRing -h: Display a help message.
- splitRing -n name -p centerX centerY centerZ minorRadius majorRadius openingAngle openingDirection: Creates a new split ring.

Interactive mode usage:

The following steps will be needed in order to create a new split ring.

1. Select center [x y z]: Center point of the ring.
2. Minor radius [double]: Inner radius of the ring.
3. Major radius [double]: Outer radius of the ring.
4. Opening angle [degrees]: The angle of aperture of the opening, in degrees.
5. Opening direction [degrees]: Angle, in degrees, that determines the direction the openings are facing in. An angle of zero degrees means the openings will be facing in the direction of the Y axis. Larger angles will be interpreted as counter-clockwise rotations.

Remark The opening angle MUST be between 0 and 180.

Example:

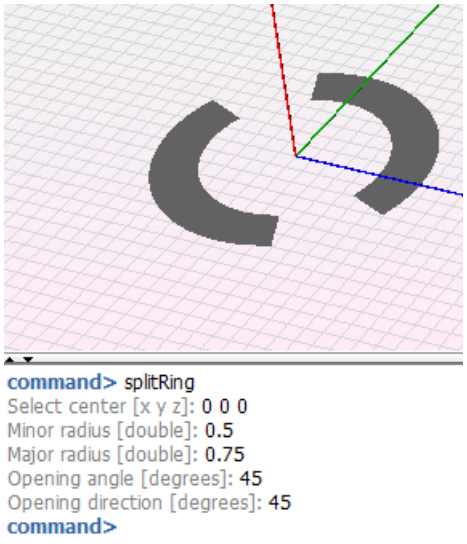


Figure 581: splitRing example

6.3.101 splitSurfaceByIsoparam

This option is useful when a surface needs to be split at a given isoparametric curve. This operation may be used to separate surfaces with extreme curvatures in different parts.

Inline mode usage

splitSurfaceByIsoparam -h
Displays the help file that summarizes the parameters for this command.

splitSurfaceByIsoparam–s <surface> -p <parametric_value>
Splits a selected surface by an isoparametric line in the u-dimension

splitSurfaceByIsoparam–u –s <surface> -p <parametric_value>
Splits a selected surface by an isoparametric line in the u-dimension

splitSurfaceByIsoparam–v –s <surface> -p <parametric_value>
Splits a selected surface by an isoparametric line in the v-dimension

Interactive mode usage

Invocation
splitSurfaceByIsoparam

Parameters

- Selected surface.
- Parametric value.

Example

A curve surface are necessary for this example.

```
>splitSurfaceByIsoparam
```

```
Select one surface to be split on screen (Press enter when done):
```

```
Tip v to change the dimension or insert the parametric value of u(0,1): 0.25
```

```
>
```

Result:

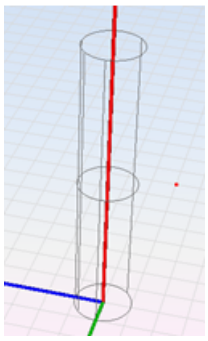


Figure 582: Split surface on u-dimension

6.3.102 splitSurfaceByIsoparamGrid

This option is useful when a surface needs to be split at a given grid of isoparametric curves. This operation may be used to separate surfaces with extreme curvatures in different parts.

Inline mode usage

splitSurfaceByIsoparamGrid -h
: Displays the help file that summarizes the parameters for this command.

splitSurfaceByIsoparamGrid -n <name> -s <selectedSurface> [-u u1 u2 ...uM] [-v v1 v2 ... vN]
: Split the surface *selectedSurface* at the *M* coordinates listed for *U*-dimension and the N coordinates listed for V-dimension, and the resulting surfaces are named as a sequence of (*M*+1)x(*N*+1) surfaces that start with *name_*.

Interactive mode usage: This command cannot be used in interactive mode, and it is only available by using the **Geometry - Tools - Split Surfaces By Curvature** option because of its complexity.

6.3.103 splitSurfaces

Split operation on surfaces.

Script usage:

- splitSurfaces -s surface1 surface2 ... object1 object2...
- splitSurfaces -s surface1... object1... -c surface2 ... object2...

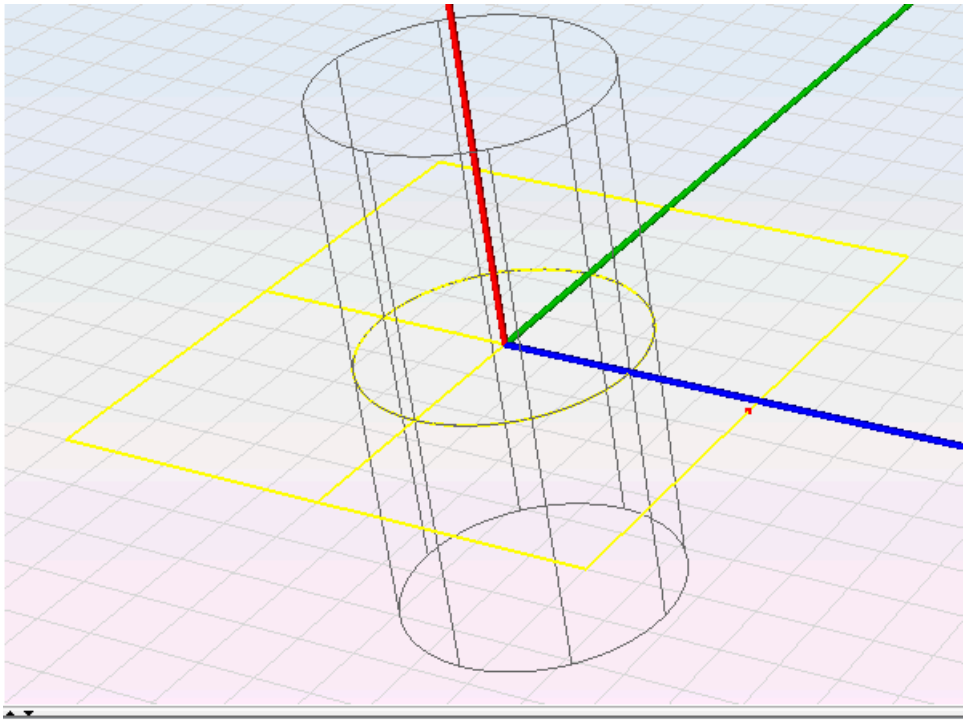
Screen usage:

- **Invocation** splitSurfaces [-c|-tips|-h]
- - no_param: Split between all selected surfaces
  - -c: Selected surfaces are split by the cutting objects
  - -tips: Show the most practical tips on solving unexpected results for this command
  - -h: Print this help

Additional tips:

- The selected objects must produce intersections on the surfaces.
- The selected objects cannot have curvatures larger than 90 degrees.

Example



```
command> cylinder
Select center [x y z]: 0 0 -0.5
Radius [double]: 0.25
Height [double]: 1
command> plane
First corner of plane [x y z]: -0.5 -0.5 0
Plane size [width depth]: 1 1
command> splitSurfaces
Select the surfaces to be split on screen (Press enter when done):
Please Wait...
command>
```

Figure 583: splitSurfaces example

6.3.104 stripCross

Adds new strip cross in geometry.

Inline mode usage:

- stripCross -h: print help message
- stripCross -n name -p x y z a b thickness: create cross

Interactive mode usage:

- Center [x y z]: provide center
- Size [a b]: provide cross dimensions
- Thickness [double]: provide outer thickness

Example:

```
> stripCross
Center [x y z]: 0 0 0
Size [a b]: 0.6 1.3
Thickness [double]: 0.10
```

Result:

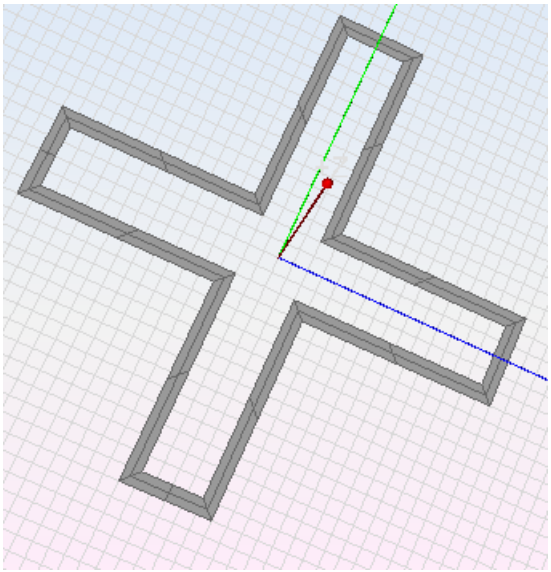


Figure 584: stripCross example

6.3.105 stripHexagon

Adds new strip hexagon in geometry.

Inline mode usage:

- stripHexagon -h: shows usage guide.
- stripHexagon -n name -p x y z radius thickness: create a strip hexagon

Interactive mode usage:

The following steps will be needed in order to create a new strip hexagon:

1. Select center [x y z]: x y z position for the center
2. Radius [double]: radius for the strip hexagon
3. Thickness [double]: outer thickness

Example:

```
> stripHexagon
Center [x y z]: 0.3 0.8 0
Radius [double]: 1.1
Thickness [double]: 0.3
```

Result:

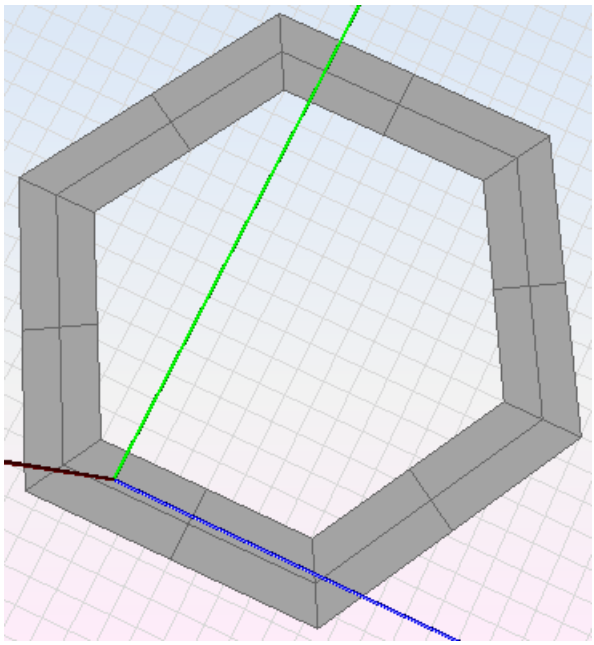


Figure 585: stripHexagon example

6.3.106 stripTrifilar

Add a new hollow trifilar surface to the geometry.

Inline mode usage:

- stripTrifilar -h: Display a help message.
- stripTrifilar -n name -p x y z thickness length holeThickness: Create a new trifilar strip.

Interactive mode usage:

The following steps will be needed in order to create a new trifilar strip:

1. Center [x y z]: Central point of the trifilar.
2. Thickness [double]: Thickness of the entire trifilar (without hole).
3. Length [double]: Length of the segments of the trifilar.
4. Hole thickness [double]: Thickness of the trifilar-shaped hole.

Example:

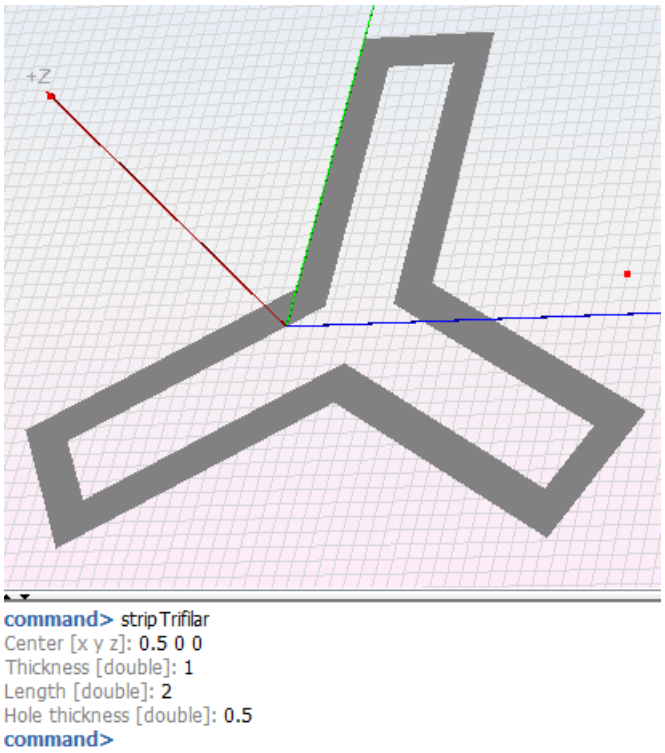


Figure 586: stripTrifilar example

6.3.107 surface

This command allows the user to create a NURBS (Non-Uniform Rational Basis Spline) surface specifying its control points.

Inline mode usage

surface -h
: Displays the help file that summarizes the parameters for this command.

surface –n <name> -p <u> <v> <x y z>
: Allows to create a NURBS surface.

Interactive mode usage

Invocation
: surface

Parameters

- Number of points in the u direction.
- Number of points in the v direction.
- The control points of the surface. For each u coordinate, a point for each v coordinate will be asked. The points are specified by entering its Cartesian (x, y and z) components and a weight parameter that indicates how much that point will pull the surface into it. All these values need to be separated by a blank space.

Example

```
>surface
```

```
Points in u direction [3]: 2
```

```
Points in v direction [3]: 3
```

```
Point [0][0] [x y z w]: 0 2 3 1
```

```
Point [0][1] [x y z w]: 2 2 3 0.7
```

```
Point [0][2] [x y z w]: 2 0 3 1
```

```
Point [1][0] [x y z w]: 0 2 0 1
```

```
Point [1][1] [x y z w]: 2 2 0 0.7
```

```
Point [1][2] [x y z w]: 2 0 0 1
```

```
>
```

Result

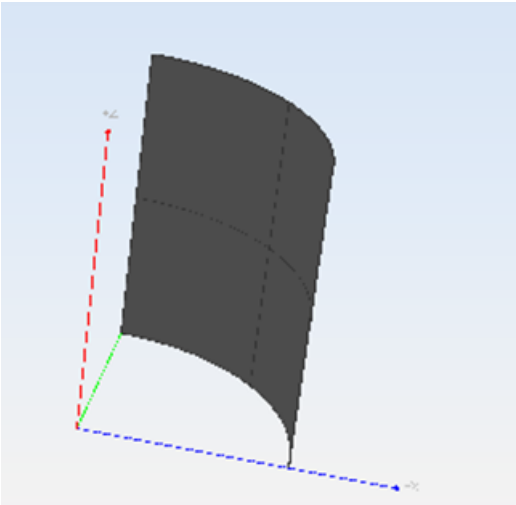


Figure 587: Resulting NURBS surface

6.3.108 surfaceCornerPoints

This command allows drawing a surface that is bound by the corner points specified by the user.

Script usage: surfaceCornerPoints -n name -p x1 y1 z1 x2 y2 z2 x3 y3 z3 [x4 y4 z4]

Screen usage:

Invocation: surfaceCornerPoints [-h]

- -h: Print this help
- no_param: Follow these steps:
- - Step 1: Corner 1 [x y z] x y z
  - Step 2: Corner 2 [x y z] x y z
  - Step 3: Corner 3 [x y z] x y z
  - Step 4: Corner 4 [x y z] x y z, may be empty for a triangle

Example

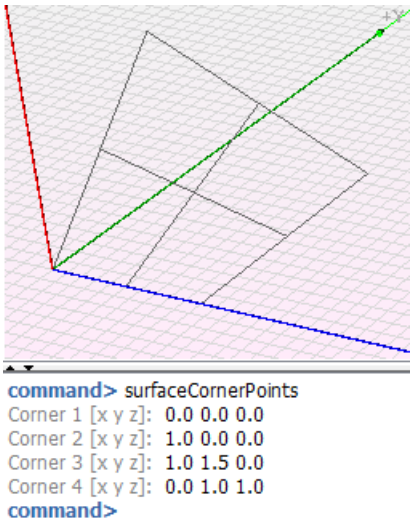


Figure 588: surfaceCornerPoints example

6.3.109 surfaceEdges

This command creates the curves that delimit a surface.

Inline mode usage

surfaceEdges -h
: Displays the help file that summarizes the parameters for this command.

surfaceEdges –s <name>
: Generate the bound curves of the surface selected.

Interactive mode usage

Invocation
: **`surfaceEdges`**

Parameters
: The surface to generate the edges for.

Example

Let’s assume that a planar surface has been created on the geometry. We are going to generate the curves that represent the edges of a surface. Initially, we can see there are no curves in the geometry.

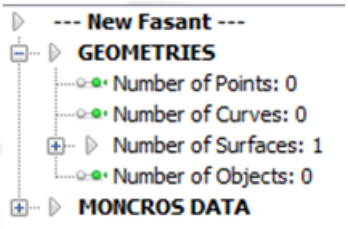


Figure 589: Initial state of the tree view

If we enter the**`surfaceEdges`**command into the command line, select the surface and press Enter, we can see that four curves have been created by looking at the tree view.

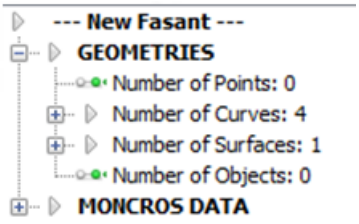


Figure 590: Tree view after the surfaceEdges operation.

Note that the original surface is still in the geometry. We can delete this surface to view more clearly the created curves by selecting it and pressing the Delete key.

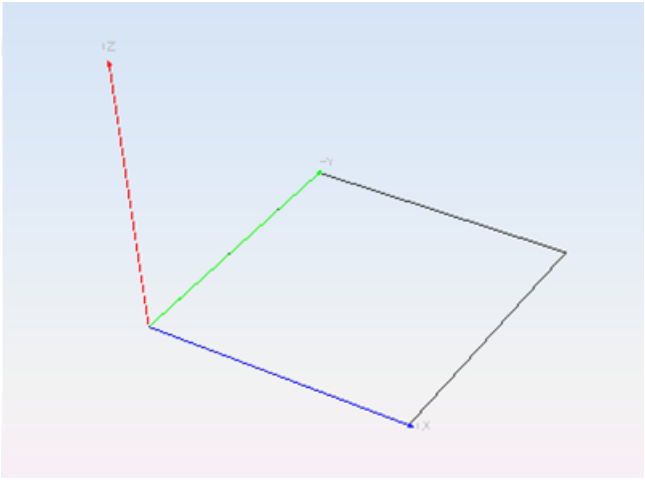


Figure 591: Edges of the surface

6.3.110 surfaceEllipse

Adds a new ellipse to the geometry.

Inline mode usage:

- surfaceEllipse -h: Display a help message.
- surfaceEllipse -n name -p x y z radiusX radiusY: Generates an ellipse surface.

Interactive mode usage:

The following steps will be performed in order to create a new ellipse surface:

1. Select center [x y z]: Center point of the ellipse.
2. Radius x [double]: Radius in the X axis.
3. Radius y [double]: Radius in the Y axis.

Example:

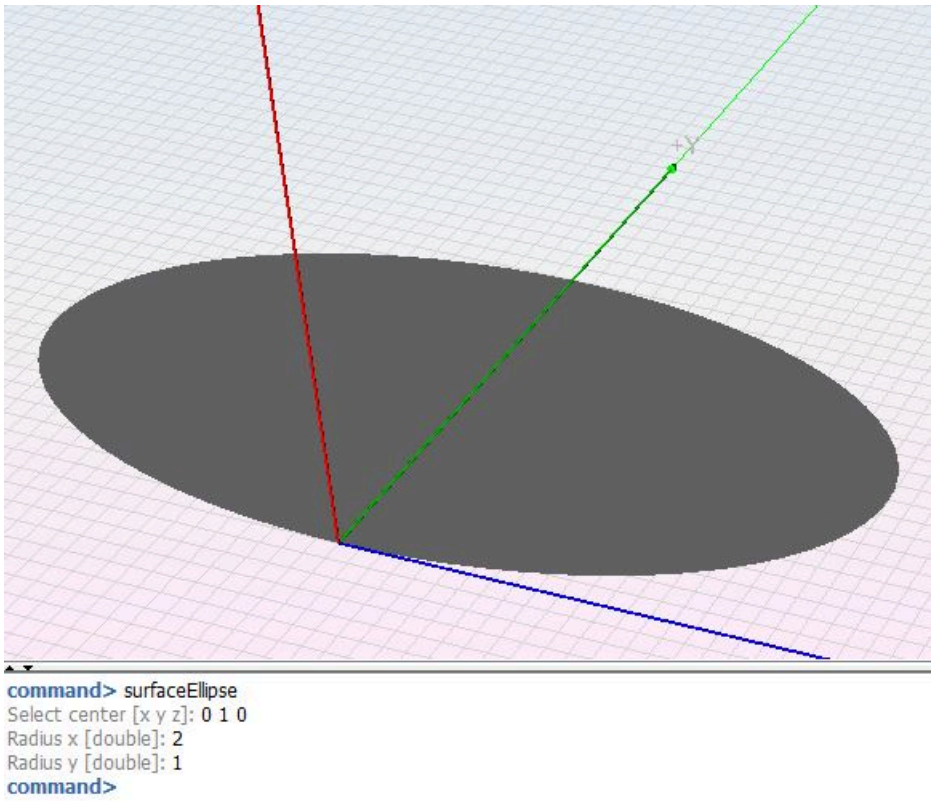


Figure 592: surfaceEllipse example

6.3.111 surfaceFromEdges

Inline mode usage

surfaceFromEdges -h
Displays the help file that summarizes the parameters for this command.

surfaceFromEdges –s all
Selects all curves of the scene.

surfaceFromEdges –s <name>
Selects <name> curve.

Interactive mode usage

Invocation
surfaceFromEdges

Parameters

- Two, three or four curves used for the creation of the coons patch. The set of selected curves must form a closed loop.

Example

In this example, four boundary arcs are considered to create a new coons surface.

```
command> arc
Center [x y z]: 0 0 0
Radius [double]: 1
Initial angle [degrees]: 0
Final angle [degrees]: 90
command> arc
Center [x y z]: 0 0 0
Radius [double]: 1
Initial angle [degrees]: 90
Final angle [degrees]: 180
command> arc
Center [x y z]: 0 0 0
Radius [double]: 1
Initial angle [degrees]: 180
Final angle [degrees]: 270
command> arc
Center [x y z]: 0 0 0
Radius [double]: 1
Initial angle [degrees]: 270
Final angle [degrees]: 360
command> surfaceFromEdges
Select a closed set of 2, 3 or 4 curves (press Enter when done):
command>
```

Result

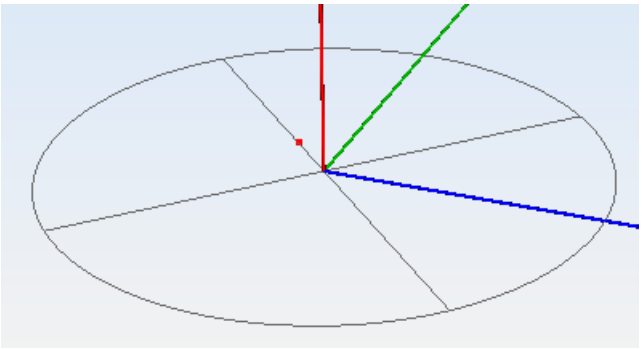


Figure 593: Disk generated from 4 edge arcs

6.3.112 surfaceFunction

This option creates a surface by defining two parameters and three functions.

Inline mode usage:

surfaceFunction -h
Displays the help file that summarizes the parameters for this command.

surfaceFunction –n <name> -p <u_start> <u_end> <u_samples> <v_start> <v_end> <v_samples> <x=f($u,$v)> <y=f($u,$v)> <z=f($u,$v)>
Create surface with three functions that define points in Cartesian.

Interactive mode usage:

Invocation
surfaceFunction

Parameters

- Parameter “u” (start end samples) in order to define range of variable “u”.
- Parameter “v” (start end samples) in order to define range of variable “v”.
- x = f($u,$v) for X coordinate function
- y = f($u,$v) for Y coordinate function
- z = f($u,$v) for Z coordinate function

Remark: The parameters u and v of the functions are bound to the variables $u and $v.

Example:

```
>surfaceFunction
```

```
Parameter '$u' (start end samples): 0 24 48
```

```
Parameter '$v' (start end samples): 0 24 48
```

```
x = f($u,$v) = $u
```

```
y = f($u,$v) = $v
```

```
z = f($u,$v) = sin($u) + sin($v)
```

```
>
```

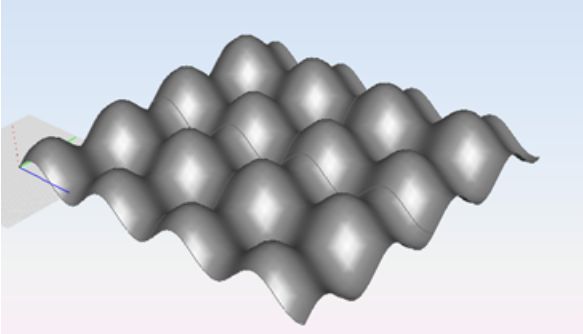


Figure 594: Result of surface function

6.3.113 surfaceInterpolation

This command allows drawing a surface that goes through a set of points specified by the user.

Inline mode usage

surfaceInterpolation -h
Displays the help file that summarizes the parameters for this command.

surfaceInterpolation–n <name> -p <u><v><x y z>
Draws a surface that passes through the points specified by the user, using interpolation.

Interactive mode usage

Invocation
surfaceInterpolation

Parameters

- Number of points in u direction (default 3).
- Number of points in v direction (default 3).
- The points used for the interpolation. For each u coordinate, the user will be asked for the points corresponding to each v coordinate.

Example

```
> surfaceInterpolation
```

```
Points in u direction [3]: 2
```

```
Points in v direction [3]: 3
```

```
Point [0][0] [x y z]: -1 -1 0
```

```
Point [0][1] [x y z]: -1 0 1
```

```
Point [0][2] [x y z]: -1 1 0
```

```
Point [1][0] [x y z]: 1 -1 0
```

```
Point [1][1] [x y z]: 1 0 1
```

```
Point [1][2] [x y z]: 1 1 0
```

```
>
```

Result

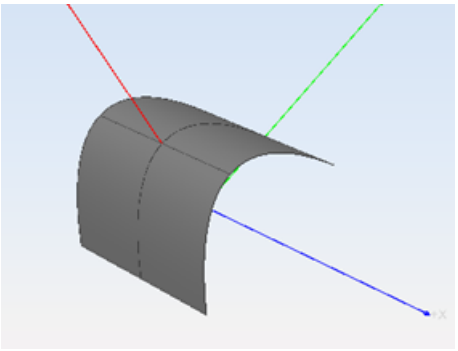


Figure 595: Resulting surface using interpolation

6.3.114 surfaceIsoparamCurve

Any isoparametric curve of a surface (in the U- or V-dimension) may be obtained by using this command.

Inline mode usage

surfaceIsoparamCurve -h
Displays the help file that summarizes the parameters for this command.

surfaceIsoparamCurve –s <surface> -p <parametric_value>
Extract an isoparametric curve in the u-dimension of the selected surface.

surfaceIsoparamCurve –u –s <surface> -p <parametric_value>
Extract an isoparametric curve in the u-dimension of the selected surface.

surfaceIsoparamCurve –v –s <surface> -p <parametric_value>
Extract an isoparametric curve in the v-dimension of the selected surface.

Interactive mode usage

Invocation
surfaceIsoparamCurve, surfaceIsoparamCurve –u or surfaceIsoparamCurve -v.

Parameters

- Parametric dimension “u” or “v”.
- Surface to extract an isoparametric curve.
- Parametric value of the desired coordinate.

Example

In order to do this example, a surface must exist in the scene.

```
>
      surfaceIsoparamCurve
```

```
Select one surface (Press enter when done):
```

```
Tip v to change the dimension or insert the parametric value of u (0,1): v
```

```
Tip u to change the dimension or insert the parametric value of v (0,1): 0.5
```

```
>
```

Result

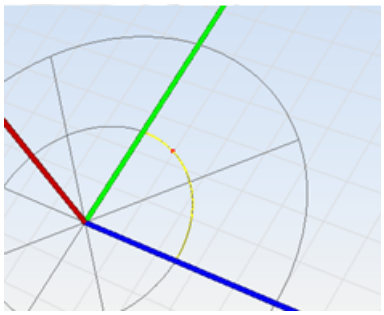


Figure 596: Extracted isoparametric curve from a surface

6.3.115 surfaceSpiral

Create a spiral surface with a given width, radius and number of turns.

Inline mode usage:

- surfaceSpiral -h: Displays the help file that summarizes the parameters for this command.
- surfaceSpiral -n <name> -p <center_x> <center_y> <center_z> <width> <radius> <numberTurns>: Creates left-turned spiral named <name>, with center at (center_x, center_y, center_z), and the given **width**, **radius** and **numberTurns** turns.
- surfaceSpiral -rightTurned -n <name> -p <center_x> <center_y> <center_z> <width> <radius> <numberTurns>: Creates right-turned spiral named < **name**>, with center at (**center_x**, **center_y**, **center_z**), and the given **width**, **radius** and **numberTurns** turns.

Interactive mode usage: surfaceSpiral [rightTurned]

- -rightTurned: Spiral with right-handed turn
- no_param: Spiral with left-handed turn
- [-rightTurned | no_param]: The following parameters are required:
- - Step 1: Center [x y z] x y z
  - Step 2: Width [double] width
  - Step 3: Radius [double] radius
  - Step 4: Number of turns [double] numberTurns

Example:

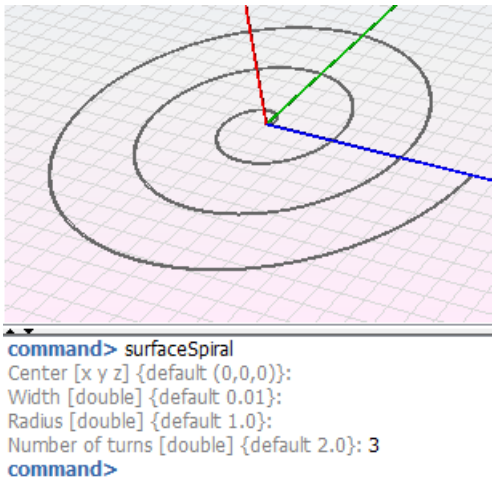


Figure 597: surfaceSpiral example

6.3.116 sweep

Interpolate a surface through the curves chosen by the user.

Script usage: sweep -s railCurve profileCurve

Screen usage: sweep

Follow these steps:

- Step 1: Select the rail curve on screen
- Step 2: Select the cross-section (profile) curve on screen

Example:

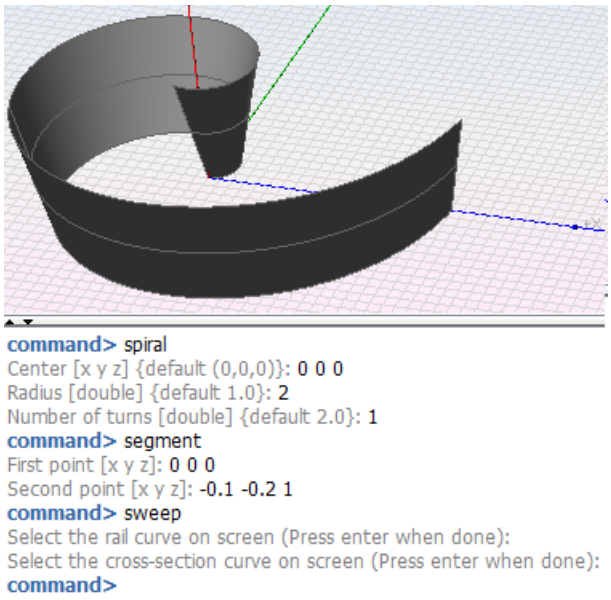


Figure 598: sweep example

6.3.117 symmetric

This command creates an object that is symmetric to the original one with respect to a plane of symmetry.

Inline mode usage

symmetric -h
Displays the help file that summarizes the parameters for this command.

symmetric –all –p <x1 y1 z1> <x2 y2 z2>
Get symmetric objects (all) moving from start point to end point.

symmetric –s <name> –p <x1 y1 z1> <x2 y2 z2>
Get symmetric object (<name> object) moving from start point to end point.

symmetric –c -all –p <x1 y1 z1> <x2 y2 z2>
Get symmetric objects (all) copying from start point to end point.

symmetric –c -s <name> –p <x1 y1 z1> <x2 y2 z2>
Get symmetric object (<name object) copying from start point to end point.

Interactive mode usage

Invocation
symmetric,symmetric -c,symmetric -allorsymmetric -c -all.

Remark
When using the-cparameter, the original object is kept in the geometry. Otherwise, the original object will be deleted.

Parameters

- Original objects (when not using theallparameter) can be selected by clicking on them in the geometry view (hold Ctrl for multiple selection) or in the tree view.
- First point of normal vector for the plane of symmetry.
- Second point of normal vector for the plane of symmetry.

Example

We are going to create a cylinder-like surface by using the example surface created for thesurfacecommand as a base. First, let's create the surface:

```
>surface
```

```
Points in u direction [3] 2
```

```
Points in v direction [3] 3
```

```
Point [0][0] [x y z w] 0 2 3 1
```

```
Point [0][1] [x y z w] 2 2 3 0.7
```

```
Point [0][2] [x y z w] 2 0 3 1
```

```
Point [1][0] [x y z w] 0 2 0 1
```

```
Point [1][1] [x y z w] 2 2 0 0.7
```

```
Point [1][2] [x y z w] 2 0 0 1
```

```
>
```

This surface is only the fourth part of our desired cylindrical surface. Now we are going to create a symmetric surface using the YZ plane as a plane of symmetry. We want to keep the original surface, so we must use the -c parameter:

```
>symmetric -c
```

```
Select objects on screen (Press enter when done) // Select the surface
```

```
First point (for normal vector of symmetry plane) [x y z] 0 0 0
```

```
Second point (for normal vector of symmetry plane) [x y z] 1 0 0
```

```
>
```

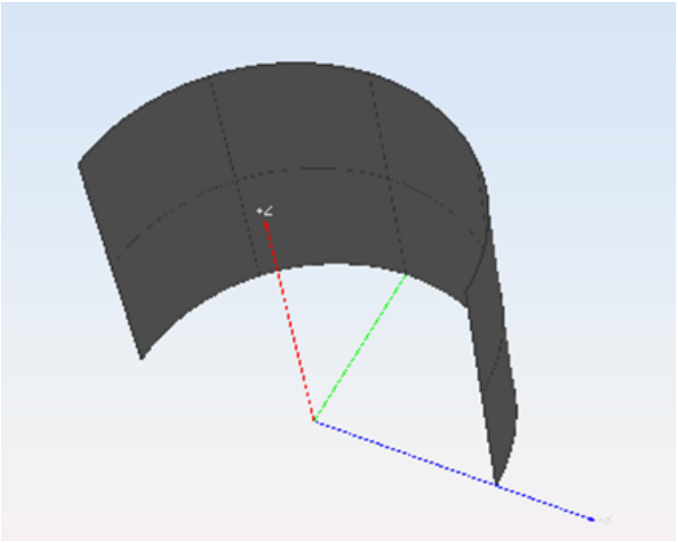


Figure 599: Half cylindrical shape

The only thing left to do to obtain the desired shape is to create the symmetric shape using the existing surfaces, with the XZ plane defined as a plane of symmetry. For convenience, we will use theallparameter this time.

```
>symmetric -c all
```

```
First point (for normal vector of symmetry plane) [x y z] 0 0 0
```

```
Second point (for normal vector of symmetry plane) [x y z] 0 1 0
```

```
>
```

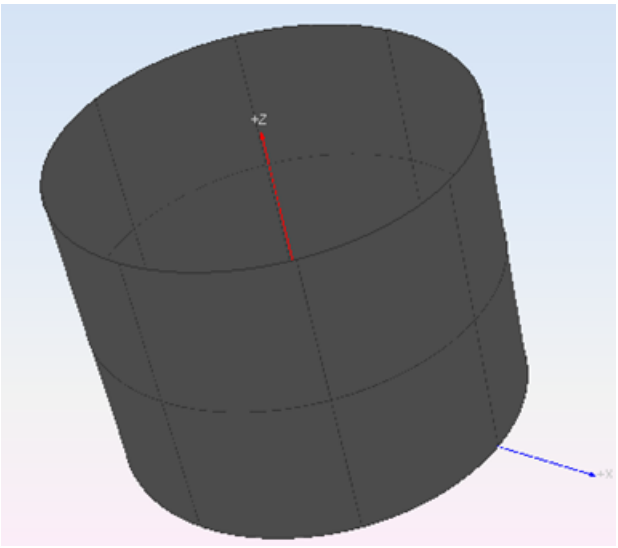


Figure 600: Cylindrical shape

6.3.118 tangent

This command generates a straight line in the geometry tangent to the selected curve that cross the introduced point.

Inline mode usage

tangent -h
Displays the help file that summarizes the parameters for this command.

tangent –n <resulName> -s <selectedCurve> -p <pX><pY><pZ><length>
Generate a straight line tangent to the selectedCurve and crossing the point <pX><pY><pZ> with the specific length.

Interactive mode usage

Invocation
tangent

Parameters

- Point in the tangent line.
- Length of the line.

Example

The following example shows how to insert a tangent line to a circle.

```
command> circle
Center [x y z]: 0 0 0
Radius [double]: 1
command> tangent
Select one curve (Press enter when done):
Insert the coordinates of a point in the tangent line [x y z]: 1.2999999523162842
 0.699999988079071 0.0
Insert the length of the tangent line 1.35
command>
```

Result

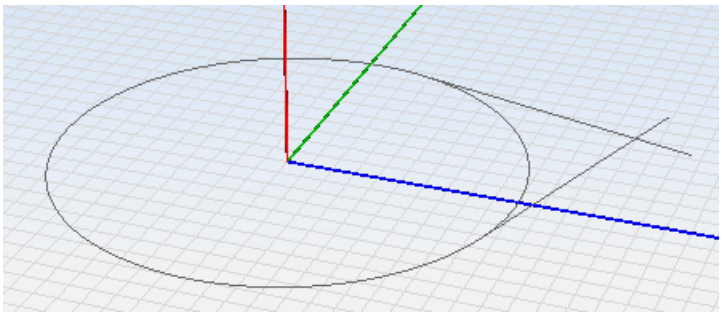


Figure 601: Resulting tangent line

6.3.119 tCross

Adds a new T cross (a cross with T-shaped ends) to the geometry.

Inline mode usage:

- tCross -h: Display a help message.
- tCross -n name -p x y z thickness firstSegmentLength secondSegmentLength: Create a new T Cross.

Interactive mode usage:

The following steps will be needed in order to create a new T cross.

1. Select center [x y z]: Central point of the cross.
2. Thickness [double]: Thickness of the segments.
3. Length of first segment [double]: Length of the first segment of each blade.
4. Length of second segment [double]: Length of the top of the segment of each blade.

Example:

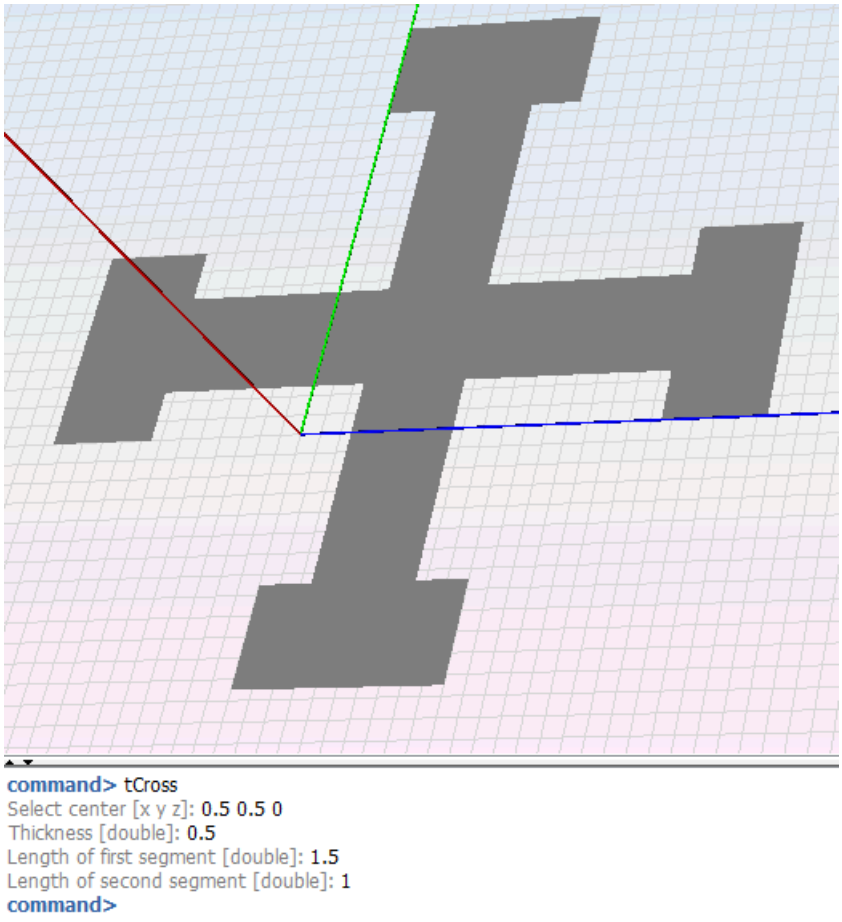


Figure 602: tCross example

6.3.120 threeDipoles

Add three dipoles to geometry.

Inline mode usage

- threeDipoles -h: Display a help message.
- threeDipoles -n name -p x y z dipoleWidth centralDipoleHeight dipoleSeparation lateralDipoleHeight: Creates three dipoles.

Interactive mode usage

The following steps will be needed in order to create new three dipoles:

1. Central point [x y z]: Central point of the set of three dipoles.
2. Dipole size [width height]: Size of the central dipole (width and height). The width will be the same for the lateral dipoles.
3. Dipole separation [gap]: Separation between the central dipole and each of the lateral dipoles.
4. Lateral dipoles height [height]: Height of the lateral dipoles.

Example:

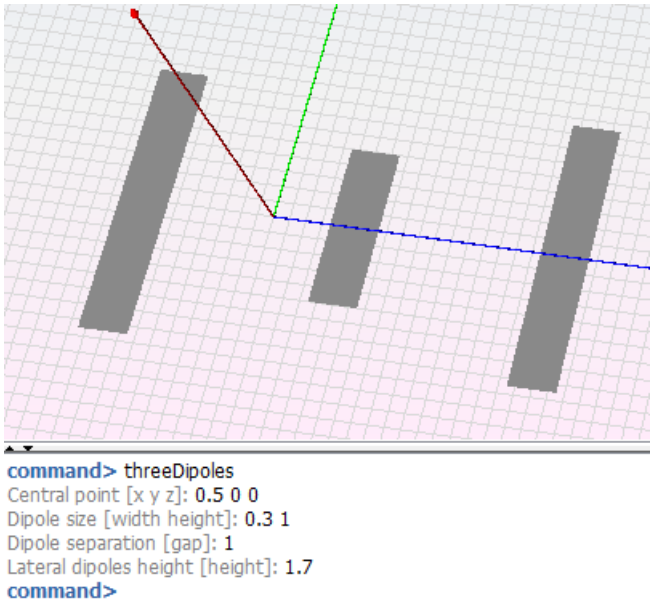


Figure 603: threeDipoles example

6.3.121 threeDipolesUnion

Add three joined dipoles to geometry.

Inline mode usage

- threeDipolesUnion -h: Display a help message.
- threeDipolesUnion -n name -p x y z dipoleWidth centralDipoleHeight dipoleSeparation lateralDipoleHeight unionWidth: Create new set of joined dipoles.

Interactive mode usage

The following steps will be needed in order to create a new set of three joined dipoles:

1. Central point [x y z]: Central point of the set of dipoles.
2. Dipole size [width height]: Size of the central dipole (width and height). The width will be the same for the lateral dipoles.
3. Dipole separation [gap]: Separation between dipoles.
4. Lateral dipoles height [height]: Height of the lateral dipoles.
5. Union width [width]: Width of the union between dipoles.

Example

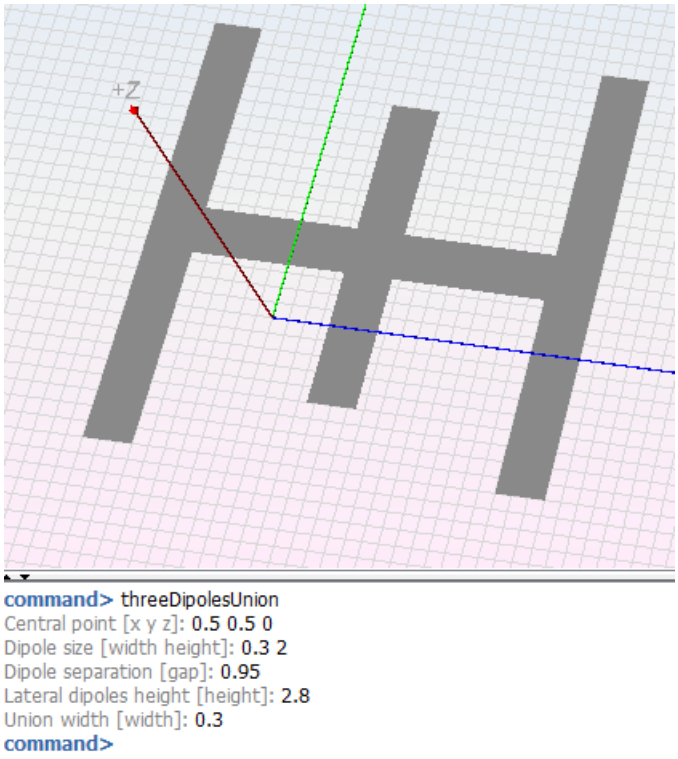


Figure 604: threeDipolesUnion example

6.3.122 torus

Adds a new torus to the geometry panel, receiving the central point, the minor (inner) radius and the major (outer) radius of the torus.

Script usage: torus -n name -p x y z minorRadius majorRadius

Screen usage: torus [-h]

- -h: Print this help
- no_param: Follow these steps:
- - Step 1: Select center [x y z] x y z
  - Step 2: Minor radius [double] minorRadius
  - Step 3: Major radius [double] majorRadius

Example:

```
command> torus
Select center [x y z]: 0 1 0
Minor radius [double]: 0.5
Major radius [double]: 2.5
command>
```

Result:

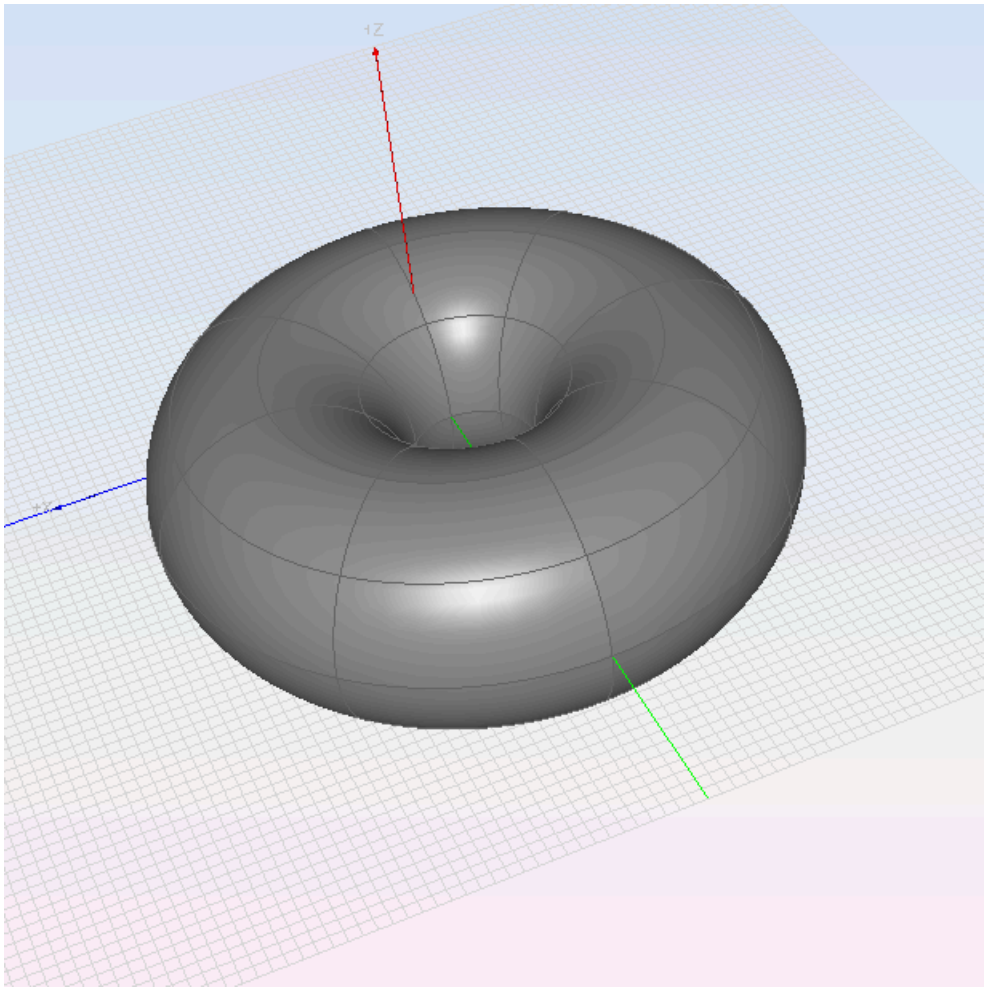


Figure 605: Resulting torus

6.3.123 trifilar

Add a new trifilar to the geometry.

Inline usage

- trifilar -h: Display a help message.
- trifilar -n name -p x y z thickness length: Create a new trifilar.

Interactive mode usage

The following steps will be needed in order to create a new trifilar:

1. Center [x y z]: Central point of the trifilar.
2. Thickness [double]: Thickness of the trifilar segments.
3. Length [double]: Length of the trifilar segments.

Example

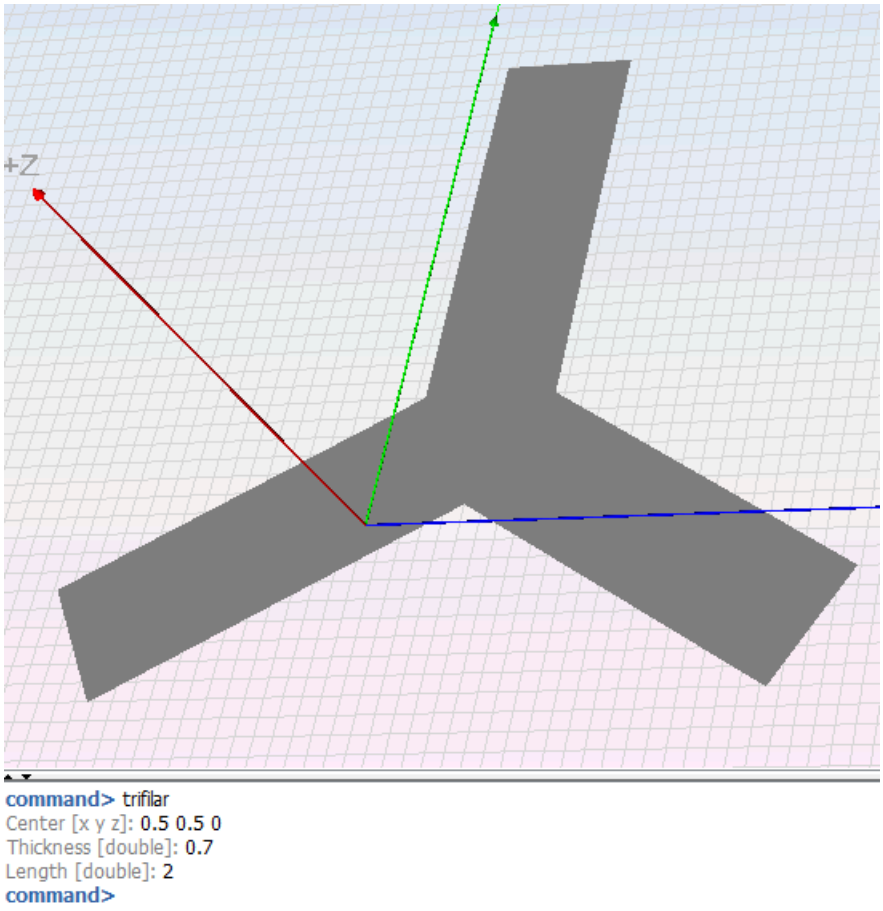


Figure 606: trifilar example

6.3.124 trim

This command allows the user to trim a surface using a curve or set of curves.

Inline mode usage

trim -h
Displays the help file that summarizes the parameters for this command.

trim –s <surface> -c <curve>
Trims the selected surface <surface> with the selected curves <curve>.

Interactive mode usage

Invocation
trim

Parameters

- Surface to trim.
- Cutting closed curves to be used for the trim operation.

Example

We want to create a circular hole (with radius 1) in the center of a plane. This plane is defined as follows:

```
>plane
```

```
Select first point [x y z]: 0 0 0
```

```
Select second point [x y]: 4 4
```

```
> explode
```

```
Select objects on screen (Press enter when done) // Select the plane and press Enter.
```

```
>
```

We need to explode the plane so it will be treated as a surface and not as an object. Next, we will define our cutting curve, in this case, a circle. This is done with the following command:

```
>circle
```

```
Center [x y z]: 2 2 0
```

```
Radius [double]: 1
```

```
>
```

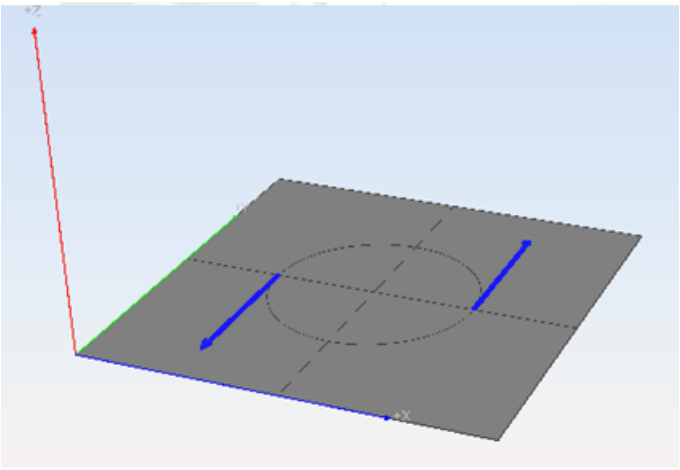


Figure 607: Plane and cutting circle

We have the surface to trim and the cutting curve, so we can proceed with the trim operation. However, the direction of the curve is going counter-clockwise (we can check this by selecting the curve and showing the normals with the **Normals** button on the toolbar), so the trim operation will delete the part of the plane that is outside the circle[2]. We want exactly the opposite, so will need to invert the normals of the circle. This can be done simply clicking on the geometry view while the normals of the curve are being shown.

Now we can finally perform the trim operation. This is done in the following way:

```
> trim
```

```
Select one NURBS surface to trim (Press enter when done): // Select the plane
```

```
Select cutting closed curve(s) (Press enter when done): // Select the circle
```

```
>
```

2. This happens because the part of the surface that is cut off is pointed by the vector resulting from the cross product of the normal vector of the curve and the normal vector of the surface. The normal vector of a plane is pointing in the positive direction of the Z-axis by default, so the cross product vector will point outwards if the normals of the curve are pointing counter-clockwise.

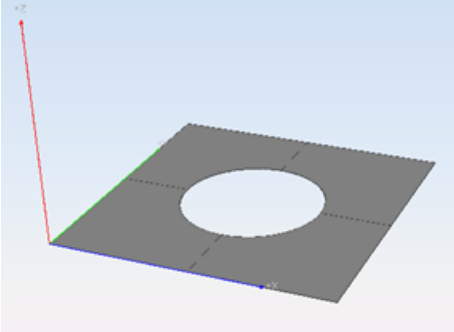


Figure 608: Trimmed plane

6.3.125 unset

Deletes already existing parameters.

Inline mode usage

- unset -h: Displays the help file that summarizes the parameters for this command.
- unset<parameter_name>: Deletes the parameter named <parameter_name>.
- unset -a: Deletes all parameters.

6.3.126 untrim

This command removes cutting loops of the selected surfaces.

Inline mode usage

untrim -h
Displays the help file that summarizes the parameters for this command.

untrim –s <name> -p indexCurve1 indexCurve2
Removes cutting loops of the surface <name>.

Interactive mode usage

Invocation
untrim

Parameters

- The surface that will have its cutting loops removed.
- The selected trimmed curves to remove their loops

Example

We can use the untrim operation to undo a previous trim operation. To demonstrate this, let’s take the example used for the trim command:

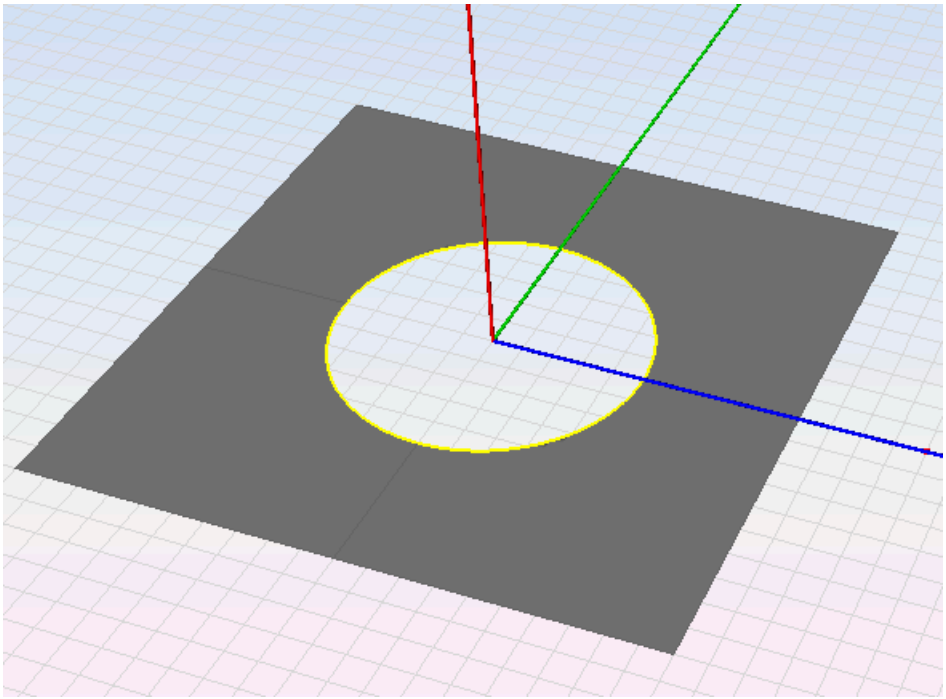


Figure 609: Trimmed surface

To undo the previous trim, the user needs to enter the untrim command in the command line, select the surface and press Enter.

```
>untrim
```

Select one NURBS surface to untrim (Press enter when done):

Press enter to remove all trim curves or select the curves to remove their loops (Press enter when done):

```
>
```

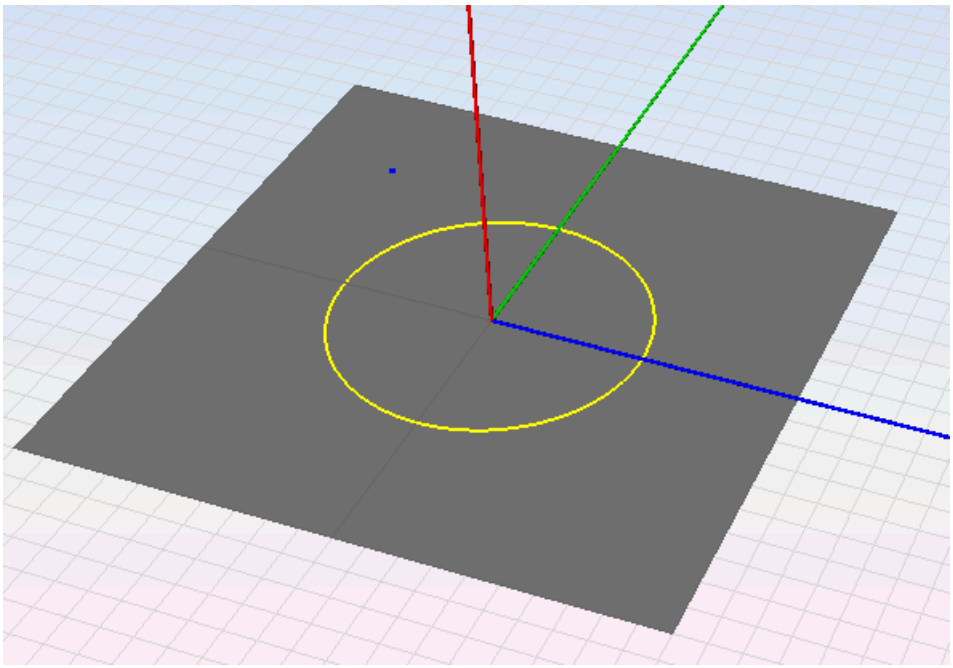


Figure 610: Untrimmed surface

6.4 Appendix I: List of Functions

The following table contains the list of functions that can be used in the value expressions for parameters:

Function	Description
abs(x)	Returns the absolute value of a value.
acos(x)	Returns the arc cosine of a value; the returned angle is in the range 0.0 through pi.
acosd(x)	Returns the arc cosine of a value; the returned angle is in the range 0.0 through 180.0.
asin(x)	Returns the arc sine of a value; the returned angle is in the range -pi/2 through pi/2.
asind(x)	Returns the arc sine of a value; the returned angle is in the range -90.0 through 90.0.
atan(x)	Returns the arc tangent of a value; the returned angle is in the range -pi/2 through pi/2.
atand(x)	Returns the arc tangent of a value; the returned angle is in the range -90.0 through 90.
atan2(y, x)	Returns the angle theta from the conversion of rectangular coordinates (x, y) to polar coordinates (r, theta); the returned angle is in the range -pi through pi.
atan2d(y, x)	Returns the angle theta from the conversion of rectangular coordinates (x, y) to polar coordinates (r, theta); the returned angle is in the range -180.0 through 180.0.
cbrt(x)	Returns the cube root of a value.
ceil(x)	Returns the smallest (closest to negative infinity) value that is greater than or equal to the argument and is equal to a mathematical integer.
cos(x)	Returns the trigonometric cosine of an angle (x) in radians.
cosd(x)	Returns the trigonometric cosine of an angle (x) in degrees.
cosh(x)	Returns the hyperbolic cosine of a value.
deg(x)	Converts an angle measured in radians to an approximately equivalent angle measured in degrees.
exp(x)	Returns Euler's number e raised to the power of a value.
expm1(x)	Returns exp(x) -1.

Function	Description
floor(x)	Returns the largest (closest to positive infinity) value that is less than or equal to the argument and is equal to a mathematical integer.
hypot(x, y)	Returns sqrt(x*x + y*y) without intermediate overflow or underflow.
ln(x)	Returns the natural logarithm (base e) of a value.
log(x, b)	Returns the base (b) logarithm of a value.
log2(x)	Returns the base 2 logarithm of a value.
log10(x)	Returns the base 10 logarithm of a value.
max(x1, x2)	Returns the greater of two values.
min(x1, x2)	Returns the smaller of two values.
pow(x, exp)	Returns the value of the first argument raised to the power of the second argument.
rad(x)	Converts an angle measured in degrees to an approximately equivalent angle measured in radians.
random()	Returns a value with a positive sign, greater than or equal to 0.0 and less than 1.0.
random(min, max)	Returns a value with a positive sign, greater than or equal to min and less than max.
rint(x)	Returns the value that is closest in value to the argument and is equal to a mathematical integer.
round(x)	Returns the closest to the argument, with ties rounding to positive infinity.
sin(x)	Returns the trigonometric sine of an angle (x) in radians.
sind(x)	Returns the trigonometric sine of an angle (x) in degrees.
sinh(x)	Returns the hyperbolic sine of a value.
sqrt(x)	Returns the correctly rounded positive square root of a value.
tan(x)	Returns the trigonometric tangent of an angle (x) in radians.
tand(x)	Returns the trigonometric tangent of an angle (x) in degrees.
tanh(x)	Returns the hyperbolic tangent of a value.
db20(x)	Returns the value corresponding to the function 20*log10(x).
db10(x)	Returns the value corresponding to the function 10*log10(x).

Function	Description
idb20(x)	Returns the value corresponding to the function pow(10,x/20).
idb10(x)	Returns the value corresponding to the function pow(10,x/10).
ridb20(p, x)	Returns the value corresponding to the function cosd(p)*idb20(x).
iidb20(p, x)	Returns the value corresponding to the function sind(p)*idb20(x).
cmplx(real, imag)	Returns new complex (real,imag).
cmplx(real)	Returns new complex (real,0.0).
real(c)	Returns the real part of a complex value.
imag(c)	Returns the imaginary part of a complex value.
conjugate(c)	Returns the conjugate of a complex value.
opposite(c)	Returns the opposite of a complex value.
mod(c)	Returns the module of a complex value.
phase(c)	Returns the phase (argument) of a complex value; the returned value is in the range 0.0 through 360.0.
arg(c)	Returns the phase (argument) of a complex value; the returned value is in the range 0.0 through 2*pi.
argd(c)	Returns the phase (argument) of a complex value; the returned value is in the range 0.0 through 360.0.
sum(c1, c2)	Returns the sum of two complex values.
sum(c, d)	Returns the sum of complex value by a value.
sum(d, c)	Returns the sum of value by a complex value.
sum(c, i)	Returns the sum of complex value by a integer value.
sum(i, c)	Returns the sum of integer value by a complex value.
sub(c1, c2)	Returns the subtraction of two complex values.
sub(c, d)	Returns the subtraction of complex value by a value.
sub(d, c)	Returns the subtraction of value by a complex value.
sub(c, i)	Returns the subtraction of complex value by a integer value.

Function	Description
sub(i, c)	Returns the subtraction of integer value by a complex value.
mul(c1, c2)	Returns the multiplication of two complex values.
mul(c, d)	Returns the multiplication of complex value by a value.
mul(d, c)	Returns the multiplication of value by a complex value.
mul(c, i)	Returns the multiplication of complex value by a integer value.
mul(i, c)	Returns the multiplication of integer value by a complex value.
div(c1, c2)	Returns the division of two complex values.
div(c, d)	Returns the division of complex value by a value.
div(d, c)	Returns the division of value by a complex value.
div(c, i)	Returns the division of integer value by a value.
div(i, c)	Returns the division of integer value by a complex value.
exp(c)	Returns Euler's number e raised to the power of a complex value.
csqrt(c)	Returns the correctly rounded positive square root of a value.
sqrt(c)	Returns the correctly rounded positive square root of a complex value.

To call a function, the user has to specify its name followed by the list of parameters, enclosed in parentheses and separated by commas, for example, `pow(4, 3)`

GTD

7

GTD can be used for 3D analysis of on board antennas and coverage parameters for radiocommunications systems.

This chapter covers the following:

- 7.1 File Menu (p. 659)
- 7.2 Edit Menu (p. 660)
- 7.3 View Menu (p. 661)
- 7.4 Geometry Menu (p. 662)
- 7.5 Materials Menu (p. 663)
- 7.6 Simulation Menu (p. 664)
- 7.7 Antennas Menu (p. 670)
- 7.8 Output Menu (p. 682)
- 7.9 Meshing Menu (p. 699)
- 7.10 Calculate Menu (p. 702)
- 7.11 Show Results Menu (p. 704)
- 7.12 Tools Menu (p. 736)
- 7.13 Help Menu (p. 737)
- 7.14 Training Examples (p. 738)

7.1 File Menu

For information about the File menu, refer to File Menu.

7.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

7.3 View Menu

For information about the View menu, refer to the View Menu.

7.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

7.5 Materials Menu

For information about the Materials menu, refer to Materials.

7.6 Simulation Menu

This menu contains the following options:

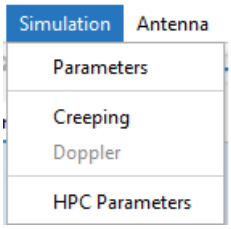


Figure 611: Simulation Parameters menu

7.6.1 Parameters

When the user selects this option, the following panel is shown:

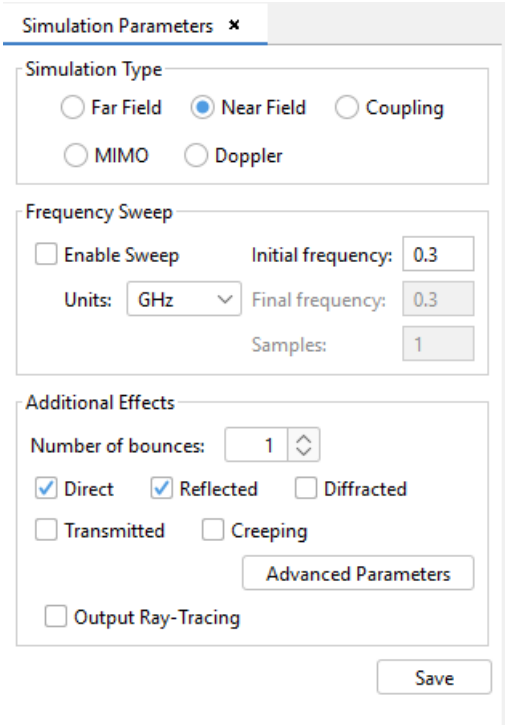


Figure 612: Simulation parameters panel

In this panel, the user can set the main parameters of the simulation:

Simulation Type

There are five types of simulations:

- A **far field** simulation calculates the Far Field in the directions specified in the Output → Observation Directions option.

- A **near field** simulation calculates the Near Field of a set of points specified by the user, in the Output → Observation Points option.
- A **coupling** simulation calculates the coupling between two antennas. The user will have to introduce one or more active antennas and one passive antenna.
- A **MIMO** simulation calculates the same parameters as the coupling simulation, as well as the MIMO channel transfer matrix, the channel correlation and overall system capacity.
- A **Doppler** simulation calculates the Doppler shift. The user will have to introduce the Doppler parameters.

Frequency Sweep

To set the simulation frequency. The following parameters may be modified:

- *Enable Sweep*
  select this option to simulate a range of frequencies.
- *Units*
  to set the magnitude of the frequency. Hz, kHz, MHz and GHz are available.
- *Initial frequency*
  the first step in the frequency range, in the selected **Units**. If the frequency **Sweep** is disabled, this value coincides with the simulation frequency.
- *Final frequency*
  the last step in the frequency range, in the selected **Units**. This parameter may be only modified when the frequency **Sweep** is enabled.
- *Samples*
  number of frequency samples to be analyzed on the frequency **Sweep**. This parameter may be only modified when the frequency **Sweep** is enabled.

Additional Effects

- **Number of bounces**: it is possible to define the number of interactions (bounces) to be taken into account in the simulation for each observation point.
- **Direct**: if this option is selected, the simulation will be performed taken into account direct rays.
- **Reflected**: if this option is selected, the simulation will be performed taken into account reflected rays.
- **Diffracted**: if this option is selected, the simulation will be performed taken into account diffraction.
- **Creeping**: if this option is selected, the simulation will be performed taken into account creeping waves.
- **Transmission**: if this option is selected, the simulation will be performed taken into account transmission.
- **Output Ray-Tracing**: if this option is selected, the `.ray` file will be generated in the simulation and the View Ray option in Show Results menu will be available.

- Selecting **Advanced Parameters** button, the following dialog will appear:

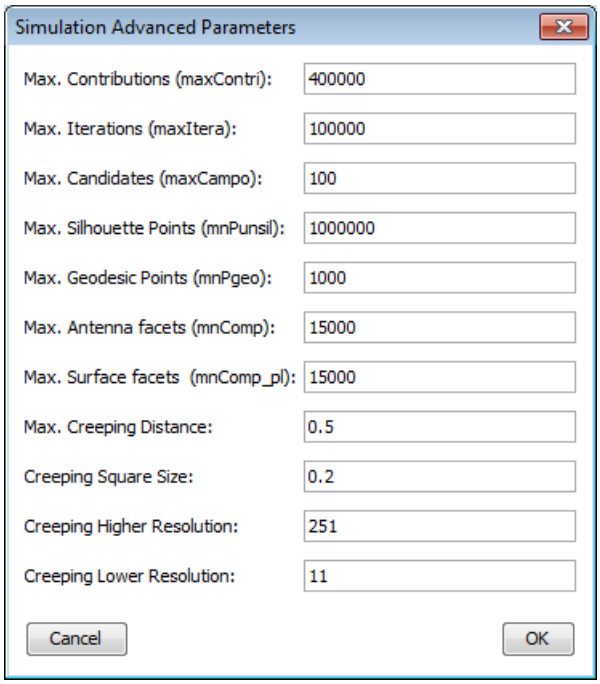


Figure 613: Advanced parameters dialog

In this dialog the user can modify several advanced parameters of the simulation, most of them relative to the analysis of the creeping waves:

- **Max. contributions**: maximum number of contributions to taken into account. A contribution is for instance that due to a given double ray-path.
- **Max. iterations**: maximum number of contributions that are taken into account.
- **Max. candidates**: maximum number of surfaces that can shadow other surfaces.
- **Max. Silhouette Points**: maximum number of points for the silhouette when creeping wave paths are computed.
- **Max. Geodesic Points**: maximum number of geodesic points when computed a creeping wave path.
- **Max Antenna facets**: maximum number of facets to be stored in the anxel of Z-Buffer matrix of the antenna.
- **Max Surface facets**: maximum number of facets to be stored in the anxel of Z-Buffer matrix of every surface.
- **Max Creeping Distance**: maximum distance between a surface and the antenna to increase the creeping sample resolution.
- **Creeping Square Size**: size edge of the square where the creeping sample resolution is increased near the antenna.
- **Creeping Higher Resolution**: higher sampling resolution per surface used to compute the creeping silhouette.
- **Creeping Lower Resolution**: lower sampling resolution per surface used to compute the creeping silhouette.

After setting up the simulation parameters, press the **Save** button to save the configuration.

7.6.2 Creeping

If the **creeping** effect has been selected in the Simulation Parameters Panel, the Creeping parameters option in the Simulation Parameters menu will be available. When this option is selected, the following panel will appear:

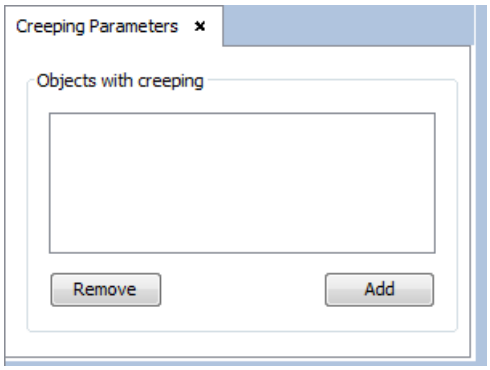


Figure 614: Creeping parameters panel

In this panel, we can add a creeping effect to objects in the geometry. To do this, select an object from the geometry view and press the **Add** button. The selected object will be added to the list of objects with **creeping**. It is possible to remove the creeping effect from an object by selecting it from the list and pressing the **Remove** button.

7.6.3 Doppler

If the Doppler simulation type has been selected in the Simulation Parameters panel, the Doppler option will be available. When this option is selected, the following panel will appear:

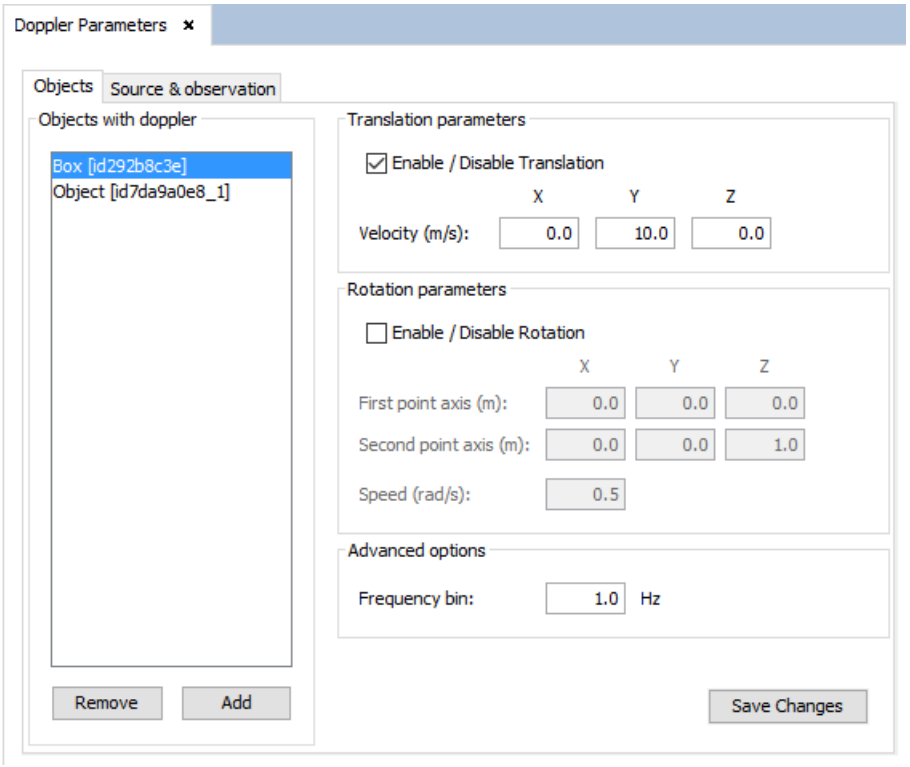


Figure 615: Doppler parameters panel (Objects tab)

In the "**Objects**" tab, the user can add Doppler effect to objects in the geometry. To do this, select an object from the geometry view and press the **Add** button. The selected object will be added to the "Objects with doppler" list. It is possible to remove the Doppler effect from an object by selecting it from the list and pressing the **Remove** button.

- **Translation Parameters**. Any object in the list can have a translation movement. Using this panel the user can define the linear velocity of the object. The **Enable / Disable Translation** check box must be selected.
- **Rotation Parameters.** Any object in the list could have a rotation movement. The **Enable / Disable Rotation** check box must be selected. Using this panel the user can define:
  - **First point axis** first point of the rotation axis.
  - **Second point axis** second point of the rotation axis.
  - **Speed** rotation speed.
- **Advanced Options.** The user can also define the frequency bin.

Once the parameters of the movement of one object have been defined, it is necessary to click on **Save Changes** button in order to save this information and assign it to the selected object.

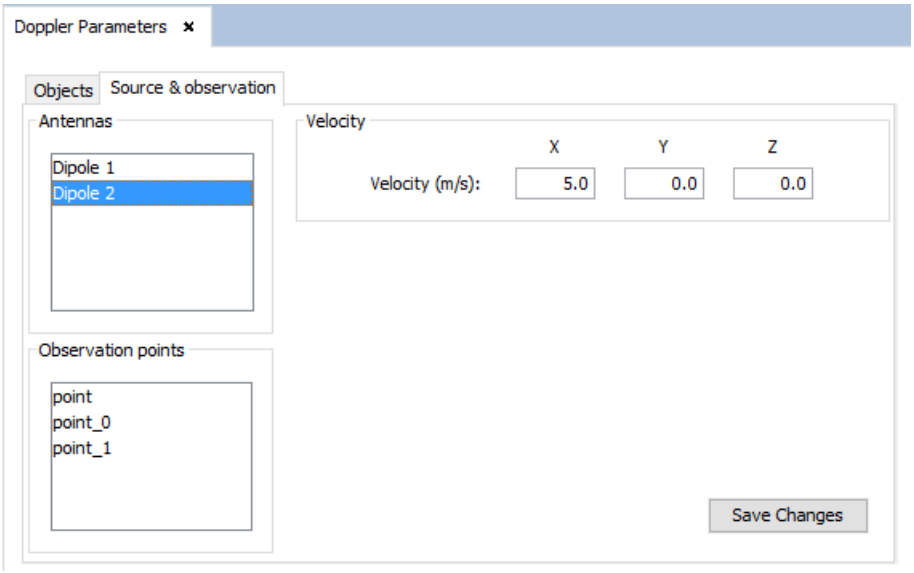


Figure 616: Doppler parameters panel (Source & Observation tab)

In the "**Source & Observation**" tab, the user can set Doppler velocities to individual antennas and observation points. To do this, the user should select an item from the "Antennas" or the "Observation points" lists and fill the Velocity textboxes with the desired velocity. It is possible to select multiple antennas or observation points by holding the Ctrl key and clicking on each antenna or observation point.

Once the velocity for the selected antennas or observation points has been specified, the user needs to press the "Save Changes" button in order to confirm the changes.

7.6.4 HPC Parameters

For more information on how to set up a queueing system, see High Performance Computing (HPC)/ Queueing.

7.7 Antennas Menu

The options in this menu allow the user to create antennas to use in simulations. This menu contains the following options:

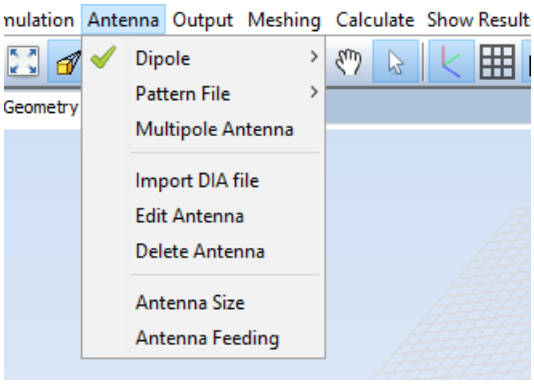


Figure 617: Antennas menu

7.7.1 Dipole

The "Dipole" menu item contains two options to add dipole antennas to the current simulation. The user may add an individual dipole antenna or a array of dipole antennas. The "Dipole Array" option allows the user to add multiple dipole antennas that will be simulated together. This allows the results to be calculated for the group of antennas as a whole, and not for individual antennas (this is useful when calculating coupling results between groups of antennas).

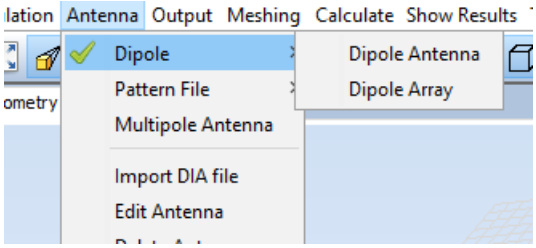


Figure 618: Dipole menu options

Dipole Antenna

The "Dipole Antenna" option allows the user to add an individual dipole to the simulation. When the option is selected, the following panel will be shown:

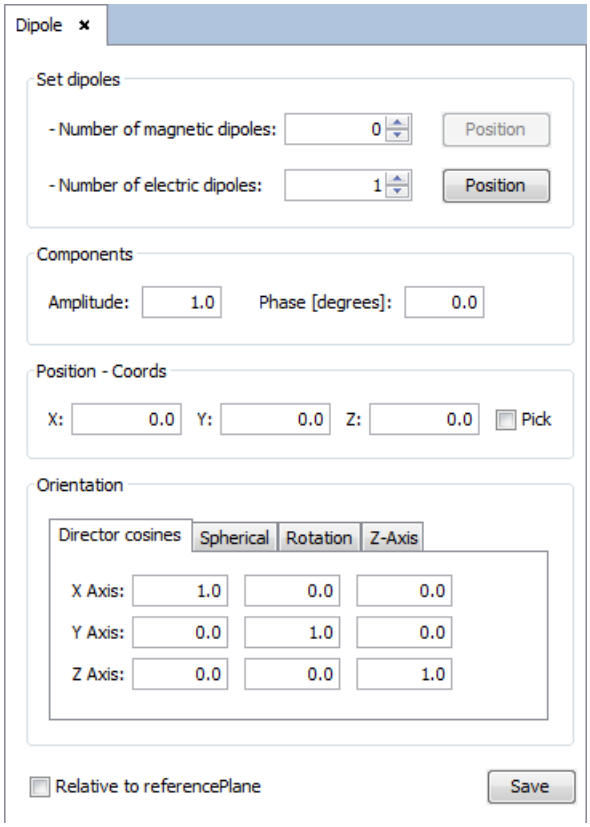


Figure 619: Add Dipole Antenna panel

In this panel, the user can set:

- *Set dipoles*
  the user can set the number of magnetic and electric dipoles. When setting the number of dipoles of each type we can press the *Position* button to adjust their properties. First of all, we need to select the dipole we want to modify by selecting from in the combo box at the top and editing its properties:
  - *Dipolar moment*
    magnitude and phase of the dipolar moment.
  - *Positon - Coords*
    location of the dipole.
  - *Orientation - Director cosines*
    orientation of the dipole coordinate system.

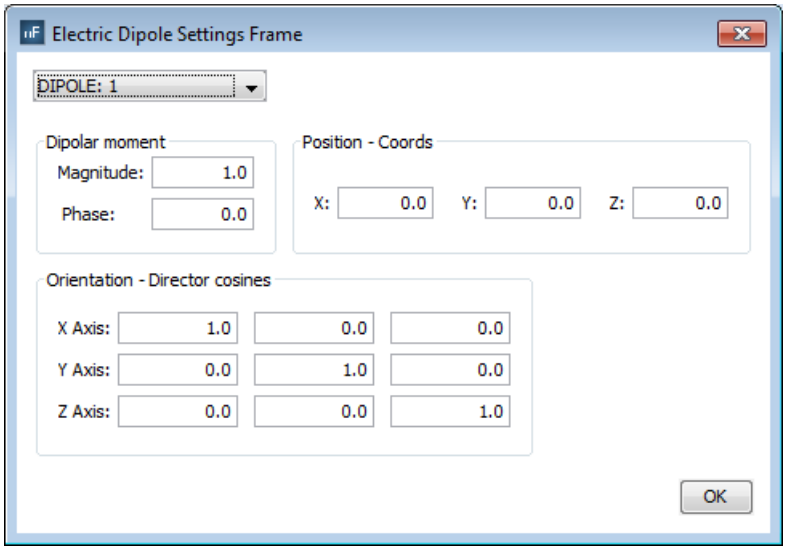


Figure 620: Dipole settings window

- *Components*
  the amplitude and phase of the antenna.

- *Position - Coords*
  location of the antenna (X, Y and Z).

- *Orientation - Director cosines*
  orientation of the antenna coordinate system related to the absolute coordinate system.

- *Relative to referencePlane*
  if this check box is selected, the position of the dipole is relative to the reference plane.

Dipole Array

This option allows the user to add an array of dipole antennas to the simulation. When the user selects this option in the menu, the following panel will appear:

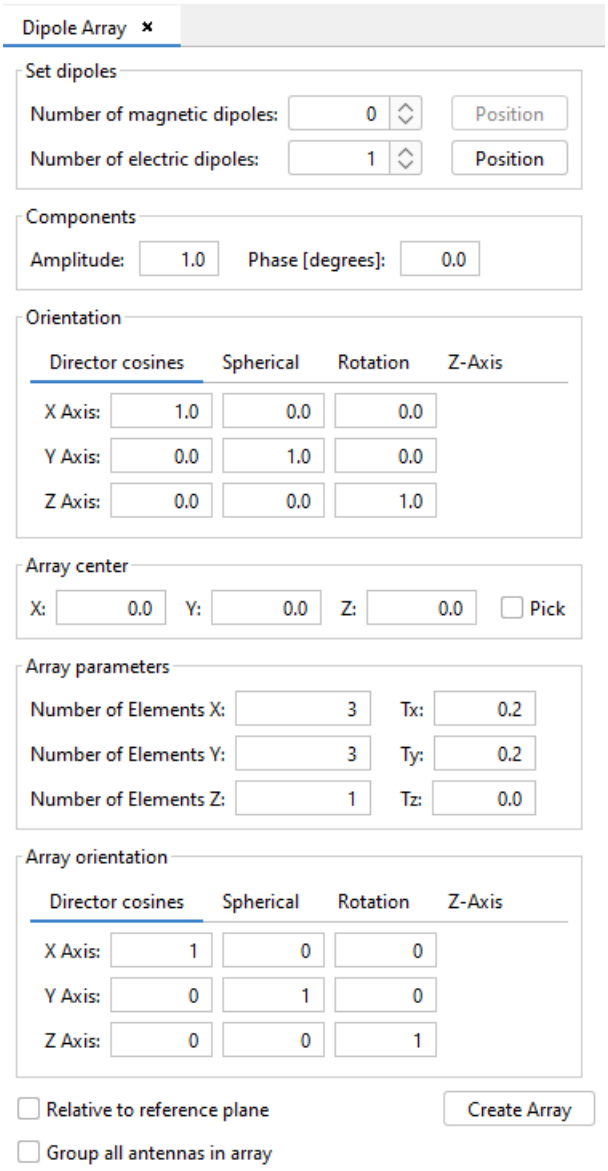


Figure 621: Dipole array panel

In this panel, the user can specify the parameters of each dipole in the same way as in the "Dipole Antenna" panel (as described in the previous section). Additionally, the user can specify parameters regarding the creation of the array:

- The point specified in the "**Array center**" panel is the point where the array center will be located. By checking the "Pick" check box, the user can pick the desired point in the geometry panel and the textboxes will be automatically filled with the point coordinates.
- The "**Array parameters**" panel contains the values that will determine the number of antennas created along each axis (**Number of Elements X**, **Number of Elements Y** and **Number of Elements Z**), as well as the separations between each adjacent antenna along each axis (**Tx**, **Ty** and **Tz**).

- The "**Array orientation**" panel allows the user to specify the directions of the axes considered when creating the array. By default, the array axes are the same as the absolute X, Y and Z axes when the "Relative to reference plane" option is selected.

If the "**Relative to reference plane**" check box is selected, the axes given in the "Array orientation" panel, as well as the point given in the "Array center" panel will be transformed according to the current reference plane coordinate system.

If the **Group all antennas in array** option is selected, the array's antennas will be considered a unique antenna, meaning that they will be simulated as a whole, and the results will be shown for the array – not for each individual antenna.

After specifying the parameters, the user needs to click on the "Create Array" button for the array of dipoles to be added to the simulation.

7.7.2 Pattern File

The "Pattern File" menu item contains two options to add pattern file antennas to the simulation. While "Pattern File Antenna" allows the user to add an individually defined pattern file antenna to the simulation, the "Pattern File Array" option allows them to add multiple pattern file antennas that will be simulated together. This allows the results to be calculated for the group of antennas as a whole, and not for individual antennas (this is useful when calculating coupling between groups of antennas).

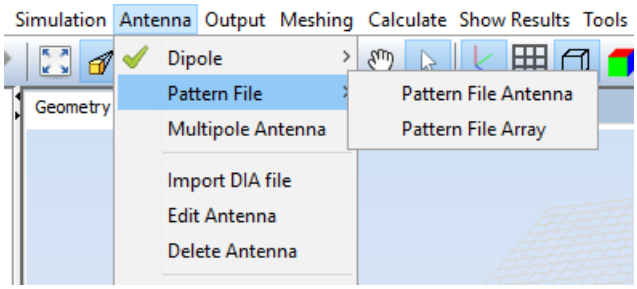


Figure 622: Figure:Pattern File menu

Note In order to add a pattern file antenna or pattern file array to the simulation, it is necessary to import a DIA file (radiation pattern file) before. To do that, select Antenna -> Import DIA File.

Pattern File Antenna

This option allows the user to introduce several files (at different frequencies) that define the radiation pattern. Firstly, the user has to import a radiation pattern file using the Import DIA file option of the Antenna menu.

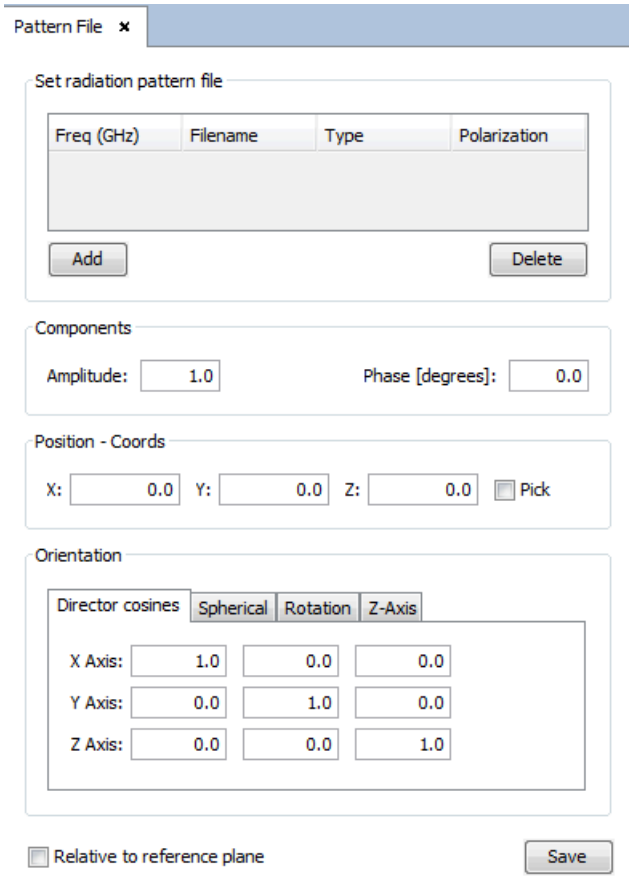


Figure 623: Add Pattern File Antenna panel

The following parameters may be edited:

Set Radiation pattern file
: Click on the Add button to add the imported file to the table where it is possible to specify the frequency, the type and the polarization of the file.

Components
: the amplitude and phase of the antenna.

Position - Coords
: location of the antenna (X, Y and Z).

Orientation - Director cosines
: orientation of the antenna coordinate system related to the absolute coordinate system.

Relative to reference plane
: if this check box is selected, the position of the radiation pattern is relative to the reference plane.

The required format of the radiation pattern file is described in DIA Files.

Pattern File Array

This option allows the user to add an array of pattern file antennas to the simulation. When the user selects this option in the menu, the following panel will appear:

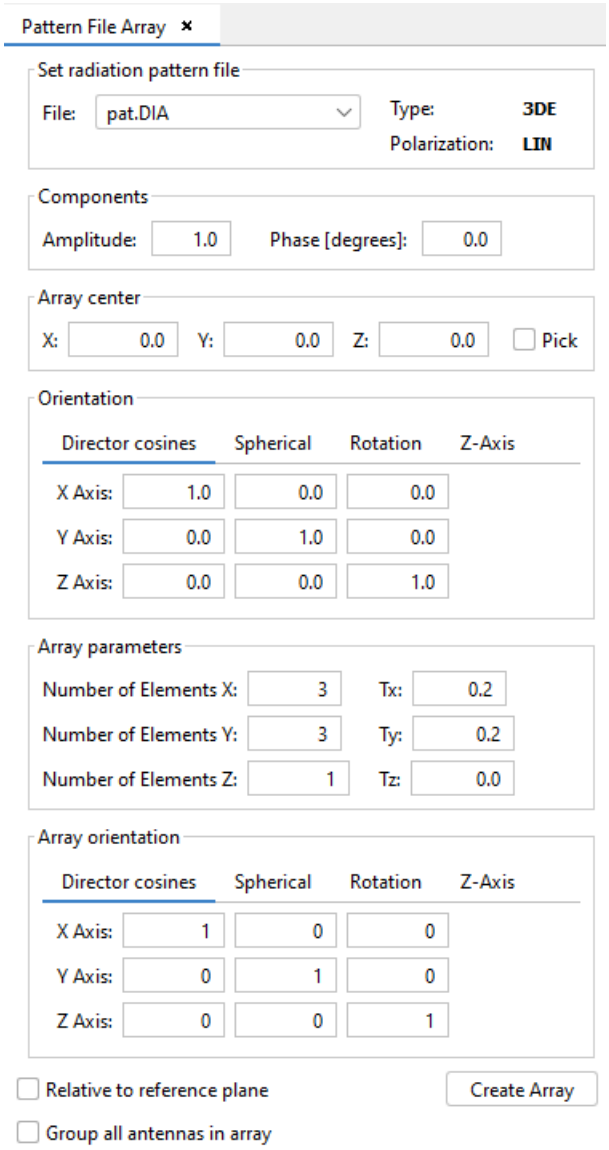


Figure 624: Pattern File Array panel

In this panel, the user can specify the parameters of each pattern file in the same way as in the "Pattern File Antenna" panel (as described in the previous section). Additionally, the user can specify parameters regarding the creation of the array:

- The point specified in the "**Array center**" panel is the point where the array center will be located. By checking the **Pick** check box, the user can pick the desired point in the geometry panel and the textboxes will be automatically filled with the point coordinates.
- The "**Array parameters**" panel contains the values that will determine the number of antennas created along each axis (**Number of Elements X**, **Number of Elements Y** and **Number of**

Elements Z), as well as the separations between each adjacent antenna along each axis (**Tx**, **Ty** and **Tz**).

- The "**Array orientation**" panel allows the user to specify the directions of the axes considered when creating the array. By default, the array axes are the same as the absolute X, Y and Z axes when the "Relative to reference plane" option is selected.

If the **Relative to reference plane** check box is selected, the axes given in the **Array orientation** panel, as well as the point given in the **Array center** panel will be transformed according to the current reference plane coordinate system.

If the **Group all antennas in array** option is selected, the array's antennas will be considered a unique antenna, meaning that they will be simulated as a whole, and the results will be shown for the array – not for each individual antenna.

After specifying the parameters, the user needs to click on the "Create Array" button for the array of pattern files to be added to the simulation.

7.7.3 Multipole Antenna

When this option is selected, the following panel is shown:

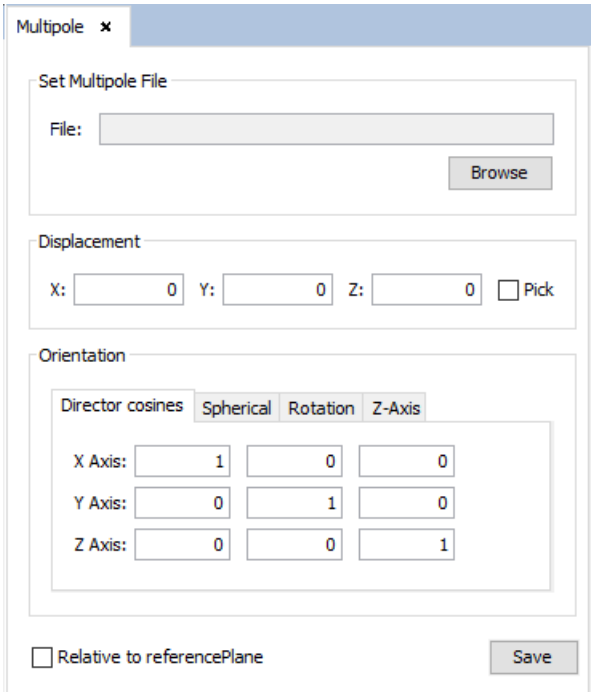


Figure 625: Multipole antenna panel

In this panel, the user can select a multipole antenna file (`.suj`) from the local file system. This file contains the specification for a multipole antenna arrangement. The user can also specify a transformation to apply to the loaded multipole. The transformation consists on a displacement (given by a three coordinate vector) and a rotation (given by the X, Y and Z axes of the local coordinate system of the rotation). If the **Relative to reference plane** check box is selected, the entered

displacement and rotation will be relative to the reference plane coordinate system. Pressing the **Save** button will add the multipole antenna.

7.7.4 Import DIA File

Before defining an antenna as a radiation pattern file, it is necessary to import the file. Click on the icon on the left and select the file to import. This file will be listed in the panel.

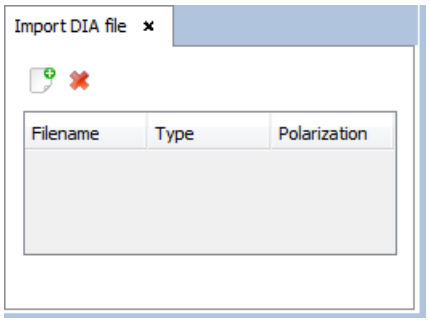


Figure 626: Import DIA file panel

7.7.5 Edit Antenna

When this option is selected and, previously, an antenna of the scene has been also selected, the corresponding panel of the antenna parameters will be opened. For example, if the selected antennas is a dipole, the following panel is shown:

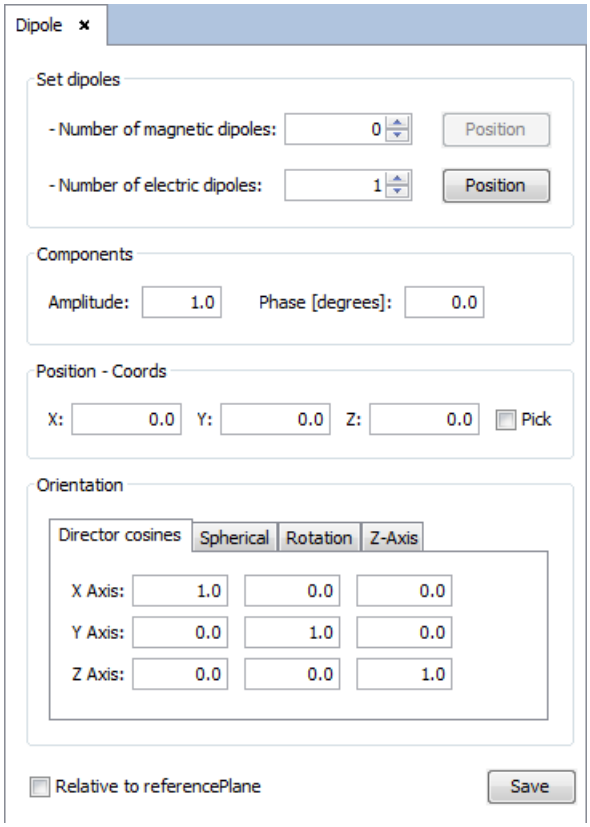


Figure 627: Dipole antenna panel

In this panel the user can modify the parameters of the selected antenna according to the information in Dipole, Pattern File and Multipole Antenna.

7.7.6 Delete Antenna

When this option is selected, the antenna selected on the scene is deleted.

7.7.7 Antenna Display Size

When this option is selected, the following panel is shown:

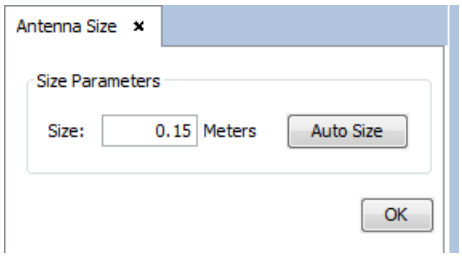


Figure 628: Scale Antennas panel

This option allows the user to scale the antennas represented in the geometry view. The user needs to indicate the factor of scale. Pressing **OK** will cause the antennas in the geometry view (and the new created antennas) to be scaled. The **Auto Size** button allow the user to perform this task automatically.

7.7.8 Antenna Feeding

When the user selects "Antenna Feeding" in the Antenna menu, the following panel is shown:

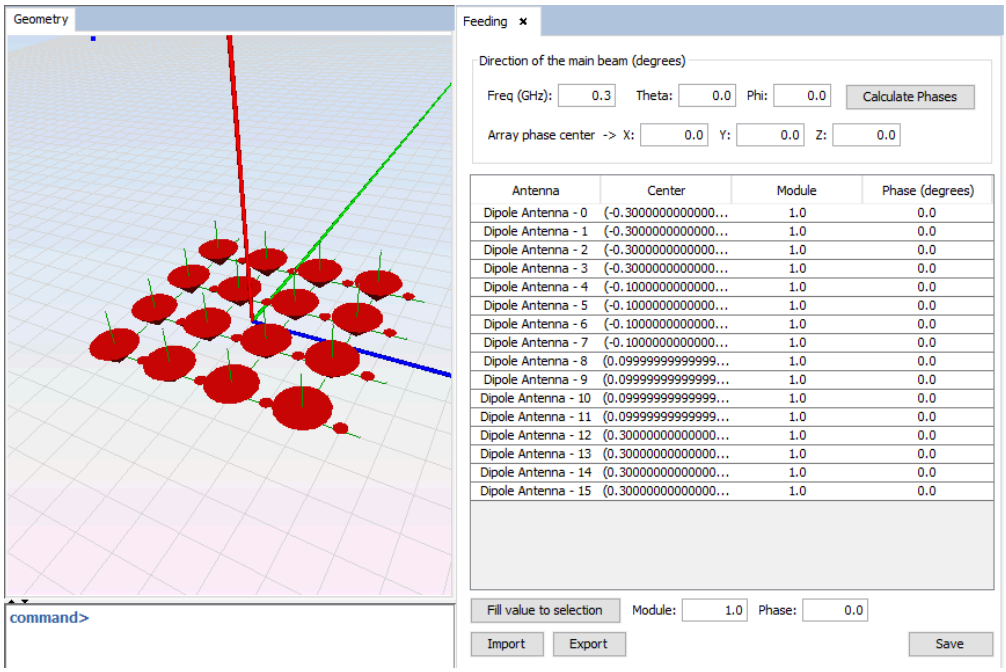


Figure 629: Antenna Feeding panel

This panel allows the user to modify the antenna feeding for each individual antenna (even if it is part of an array). The table shows the position, module and phase of each antenna, all of which can be modified independently.

In the panel labelled as "**Direction of the main beam (degrees)**", the user can set global parameters for the feeding of all antennas. By pressing the "Calculate Phases" button, the application will calculate the module and phase for each antenna based on the given array phase center, frequency and angle of the beam (both theta and phi must be given in **degrees**).

It is also possible to set an specific module and phase to an antenna or group of antennas. To do this, the user should fill the "**Module**" and "**Phase**" textboxes below the table, select the row or rows of the table they want to modify and press the "Fill value to selection" button.

In order to export a text file with the list of modules and phases of the antennas, the user needs to click the "**Export**" button. Reciprocally, in order to import a text file with the list of modules and phases, the user needs to click the "**Import**" button.

The format of the input/output text files used by the import/export functionality is the following:

```
<Number of antennas (N)>
<Module of antenna 1> <Phase of antenna 1 (degrees)>
```

```
<Module of antenna 2> <Phase of antenna 2 (degrees)>
...
<Module of antenna N> <Phase of antenna N (degrees)>
```

7.8 Output Menu

This menu allows the user to set different parameters that determine what results are calculated in the simulation. Depending on the simulation type selected in the **Simulation Parameters** menu, the options shown in this menu will be different:

- If " **Far Field**" simulation type is selected, the **Observation Directions** option will be shown in the **Output** menu.
- If " **Near Field**" or "**Doppler**" simulation type is selected, the **Observation Points** option will be shown in this menu.
- If " **Coupling**" simulation type is selected, the **Coupling** option will be shown in this menu.

Next, the different **Output** options are explained in detail:

7.8.1 Observation Directions

When selecting this option (remember that it's only active when the type of the simulation is **Far Field**), the following panel will appear:

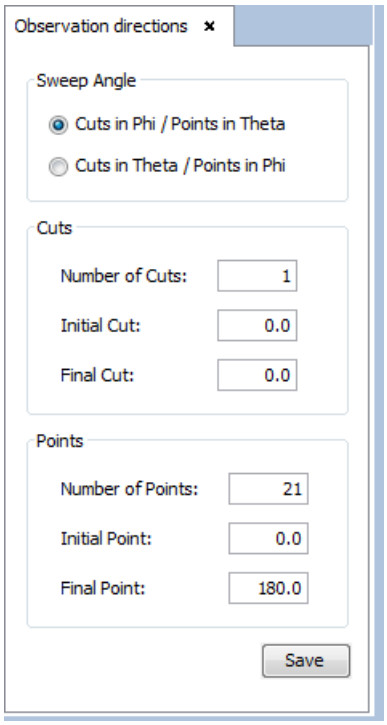


Figure 630: Observation directions panel

In this panel, we can specify the observation directions:

- **Sweep angle**
  - the type of cut (phi cut or theta cut)
- **Cut**:

  - Number of cuts: describes the number of cuts. It is an integer value.
  - Initial cut: defines the angular initial value (in degrees) of the cuts.
  - Final cut: defines the angular final value (in degrees) of the cuts.
- **Number of points**
  - Number of points: describes the number of points. It is an integer value.
  - Initial point: defines the angular initial value (in degrees) of the points.
  - Final point: defines the angular final value (in degrees) of the points.

7.8.2 Observation Points

If the user selects the **Observation Points** option (only available when the **Near Field** or **Doppler** simulation types is selected), the following panel will appear:

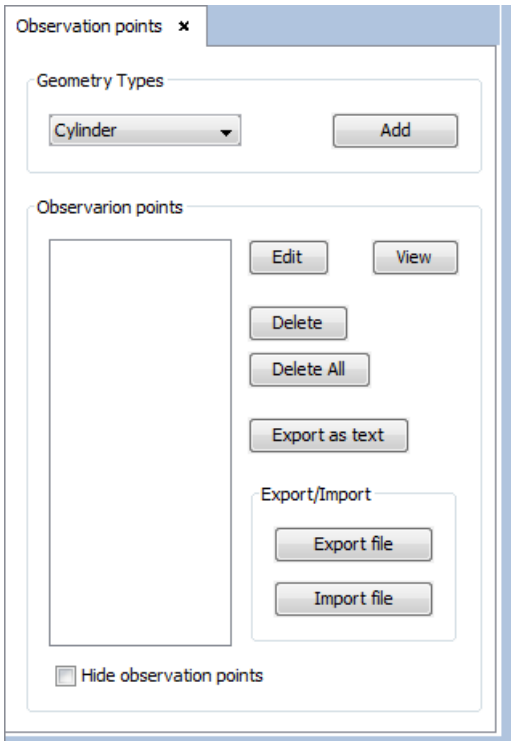


Figure 631: Observation points panel

The user can select near field observation points in different ways. The next options are available:

- **Geometry Types** observation points may be inserted as a group that defines different shapes. Select the desired type entity and click on **Add** button to set its parameters. The available **Geometry Types** are detailed in next subsections. Before adding new elements, check that the **Name** of the new entity to be added is different from the already existing ones; otherwise, an error will be shown.
- **Observation points** this section contains several options to work with observation points.
- ◦ The left list contains all the observation shapes that have been added as observation points.

  - The **Edit** button allows editing an observation points shape selected on the left list.
  - The **Delete** button allows removing the observation entities selected on the left list.
  - Use the **Delete All** button to remove all the observation points shapes on the left list.
  - The **Export as Text** button allows saving all the generated observation points. Every point is defined in a different line by its X, Y and Z coordinates.
  - The **Import File** allows the user to add a previously saved observation points file. Only `.obs` files are supported, which are the internal format generated by the application.
  - The **Export File** allows the user to save the current observation points configuration in a new file. It must be saved as `obs` file, which is the internal format generated by the application.
  - Check the **Hide observation points** option to hide the observation points from the main geometry window but without deleting them. Disable this option to show the observation points.
- **Hide observation points** this option hide the observation points.

Cylinder

The observation cylinder may be defined as a closed cylinder (circular section) or only a cylindrical sector (arc section). The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Base Center** coordinates where the base of the cylinder will be placed (in the XY plane).
- **Radius** radius of the cylinder
- **Height Coordinates** to define the hight sampling rate of the cylinder.
  - **Number of points** define the number of samples to be generated along the height of the cylinder (in the Z axis).
  - **Height** defines the total height of the cylinder.
- **Angle Coordinates** to define the circular sampling rate of the cylinder.
  - **Number of points** define the number of samples to be generated along the transversal section of the cylinder (in the XY plane).
  - **Initial angle** angle where the first circular sample is generated in the transversal section, in degrees.
  - **Final angle** angle where the last circular sample is generated in the transversal section, in degrees.

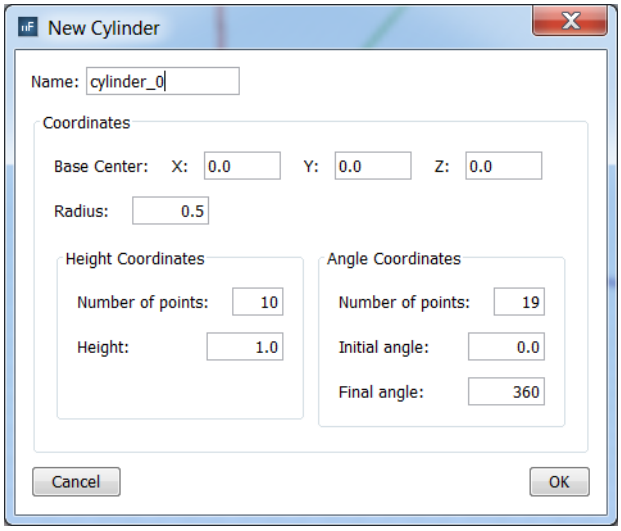


Figure 632: New Cylinder parameters

Click on **OK** button to insert the observation points from the cylinder, or abort the insertion by clicking on **Cancel** button.

External File

The user can also import the near field points from a text file. This file can then be imported by selecting **External File** into the **Geometry Types** combo-box. Note that this option imports a file of points, but does not preserve the near field type (i.e. every point is imported as a separate type). In order to preserve the near field type, the **Export file** and **Import file** must be used instead.

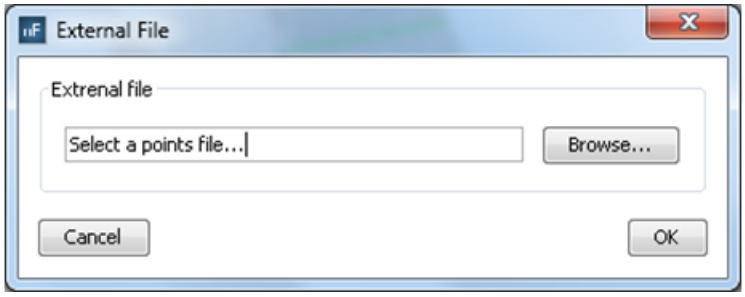


Figure 633: Open External File dialog

The external file must contain a list of points given by its **X**, **Y** and **Z** coordinates, with every point specified in separated lines.

Click on **OK** button to insert the observation points from the file, or abort the insertion by clicking on **Cancel** button.

From Curve

This option lets you select a line or curve of the main panel and sampling it as observation points. Before clicking on **Add** button having select the **From Curve** type, the curve has to be selected in the **Geometry** panel. As shown in the below figure, the following parametrs are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Number of points** to specify the number of samples generated along the curve.

Click on **OK** button to insert the observation points from the curve, or abort the insertion by clicking on **Cancel** button.

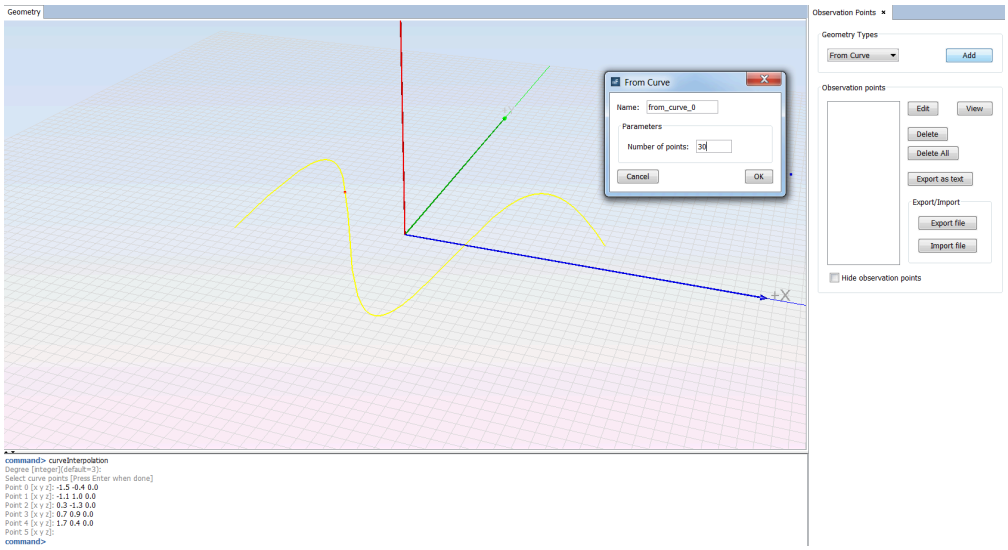


Figure 634: The next figure shows that the grid points are added to the Observation points list, and represented together the geometry as a dotted grid.

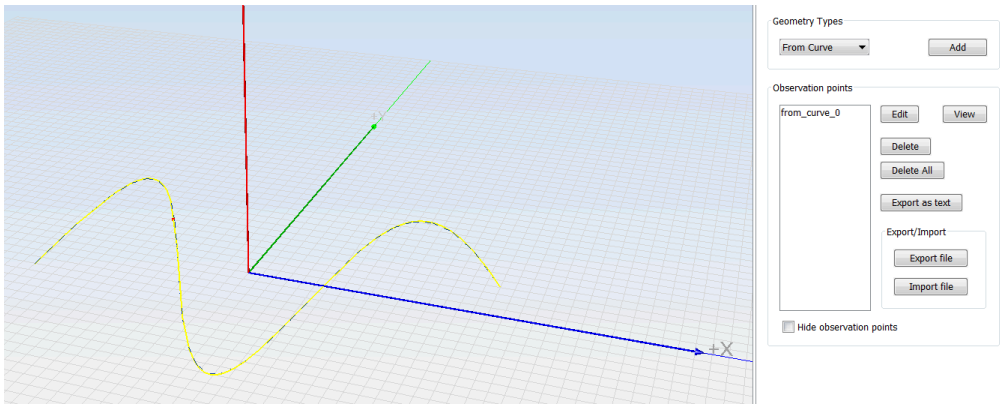


Figure 635: Observation points generated on the curve

From Surface

This option lets you select a surface of the main panel and sample it as observation points. Before clicking on **Add** button having select the **From Surface** type, the **surface** has to be selected in the **Geometry** panel. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Parameters** the generated observation entity will be a matrix of U x V points obtained by sampling the surface.
  - **U-Dimension** to specify the number of samples generated along the the U-Dimension of the surface.
  - **V-Dimension** to specify the number of samples generated along the the V-Dimension of the surface.

Click on **OK** button to insert the observation points from the surface, or abort the insertion by clicking on **Cancel** button.

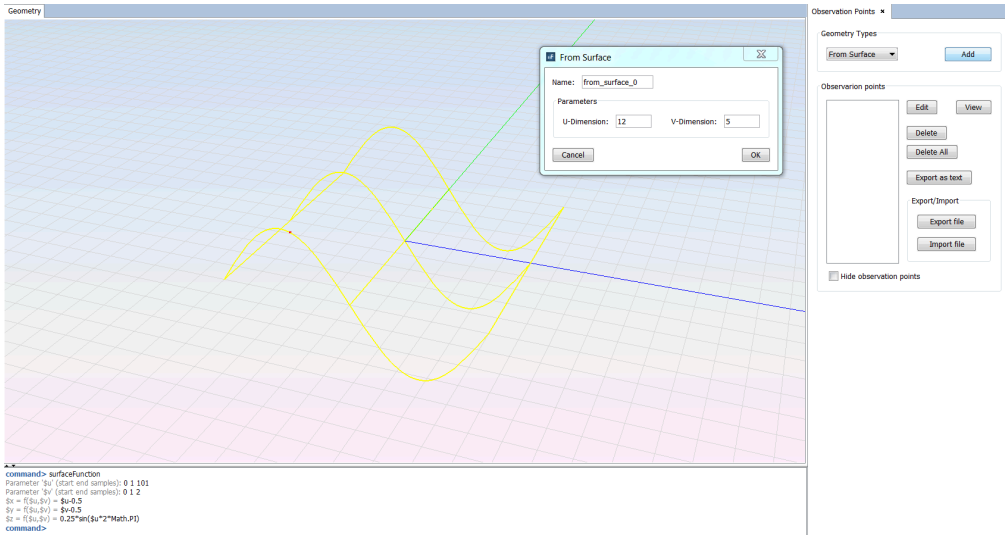


Figure 636: Adding a surface of observation points

The next figure shows that the grid points are added to the **Observation points** list, and represented together the geometry as a dotted grid.

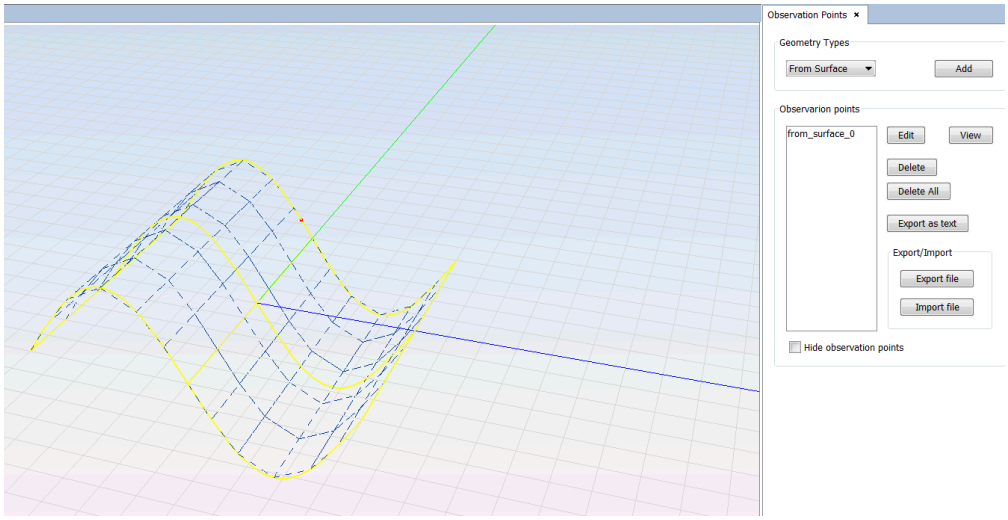


Figure 637: Observation points generated on the surface

Line

Observation points also may be generated along a virtual straight line sampled between its extremes. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Coordinates** the line is defined by its side points.
  - **Initial Point** first extreme of the straight line.
  - **Final Point** opposite extreme of the straight line.

Click on **OK** button to insert the observation points from the line, or abort the insertion by clicking on **Cancel** button.

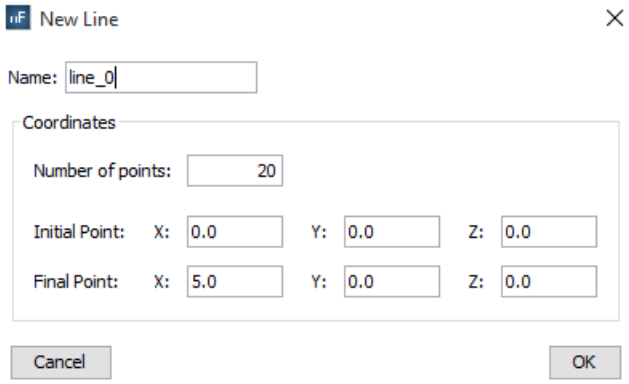


Figure 638: New Line parameters

Parallelogram

Use this option to generate a parallelogram of coplanar observation points. The parameters required to its shape are:

- Name to define the name of this type of observation points. The name must be different of the already defined ones.
- Coordinates the parallelogram is defined by three points that form its two main axis, so the generated observation entity will be a matrix of Side x Botton points obtained by sampling the parallelogram.
  - Side left axis is defined from **Point 1** to **Point 2**.
  - Bottom axis is defined from **Point 2** to **Point 3**.

Click on **OK** button to insert the observation points from the parallelogram, or abort the insertion by clicking on **Cancel** button.

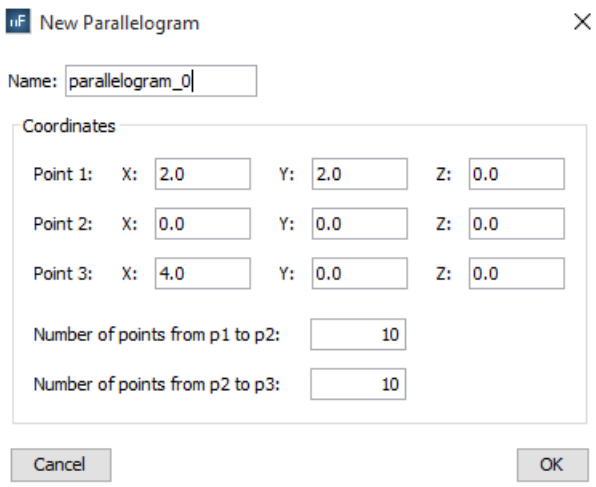


Figure 639: New Parallelogram parameters

Plane

In this window, the user can create a set of observation points laid out in a grid pattern. To do so, the user needs to specify several parameters:

- **Name** A short name that will be used by the user to identify the created set of observation points.
- **Constant axis** The normal axis of the plane that will contain the grid of observation points.
- **Coordinate value** Value of the coordinate of the axis specified in the previous option.

For each of the axes different than the one selected in the **Constant axis** drop-down list, the user needs to specify the following parameters:

- **Initial value** Initial value of the coordinate along each axis.
- **Total size** The total length of the grid of observation points along the axis.
- **Number of points** The number of observation points along the axis.

Note The spacing between each pair of consecutive observation points along an axis will be given by the formula **spacing = (total size) / (number of points - 1)**. This value will be shown next to the label "Distance btw. points".

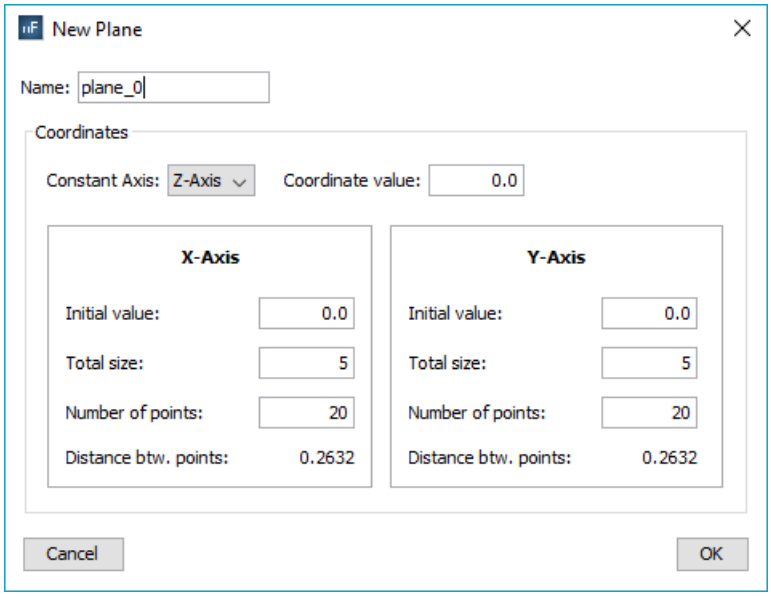


Figure 640: New Plane parameters

Point

Isolated observation points may be generated by using this option. The parameters to specify are:

- **Name** to define the name of the observation point. The name must be different of the already defined ones.
- **Coordinates** position of the observation point, specified by its X, Y and Z coordinates.

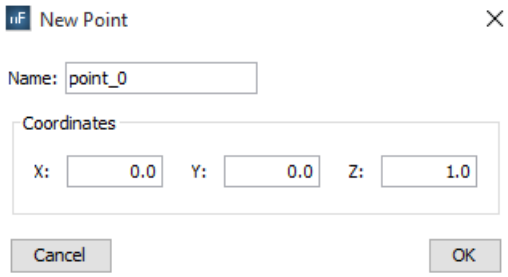


Figure 641: New Point parameters

Click on **OK** button to insert the observation point, or abort the insertion by clicking on **Cancel** button.

Sphere

The observation sphere may be defined as a full closed sphere or only an spherical sector specified. Multiple concentric spheres (or spherical sectors) may be defined within the same entity. The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Parameters**:
  - **Center Point** X, Y and Z coordinates where the sphere will be centered on.
  - **Radius** radius of the sphere. A radius sweep may be defined to generate multiple concentrical spheres.
    - **Initial Radius** radius of the first generated sphere, which is the only one if the sweep is not configured.
    - **Increment Radius** step between two adjacent spheres. It is only considered when the sweep is configured.
    - **Radius Samples** number of concentric spheres to be generated, it must be an integer number. The radius sweep is enabled when this parameter is higher than 1.
  - **Theta Coordinates** to set the spherical range where te observation points are generated, in theta dimension.
    - **Number of points** defines the number of samples to be taken in theta dimension along the specified angular range.
    - **Initial angle** defines the angular angle (in degrees) of the first theta point to be generated.
    - **Final angle** defines the angular angle (in degrees) of the last theta point to be generated.
  - **Phi Coordinates** to set the spherical range where te observation points are generated, in phi dimension.
    - **Number of points** defines the number of samples to be taken in phi dimension along the specified angular range.
    - **Initial angle** defines the angular angle (in degrees) of the first phi point to be generated.
    - **Final angle** defines the angular angle (in degrees) of the last phi point to be generated.

The total number of observation points to be generated in the sphere is (**Radius Samples**) x (**Number of Points**) $_{\text{Theta}}$ x (**Number of Points**) $_{\text{Phi}}$.

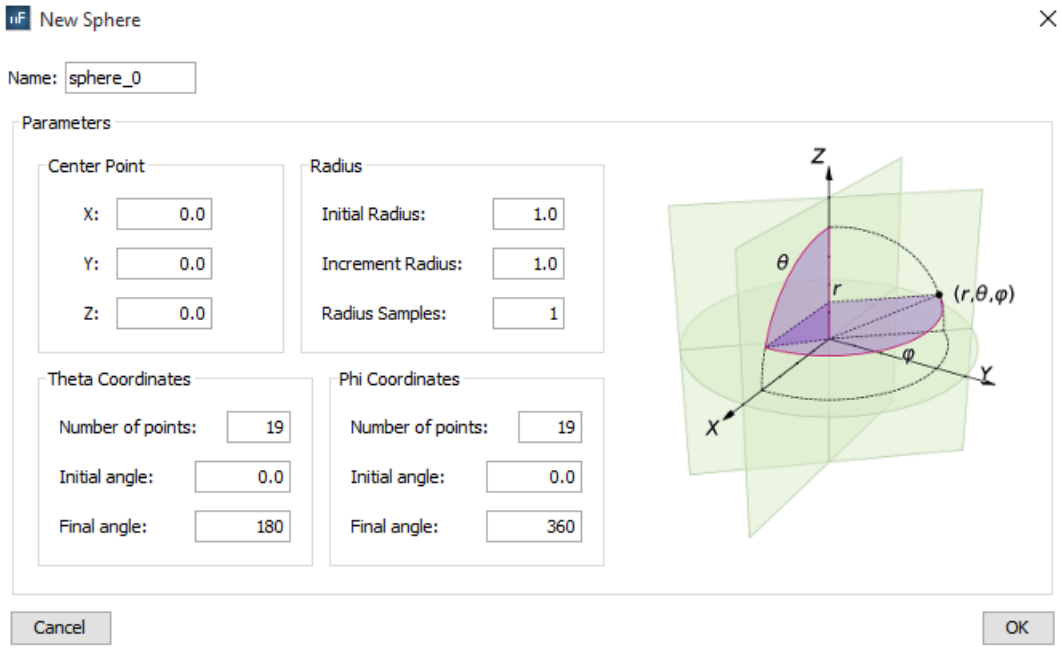


Figure 642: Spherical observation points.

Click on **OK** button to insert the observation points from the sphere, or abort the insertion by clicking on **Cancel**button.

7.8.3 Coupling

The Coupling submenu is available under the Output menu when the current simulation is a Coupling or MIMO simulation. This menu allows the user to configure the passive antennas of the simulation by adding individual antennas or creating arrays of passive antennas. The available options under the Coupling submenu are shown in the following figure:

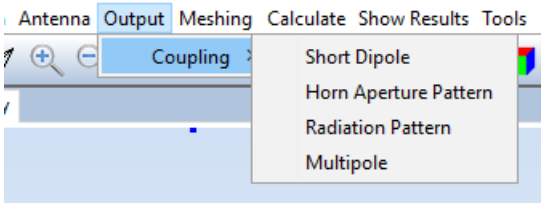


Figure 643: Coupling submenu

If the user selects the **Short Dipole** option, the following panel is shown:

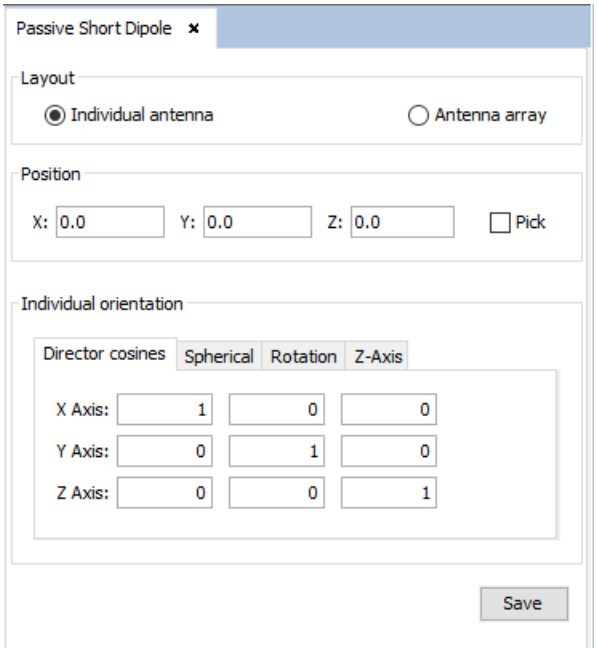


Figure 644: Passive short dipole panel

The user is able to select whether they want to add an individual short dipole or an array of short dipoles. In the first case, the user needs only to specify its location and orientation. In the second case, however, the user will need to specify the following parameters:

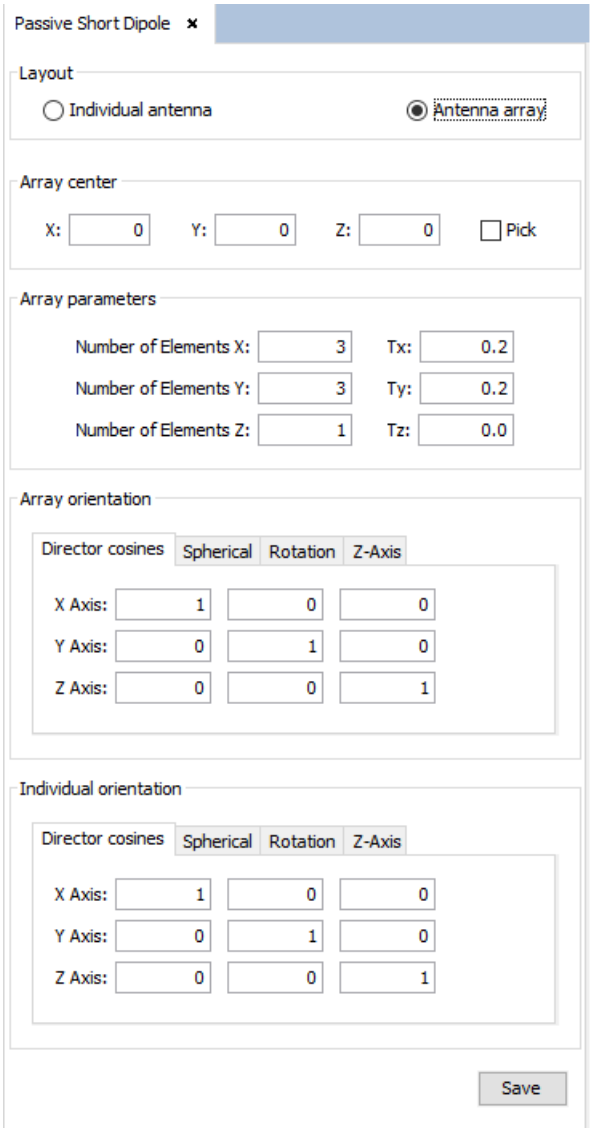


Figure 645: Passive short dipole panel (array layout)

- **Array center** The Cartesian coordinates of the geometrical center of the array.
- **Array parameters** The "Number of elements" determine the number of elements of the array in each of the Cartesian axes. The "Tx", "Ty" and "Tz" determine the spacing between two consecutive antennas along each axis.
- **Array orientation** Determines the axes used to calculate the position of each antenna in the array. This parameter does not affect the rotation of the individual antennas.
- **Individual orientation** Determines the rotation of each antenna in the array.

If the user selects the **Horn Aperture Pattern** option in the Coupling submenu, the following panel will appear. **Note** it is required to import at least one DIA file before adding an antenna of this type.

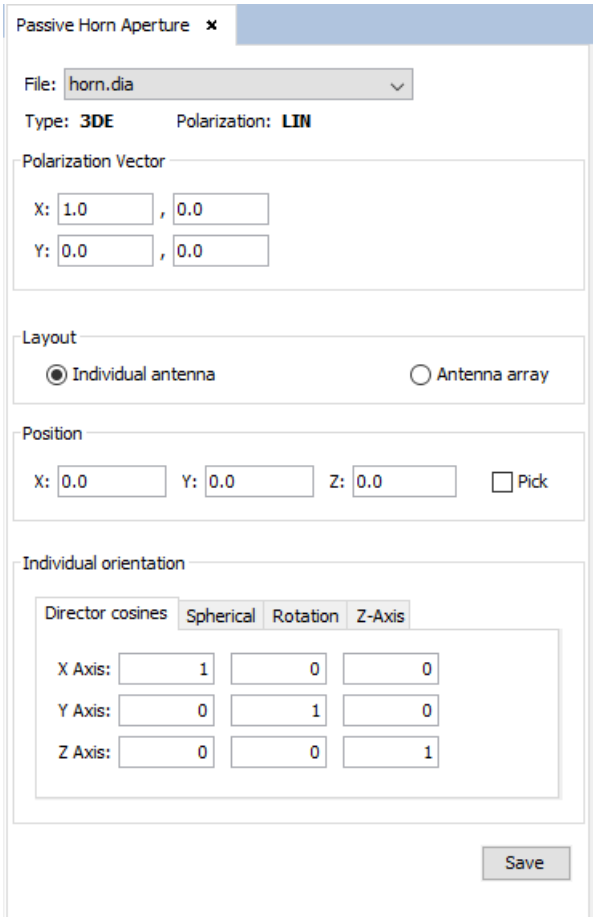


Figure 646: Passive horn aperture panel

In this panel, the user needs to select the DIA (radiation pattern) file to be used for the antenna feeding and set its polarization vector (given by the real and imaginary parts along each coordinate). When the user selects a DIA file, the type and polarization of the radiation pattern file will be shown.

As with the "Short Dipole" antenna type, the user can add arrays of horn aperture antennas. The parameters needed to configure an array of this type of antennas are the same as in the short dipole case.

When the **Radiation Pattern** option in the Coupling submenu is selected, the following panel will be shown:

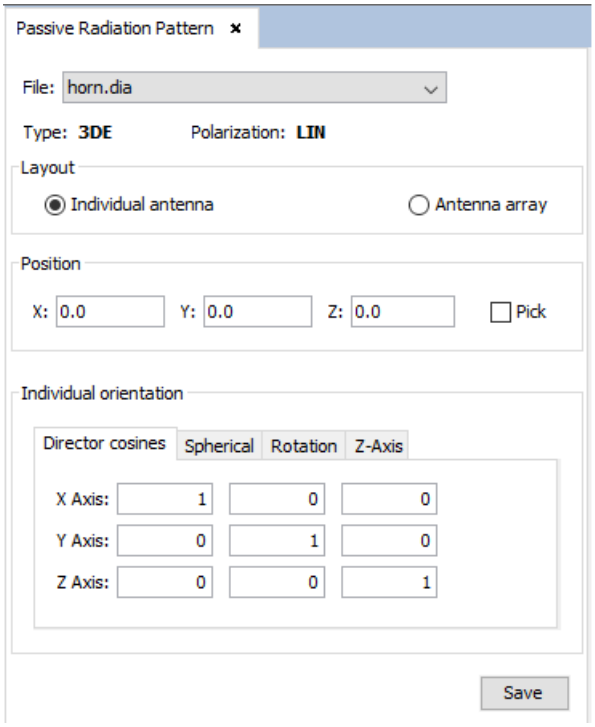


Figure 647: Passive radiation pattern panel

In this panel, the user will need to select the DIA file to be used for the antenna feeding. Like the Horn Aperture Pattern type, the user needs to import at least one DIA file to be able to select this option. Creation of antenna arrays is also possible and the parameters needed to create an arrays are the same as in the previous cases.

If the **Multipole** option in the Coupling submenu is selected, the following panel will be shown:

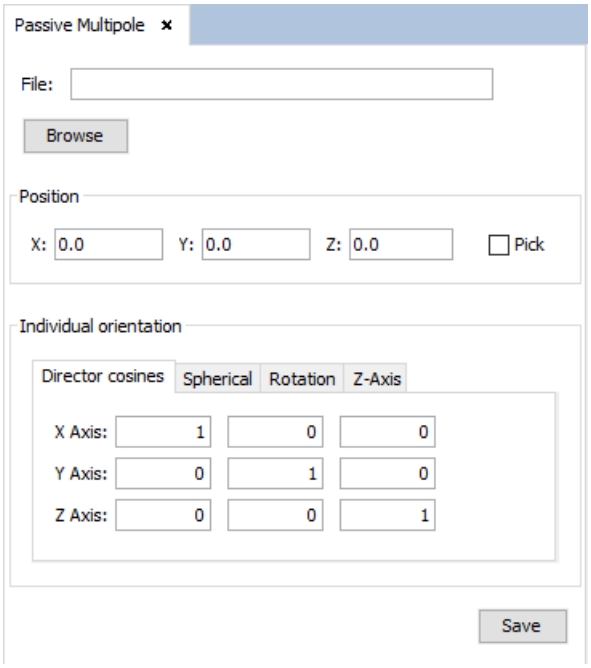


Figure 648: Passive multipole panel

In this panel, the user needs to select the SUJ file that contains the multipole definition. Unlike the previous antenna types, it is not possible to create arrays of passive multipole antennas.

When the user has finished adding and configuring an antenna or array of antennas, they need to press the "Save" button to confirm the changes. If all the information is correct, the created antennas will be shown on the screen, represented by magenta cones.

The user can add multiple passive antennas, as long as all the antennas are of the same type. This means that only one kind of passive antenna can be configured in a project at a given time. To change the passive antenna type, the user only needs to add an antenna of the type they want to use. If, when doing this, one or more passive antennas of a different type have been added before, the application will warn the user that those antennas will be deleted before adding the new passive antenna:

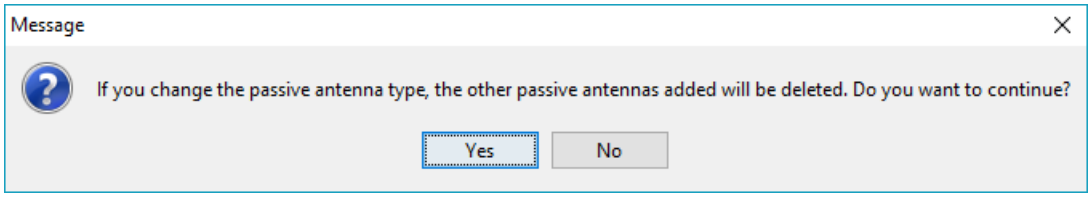


Figure 649: Warning when adding passive antennas of a different type

Pressing "Yes" will confirm the addition of the new antenna and therefore, the deletion of the previous passive antennas. Otherwise, the previous antennas will not be deleted and the creation of the new antenna will be aborted.

The currently configured type of passive antenna will be marked with a green tick next to the corresponding option in the Coupling submenu, as shown in the next image:

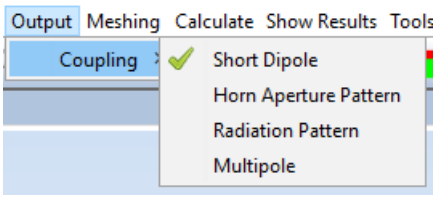


Figure 650: Green tick showing the current type of passive antenna

7.9 Meshing Menu

7.9.1 Ignore Edges

In some cases, the geometry designed for a case (such as an urban scenario case) features a plane that is used as a ground plane for the simulation. This plane is always a plane with finite dimensions and therefore the edges of the plane are taken into account when calculating reflected and diffracted rays using the GTD approach.

The "Ignore Edges" menu option under the "Meshing" menu allows the user to ignore specific edges of the geometry, so that these edges are not taken into account when performing the ray-tracing. In the example specified before, this option would allow a user to ignore the edges of the ground plane, so that this ground plane would behave like an infinite ground plane.

In order to specify the edges to be ignored, the user needs to select the "Ignore Edges" under the "Meshing" menu:

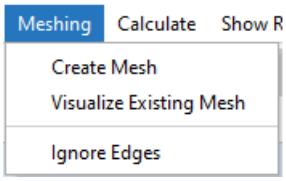


Figure 651: "Ignore Edges" menu option

The following panel will appear when this option is selected. Note that it may take a few seconds to detect all the edges in the geometry.

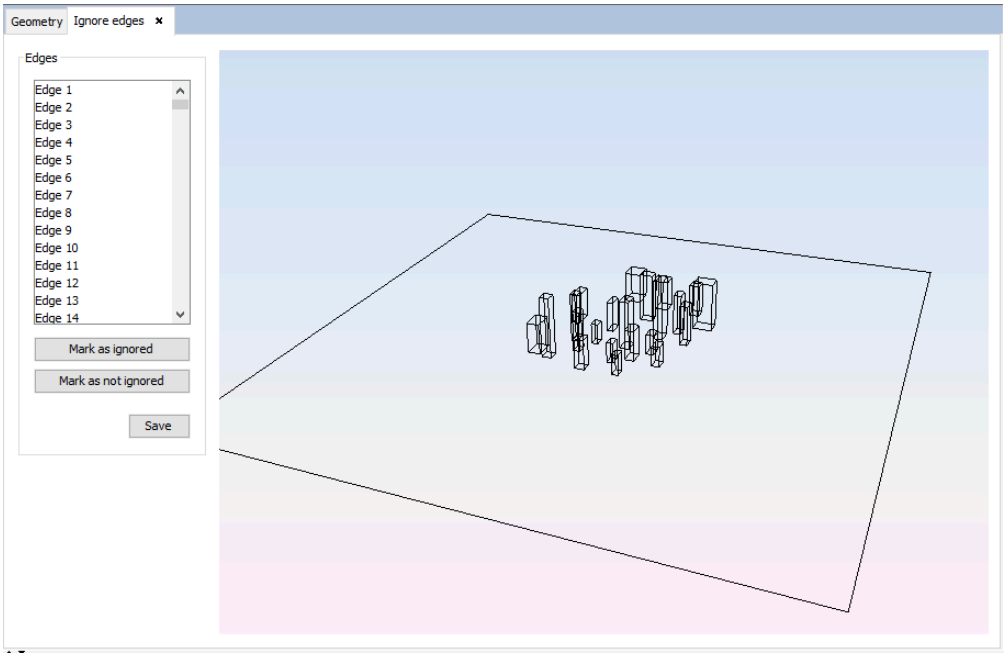


Figure 652: "Ignore Edges" panel

It is possible to select the edges to be removed by clicking on them in the geometry panel (holding the Ctrl key allows selection of many edges at once). It is also possible to select the edges on the list at the left side of the panel. When an edge is selected, it is drawn as a thicker yellow-colored line. The "Mark as ignored" button allows the user to ignore the currently selected edges. The ignored edges will be drawn as a dashed line in the geometry view, as shown in the following figure:

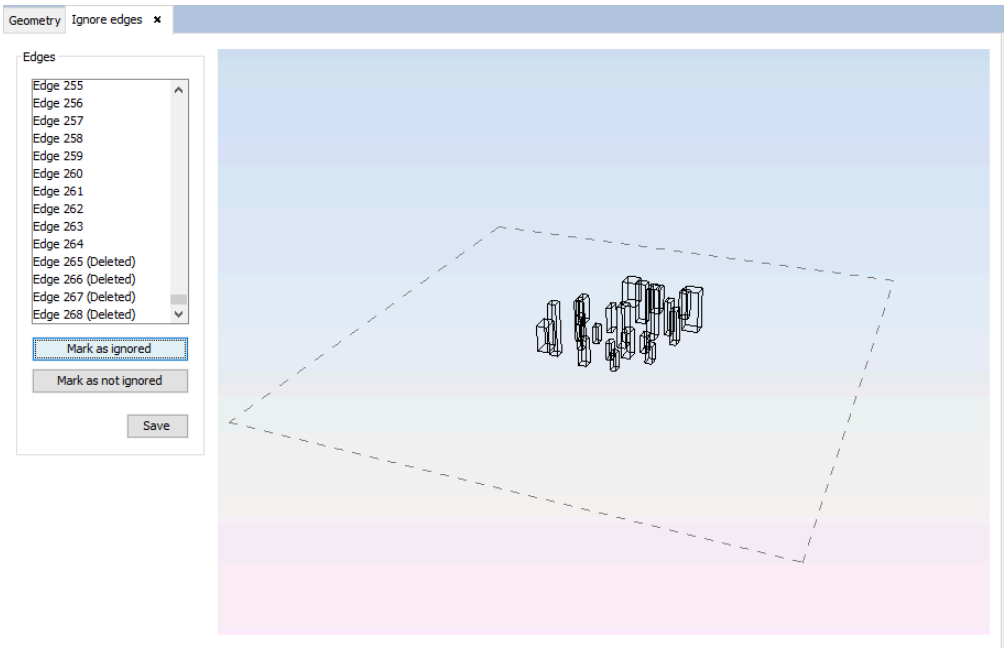


Figure 653: "Ignore Edges" panel, showing some ignored edges

In order to undo the "Mark as ignored" operation, the user needs to select the currently ignored edges and then press the "Mark as not ignored" button.

When the user has finished marking edges as ignored, they need to press the "Save" button to save the changes. The ignored edges will not be taken into account in subsequent simulations.

7.9.2 Create Mesh

When the "Meshing → Create Visibility Matrix" option is selected, the following panel is shown:

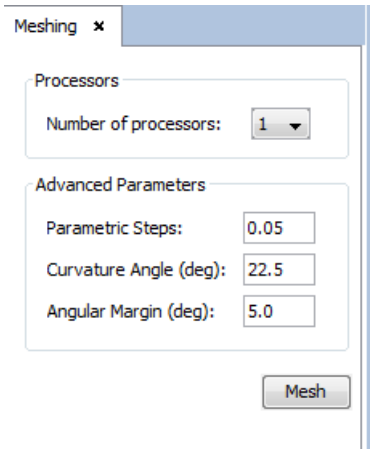


Figure 654: Meshing panel

The user may select the following:

- **Processors** the number of processors used to run the simulation.
- **Advanced Parameters** the contributions are previously evaluated on a parametric internal mesh of the geometry where the curved surfaces are simplified. The resolution of this mesh may be edited according to the below parameters, which are set by default with the recommended values.
  - **Parametric Steps** accuracy in the curvature evaluation, normalized. Decrease this parameter to get a higher precision in the parametric discretization.
  - **Curvature angle (deg)**maximum curvature considered on targets. It indicates the maximum curvature allowed in the input surfaces for the meshing step. Decrease this parameter to get a higher resolution mesh.
  - **Angular Margin (deg)** maximum deviation allowed around the curvature reference angle. The curvature discretization is achieved by an approximated iterative method, so an angular error around the previous parameter is allowed. Decrease this parameter to increase the precision in the curvature discretization.

This process may take a while (depending on the number of processors selected and the computing power of the computer running the simulation). While it is running, the status of the meshing process will be reported in the process log panel. From this panel we can abort the running meshing process, as well as save the log to a text file.

7.10 Calculate Menu

When the meshing process is finished, the user can start the simulation. It is advised for the user to verify that all the simulation parameters are correct. If there is any problem that prevents the simulation from running, it will be notified to the user.

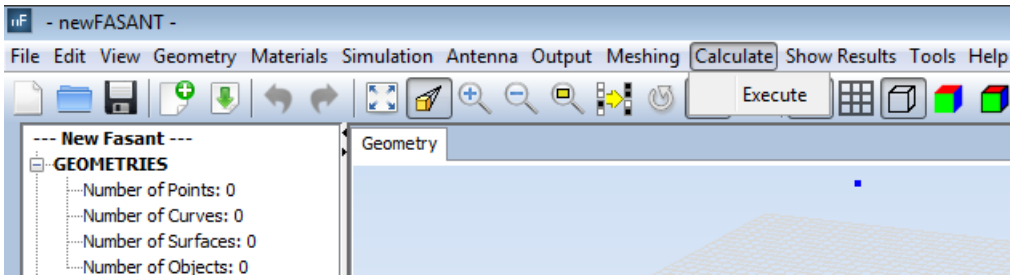


Figure 655: Calculate menu

Selecting the Calculate → Execute option from the menu will show the following panel:

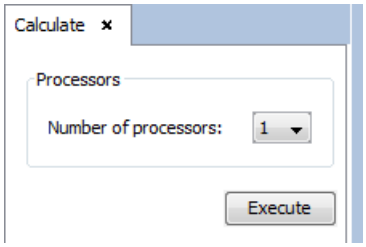


Figure 656: Calculate options panel

In this panel, the user needs to select the number of processors that will run the simulation. When the user presses the Execute button, the simulation will begin. The Process Log panel will show the status of the running simulation:

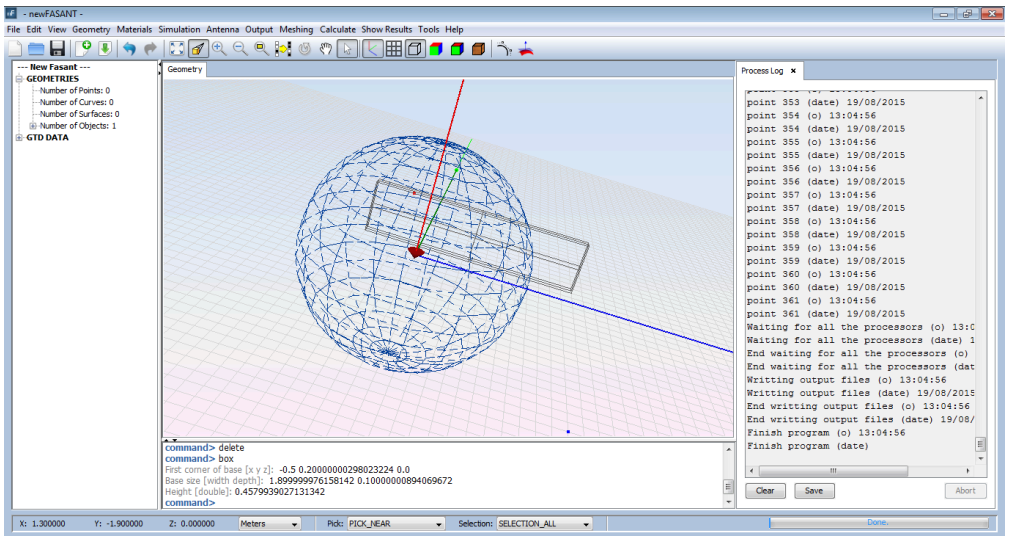


Figure 657: Process log showing the status of the simulation

It is possible to abort the simulation at any time, but only when it is running.

7.11 Show Results Menu

This menu allows the user to visualize the results of the simulation. The results are stored in external files that the user can access to obtain the information corresponding to that simulation.

Most of the plots show 2-D curves of a result field versus an input parameter. This type of graphics shares additional features of zoom, axis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

7.11.1 Common Options

View Ray

This option allows the user to view the rays emitted by the active antennas in the simulation.

This command plots the rays emitted by the active antennas in the simulation only if the check box **Output Ray-Tracing** in the **Simulation** panel has been selected.

When initially open, the panel will display the rays for a particular step, order, antenna and observation point or direction. All the effects type are selected by default. The user can change:

Ray effect
: it is possible to select one of these options (direct, reflected, transmitted, creeping and diffracted) or all of them.

Ray selection

- **Order** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Antenna** select one of the antennas that were previously added to the project.
- **Point/Theta phi** select a particular observation point or direction.

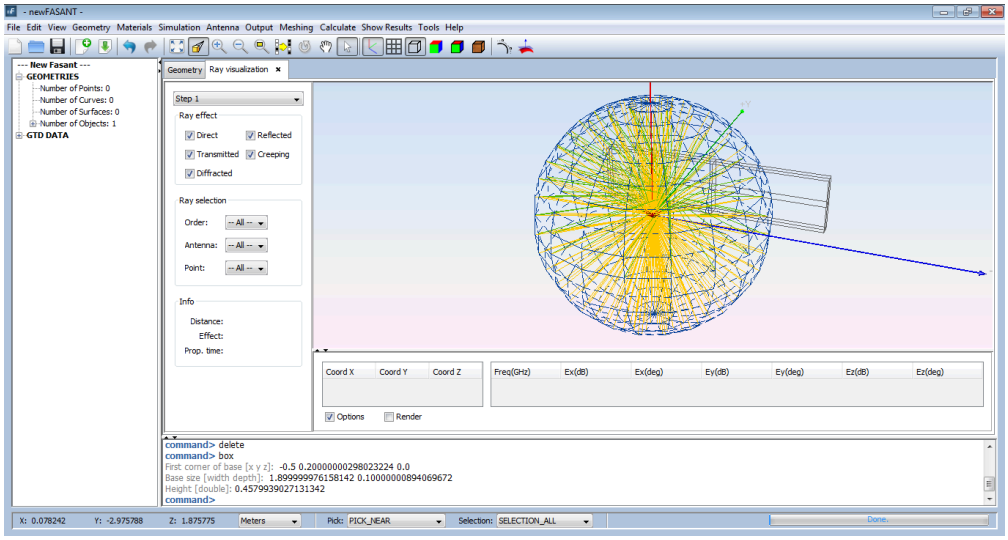


Figure 658: Ray visualization panel

Selecting a ray will highlight it in yellow color and its information will be shown in the Info panel (length of the ray, type of effect and propagation time) at the bottom. This information is composed of frequency and the modulus and argument for each of its components.

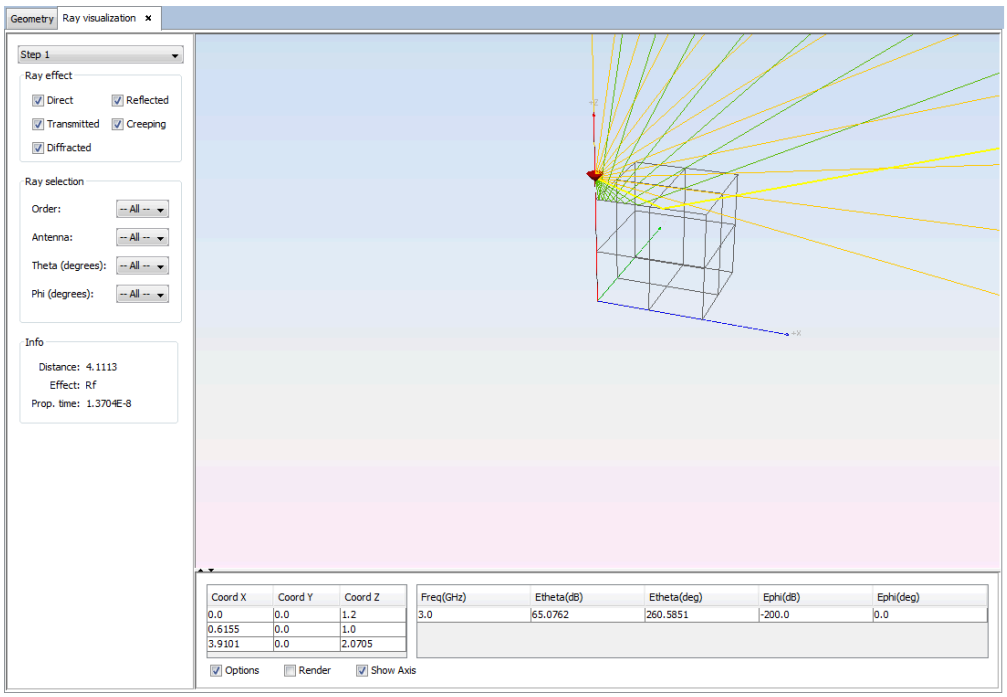


Figure 659: Ray information

View Text Files

This option shows the simulation results as a text file. First, a panel will appear after selecting this option. Selecting this option will display a panel where the user should select an antenna, a component,

the step and the order. To use this section, select a parametric step from the list, a component, one of the orders shown in the list and a step to get the results from and click on "OK" to load the data. A table will be presented with the values for the electrical field. To save the result click on the "Save as..."button below the table.

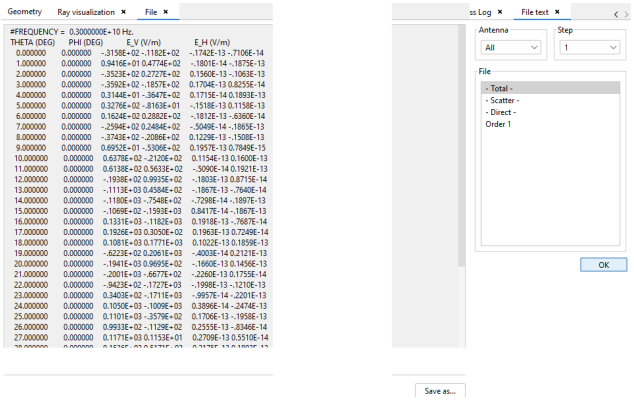


Figure 660: Showing the result text file

7.11.2 Far Field

Using this menu it is possible to analyze the Far Field through the observation directions previously defined in the Output section.

The following options are available:

- View Cuts: display a plot that relates the amplitude against a Theta cut or a Phi cut.
- View Cuts by Frequency: displays a plot that relates the amplitude against a frequency sweep.
- View Cuts by Step: displays a plot that relates the amplitude against a parametric step.

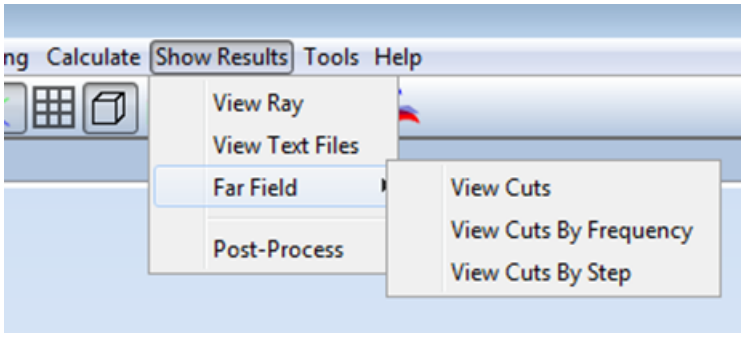


Figure 661: Results menu with a far field simulation

View Cuts

This command plots the amplitude and phase of the far field computed for each angular cut of the simulation geometry. Several cuts and frequencies can be plotted in the same graphic considering different effects and antennas. To plot a graph, the user can select the field component, the frequency,

the step, the cut, the order and the antenna. A new graph will be plotted with the button **Add Series**. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **Remove Series**. The display option also allows changing the color of the series and display points. The buttons **Import Series** and **Export Series** are used for importing and exporting the selected series in **List of Series** to a text file.

This panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point, against every possible frequency and antenna that has been simulated. More series can be added, by selecting a component, step and spherical coordinates in the control panel that appears on the left side of the plot.

- **Component**: the results may be plotted using different polarization and its components:
  - **Lineal**: Etheta, Ephi, Etotal components.
  - **Gain (dBi)**: Theta Gain, Phi Gain, Total Gain components

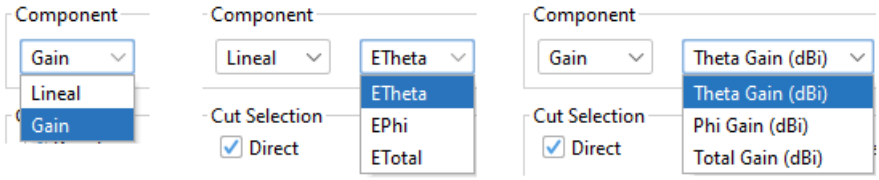


Figure 662: Component group

- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Frequency**: frequencies that have been simulated.
- **Point**: select a particular observation point.
- **Order**: select a particular order.
- **Antenna**: select a particular antenna.

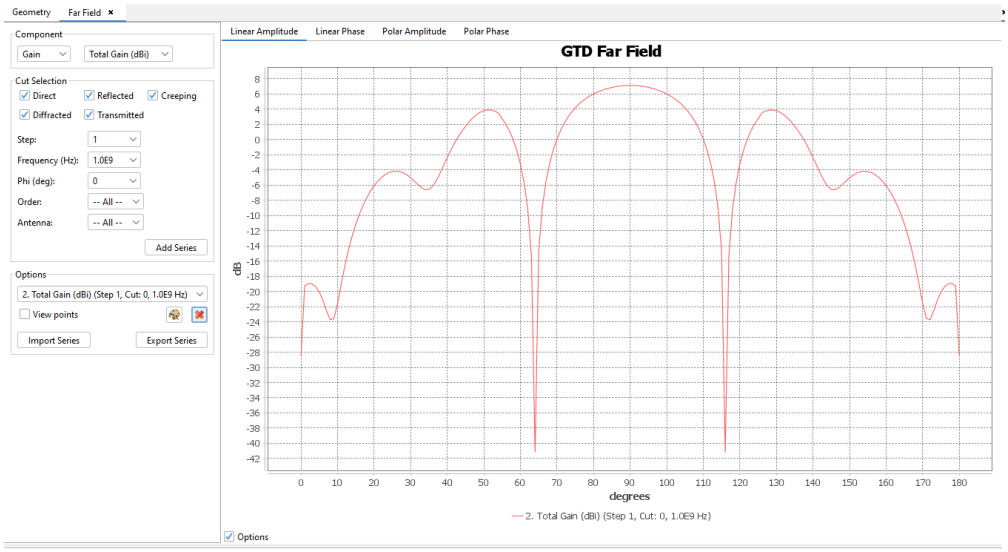


Figure 663: View cuts panel (linear representation)

The user can also draw a polar plot with the same options used in the previous window.

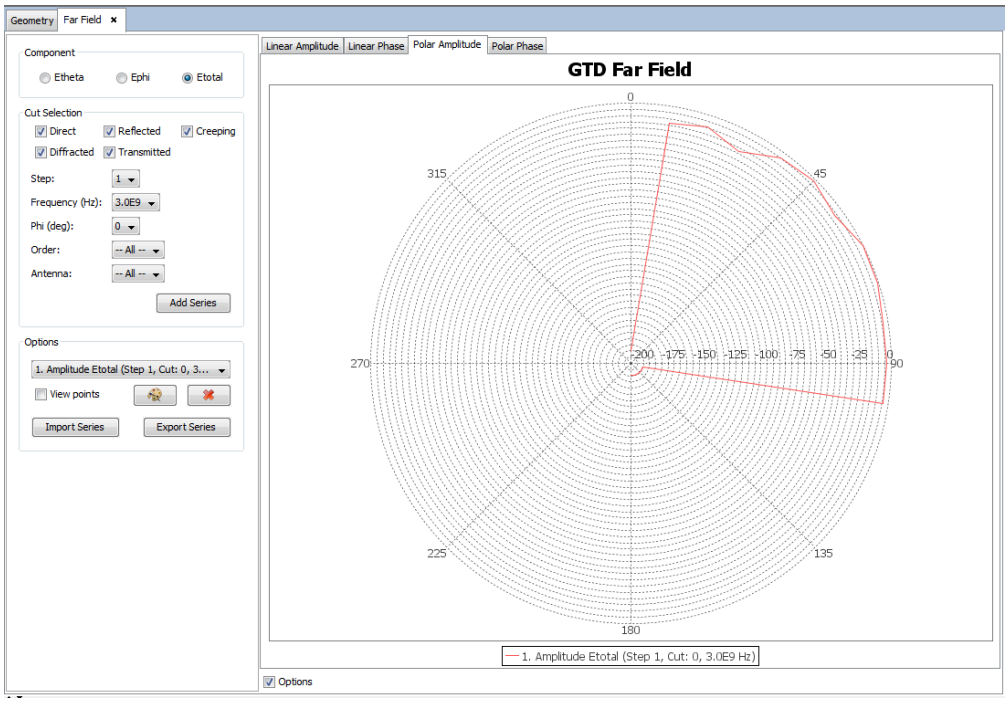


Figure 664: View cuts panel (polar representation)

View Cuts by Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, order, antenna, cut and point in the sweep, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation

Parameters, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the **Show Results** > **Far Field** > **View Cuts By Frequency** menu, the panel presented in the figure below will appear.

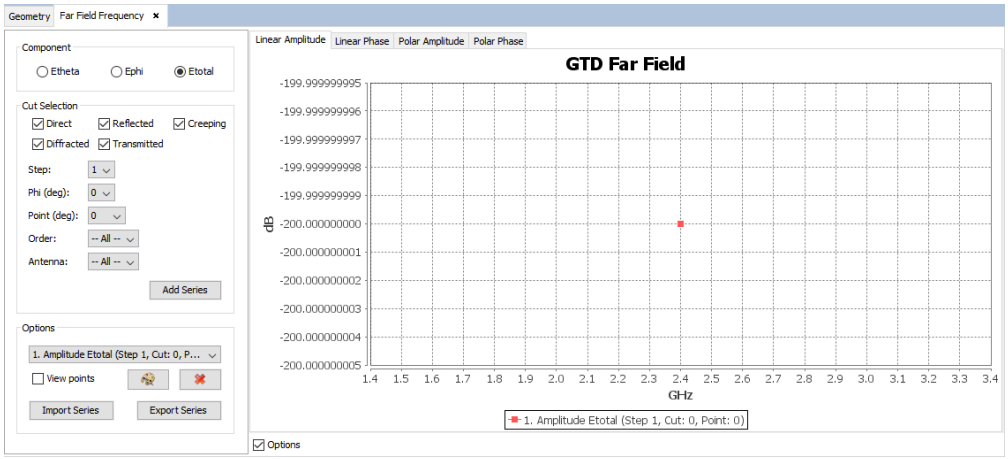


Figure 665: View cuts by frequency panel

Similar to the behavior of the standard Far Field chart, this panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point, against every possible frequency and antenna that has been simulated. More series can be added, by selecting a component, step and spherical coordinates in the control panel that appears on the left side of the plot.

- **Component**: the results may be plotted using different polarization and its components:
  - **Lineal**: Etheta, Ephi, Etotal components.
  - **Gain (dBi)**: Theta Gain, Phi Gain, Total Gain components

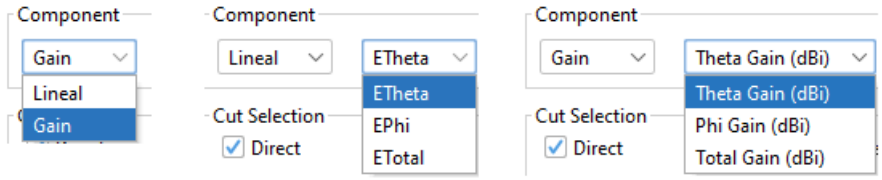


Figure 666: Component group

- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Frequency**: frequencies that have been simulated.
- **Point**: select a particular observation point.
- **Order**: select a particular order.
- **Antenna**: select a particular antenna.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Cuts by Step

This kind of plot lets the user analyze far field measurements made by the simulation process for a particular component, frequency, order, antenna, theta or phi cut and observation point against the parametric steps defined in the project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the **Show Results** > **Far Field** > **View Cuts by Steps** menu, the panel presented below will appear:

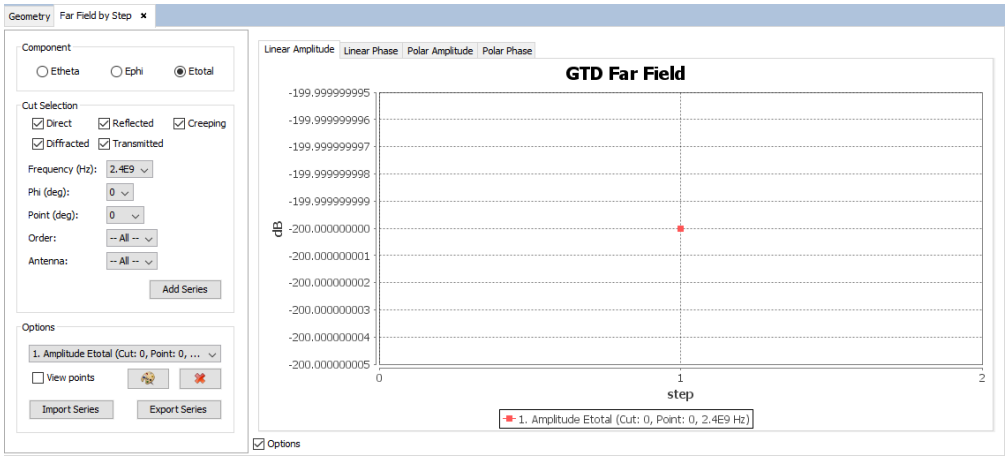


Figure 667: View cuts by step panel

Similar to the behavior of the standard Far Field chart, this panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, frequency, order, antenna, and observation point, against every possible step that has been simulated. More series can be added, by selecting component, frequency, order, antenna and spherical coordinates in the control panel that appears on the left side of the plot.

- **Component**: the results may be plotted using different polarization and its components:
  - **Lineal**: Etheta, Ephi, Etotal components.
  - **Gain (dBi)**: Theta Gain, Phi Gain, Total Gain components

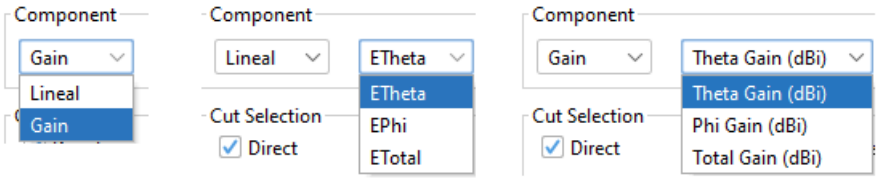


Figure 668: Component group

- **Frequency**: frequencies that have been simulated.
- **Point:**: select a particular observation point.
- **Order**: select a particular order.
- **Antenna**: select a particular antenna.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

7.11.3 Near Field

Using this menu it is possible to analyze the amplitudes in the Near Field through the observation directions previously defined in the Output section.

The following options are available:

- View Cuts: display a plot that relates the amplitude against the observation cut.
- View Cuts by Frequency: displays a plot that relates the amplitude against a frequency sweep.
- View Cuts by Step: displays a plot that relates the amplitude against a parametric step.
- Near Field: displays the numeric results using a colored map.
- Delay Process: displays the results of the delay process.
- Power Process: displays the results of the power process.

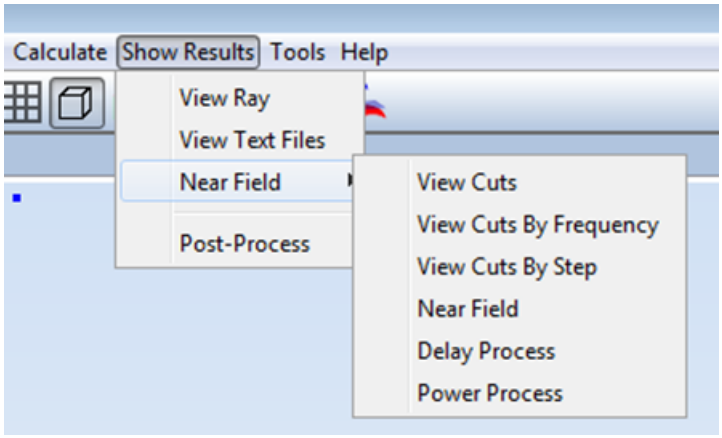


Figure 669: Show Results menu with a near field simulation

View Cuts

This command plots the amplitude and phase of the near field computed for each observation point of the simulation geometry. Several observation points (such us lines, planes and so on) and frequencies can be plotted in the same graphic. To plot a graph, the user can select the field component, the step, the frequency, the observation, the order and the antenna. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a text file.

This panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, frequency, and observation point, against every possible frequency that has been simulated. More series can be added, by selecting a component, step, observation, order and antenna in the control panel that appears on the left side of the plot.

- **Component**: the field component to inspect. Select one of the possible values Ex, Ey, Ez, Etotal.
- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Frequency**: select one frequency to inspect.
- **Observation**: select one of the observation series.
- **Order**: select a particular order.
- **Antenna**: select a particular antenna.

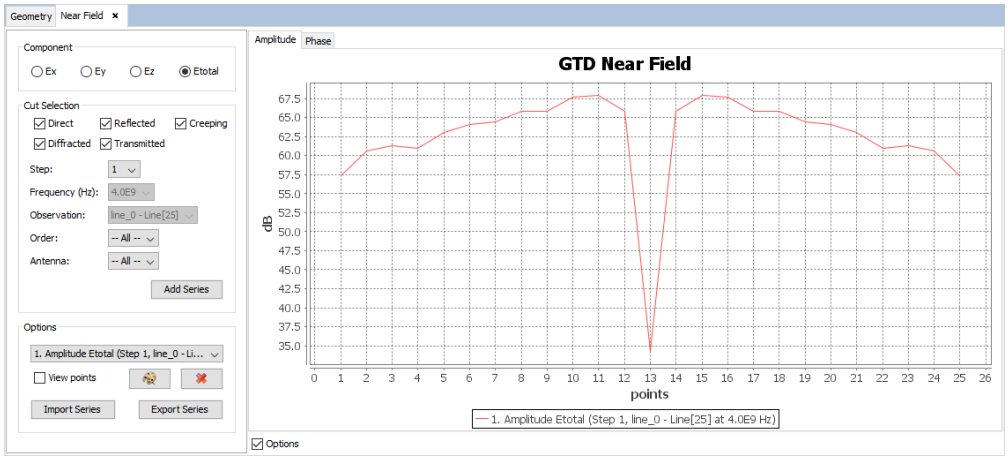


Figure 670: Near Field View Cuts panel

View Cuts by Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, observation point, order and antenna, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation Settings, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the Show Results > Near Field > View Cuts By Frequency menu, the panel presented in the figure below will appear.

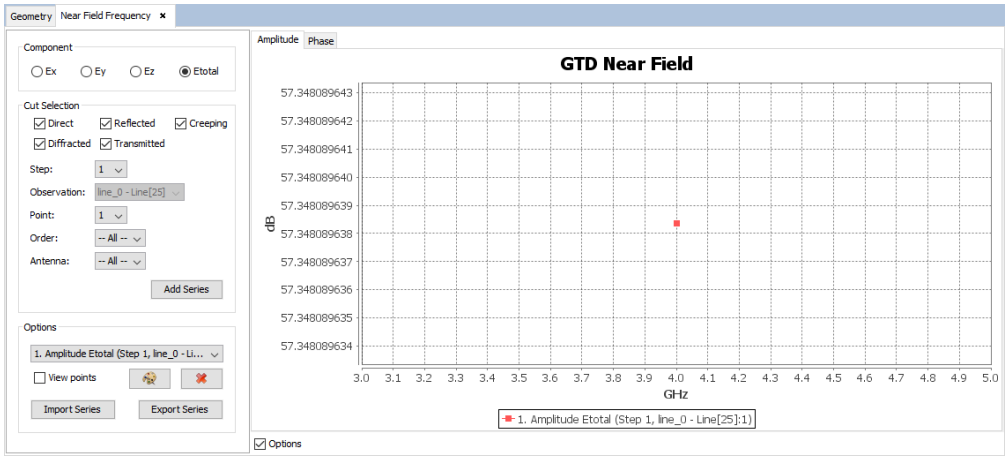


Figure 671: View cuts by frequency panel

Similar to the behavior of the standard Far Field chart, this panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, frequency, and observation point, against every possible frequency that has been simulated. More series can be added, by selecting a component, step, observation, order and antenna in the control panel that appears on the left side of the plot.

- **Component**: the field component to inspect. Select one of the possible values Ex, Ey, Ez, Etotal.
- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Observation**: select one of the observation series.
- **Point**: select the observation point.
- **Order**: select a particular order.
- **Antenna**: select a particular antenna.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Cuts by Step

This kind of plot lets the user analyze RCS measurements made by the simulation process for a particular component, frequency, theta or phi cut and observation point against the parametric steps defined in the project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the Show Results > Far Field > View Cuts by Steps menu, the panel presented below will appear:

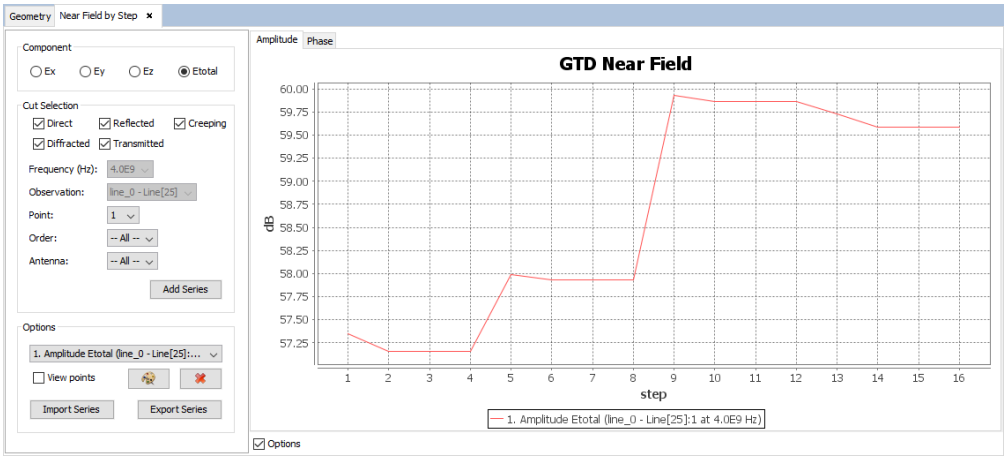


Figure 672: View cuts by step panel

Similar to the behaviour of the standard Far Field chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, frequency, and observation point, against every

possible frequency that has been simulated. More series can be added, by selecting component, step, observation, order and antenna in the control panel that appears leftside to the plot.

- **Component:** the field component to inspect. Select one of the possible values Ex, Ey, Ez, Etotal.
- **Frequency:** select one frequency to inspect.
- **Observation:** select one of the observation series.
- **Point** select one of the observation points.
- **Order:** select a particular order.
- **Antenna** select a particular antenna.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

Near Field

When this option is selected, the following panel will appear:

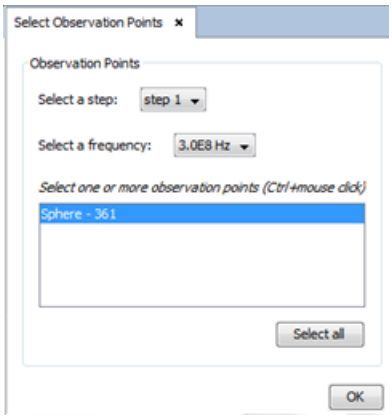


Figure 673: Near field panel

In this panel, the user can select a step, a frequency (from the set of sampled frequencies) and a set of observation points (holding the Ctrl key, the user can select more than one item from the list). When the user presses the OK button, the following window will be opened, showing a 3D diagram of the near field values on the selected observation points:

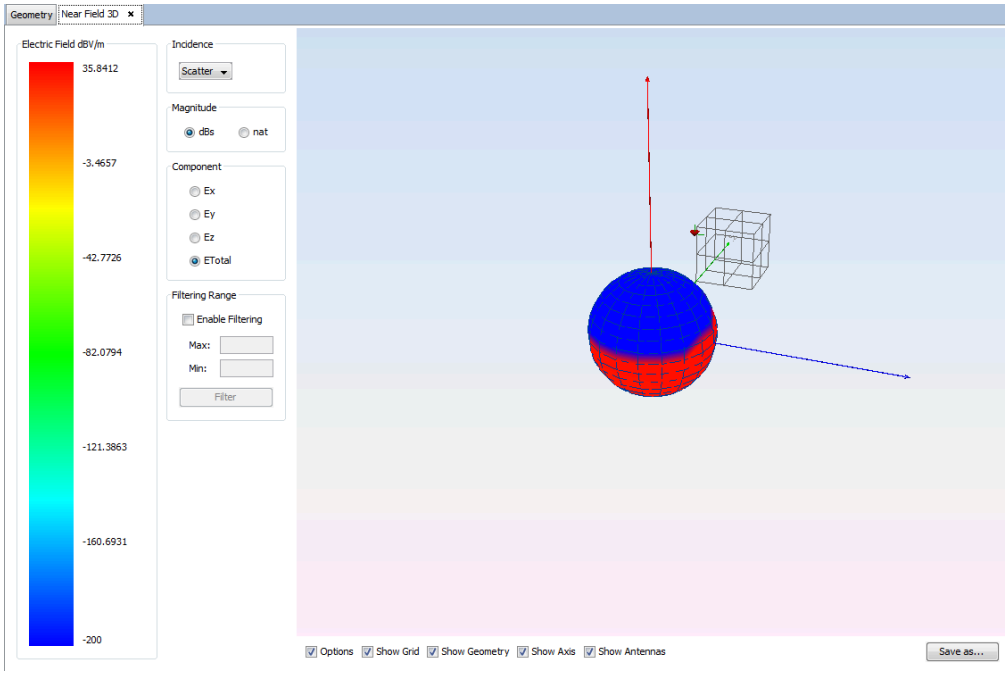


Figure 674: 3D diagram showing the near field values.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component and field in dBo natural magnitude. Different values can be visualized by changing the parameters in the control panel.

- **Incidence**: the type of field to inspect. Select one of the possible values Direct, Scatter, Total.
- **Magnitude**: dB o natural.
- **Component**: the field component to inspect. Select one of the possible values Ex, Ey, Ez, Etotal.
- **Filter**: enable the filter check box and select a particular maximum and minimum to filter the values that will be plotted.

Delay Process

The **Delay Process** is a post-process computation that shows how a given signal is distorted in the time domain because of its environment. The signal, which may be set with different shapes, is generated by every **antenna source** available in the simulation, and the delay process may be computed in multiple near field observation points.

After running a **Delay Process** type simulation, click on **View Delay Process** to open the **Delay Process** window, and the following panel is shown on the right side:

Figure 675: Delay process configuration panel

This panel contains several options for plotting the Delay Process:

- **Component:** specify the component considered for the **Delay Process** computation (Ex, Ey or Ez).
- **Selection** to define the spherical far-field ranges where the **Delay Process** will be computed. The first row defines the theta interval and the second row the phi one.
  - **Step** select the step of the parametric simulation to be consider. If it is not a parametric simulation, this value will be 1.
  - **Frequency** frequency of the simulation.
  - **Point** select the observation point index to consider in the Delay Process.
- **Signal** the original signal may be generated by using different patterns. All of them are configured in the time domain.
  - **Dirac Delta** only a pulse is generated in the initial time.
  - **CDMA** a random distribution of pulses centred with positive (high) an negative (low) levels is generated. Note that a low or high level is considered as a **Pulse**.
  - · **Length** number of pulses to be generated.
  - · **Pulse Width** time duration of each **Pulse**, in microseconds.
  - **Sinusoid** a sinusoidal signal is generated.
  - · **Carrier Freq.** frequency of the sinusoidal signal, in MHz.
  - · **Phase** initial phase of the sinusoidal signal, in degrees.
  - · **Duration** total length of the generated sinusoid in the time domain, in microseconds.
  - **Tukey Envelope** a Gaussian signal enveloped by a rectangular pulse is generated.

- · **Alpha Parameter** define the slope of the Gaussian curve. The higher is this parameter, the more enveloped by the rectangular pulse is the generated signal.
- · **Pulse Width** define the total length of the generated signal in the time domain, in microseconds.

- **Time** this section specifies the observation time range.
- *Initial*
  first observation instant in the time domain, in microseconds.

  End
  last observation instant in the time domain, in microseconds.

  Samples
  number of instants to evaluate de**Delay Process**that are equally equispaced in the time domain.

Click on **OK** button to plot de **Delay Process** results.

Only when the **CDMA** signal has been selected, the window represented below aks the selection of the **Correlation Signal** to be plotted with the following options.

- **Input (x[n])** the original CDMA signal is represented.
- **Output (y[n])** the received CDMA signal is represented **Correlation (c[n])** the correlation on the received signal is represented.

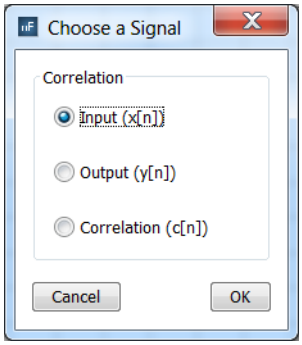


Figure 676: Selection of signal to be plotted with CDMA

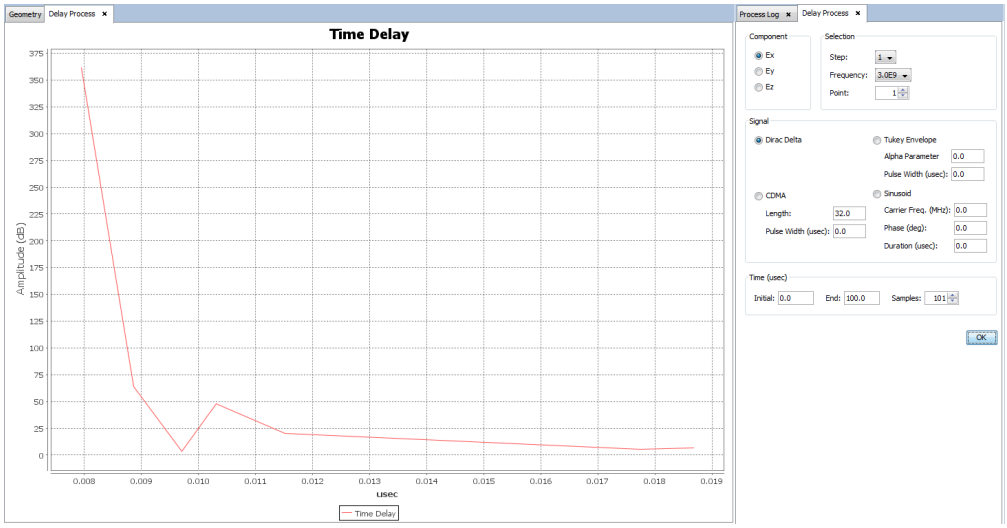


Figure 677: Delay Process generated with a Dirac Delta signal

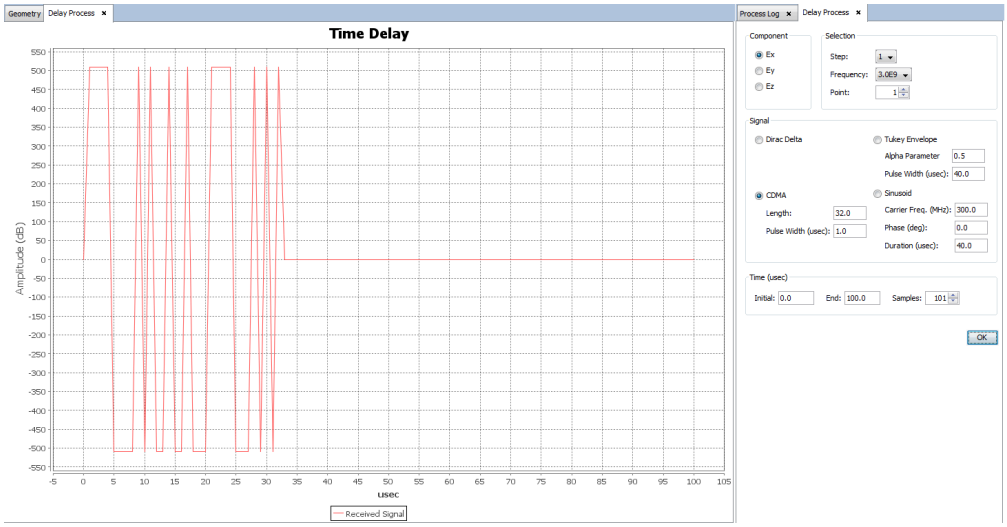


Figure 678: Delay Process generated with a CDMA (output y[n]) signal

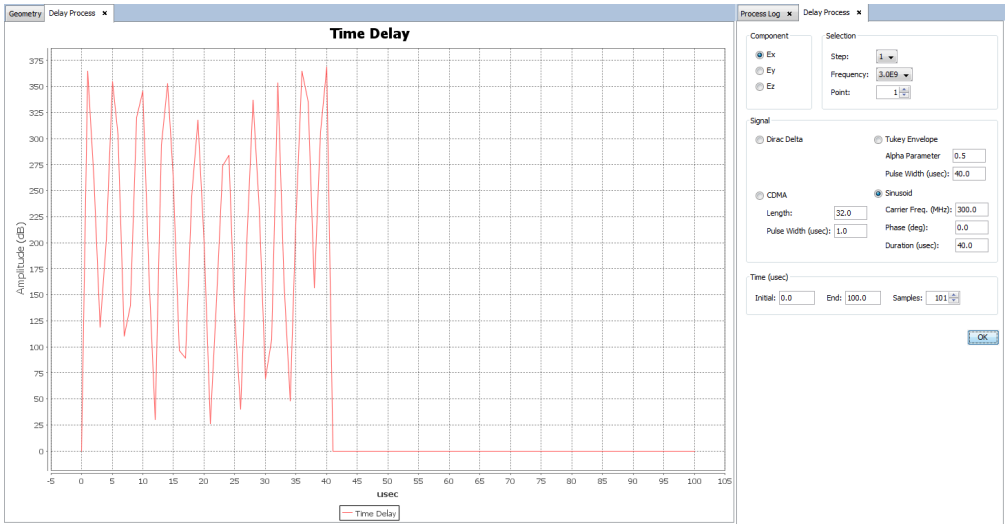


Figure 679: Delay Process generated with a Sinusoid signal

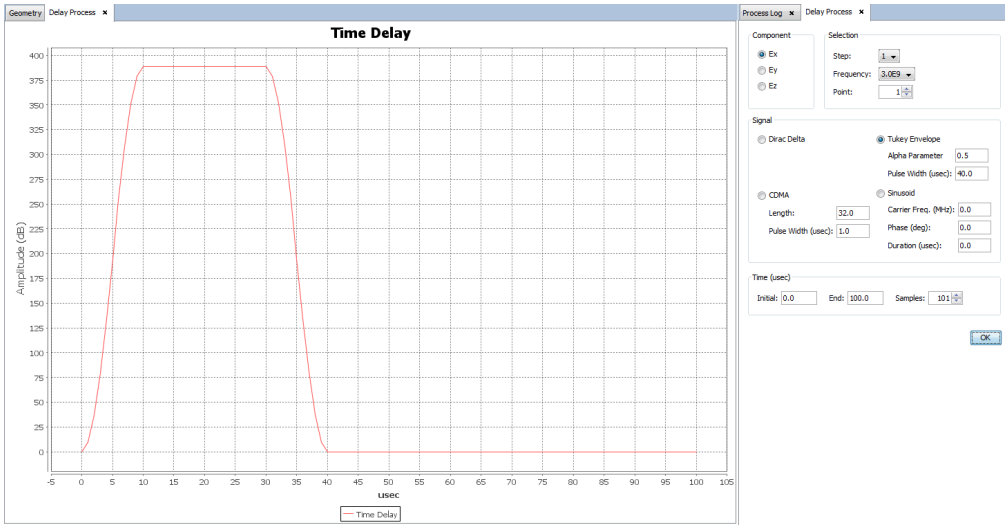


Figure 680: Delay Process generated with a Tukey Envelope signal

Power Process

When this option is selected, the following panel will appear:

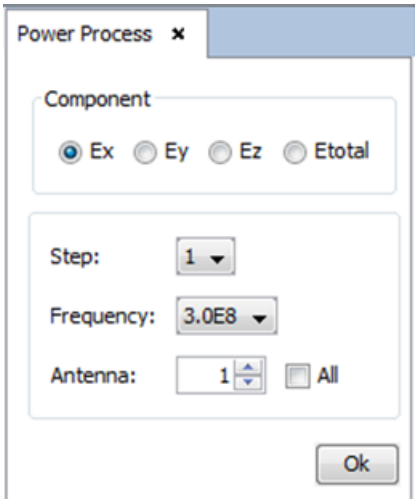


Figure 681: Power process configuration panel

In this panel, the user can configure the component to be shown, the step, the frequency and the antenna to show the results for. Pressing the OK button will show the chart.

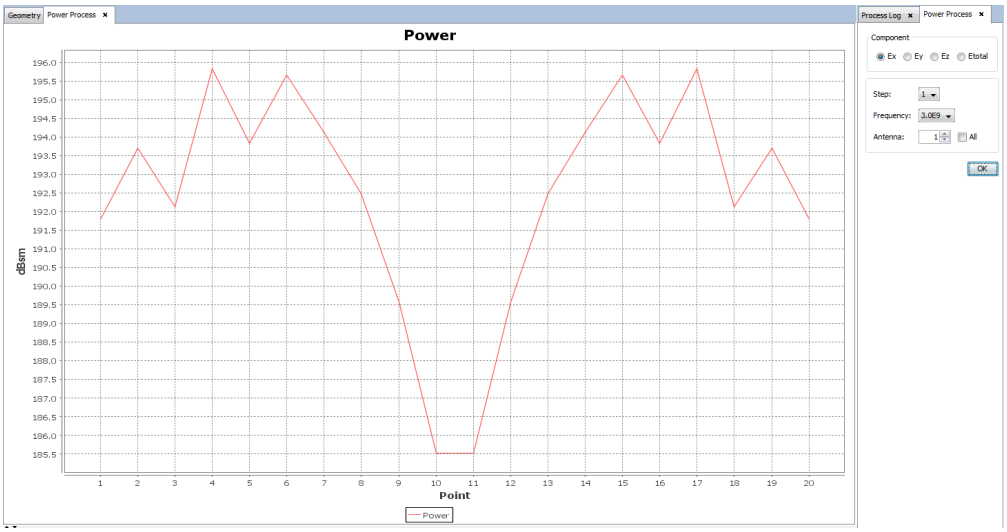


Figure 682: Power Process generated

7.11.4 Doppler Spectrum

Using this menu it is possible to analyze the Doppler spectrum through the observation point previously defined in the Output section.

The following options are available:

- Chart display a plot that relates the Doppler spectrum.
- Text Files displays the numeric results obtained in the simulation process.

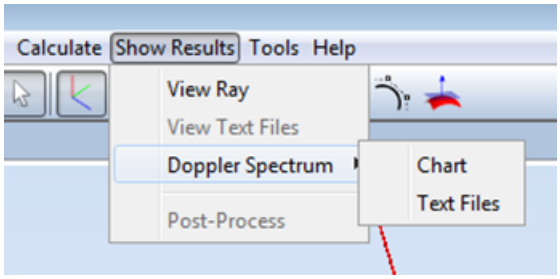


Figure 683: Showing the result text file

Chart

This command plots the Doppler frequency shift computed. Several cuts and frequencies can be plotted in the same graphic. To plot a graph, the user can select the field component, step, antenna and observation point. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a text file.

This panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, antenna, observation point and step. More series can be added, by selecting component and step in the control panel that appears on the left side of the plot.

- **Component**: the field component to inspect. Select one of the possible values Ex, Ey, Ez, Etotal.
- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Antenna group**: The antenna to show the Doppler spectrum results for. The user can select a specific antenna group or they can choose to show the results for all antennas.
- **Observation group**: The observation points group that contains the point that the user wants to see the results for.
- **Observation point**: The observation point to visualize the results for. This drop-down list shows the observation points included in the selected observation group.

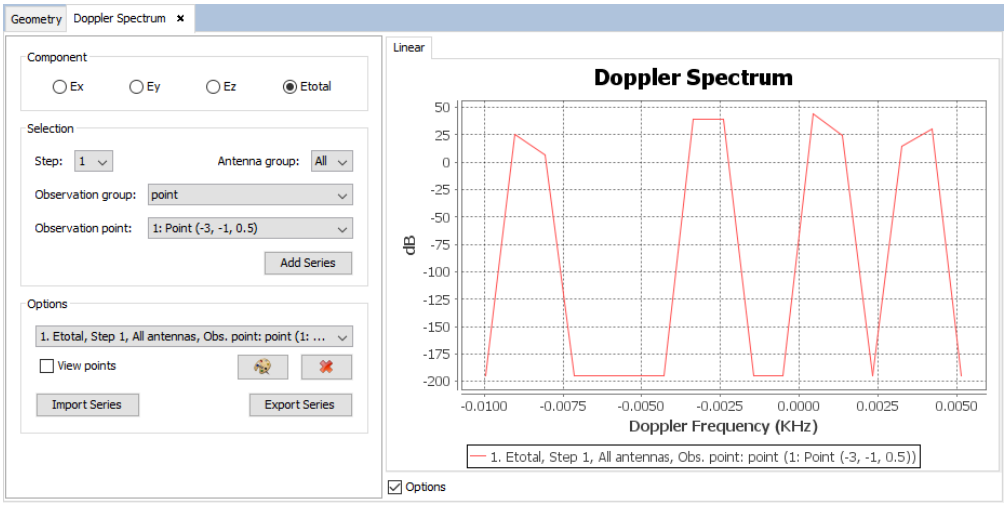


Figure 684: View Doppler spectrum panel

Text Files

When the user selects this option, the following panel will appear:

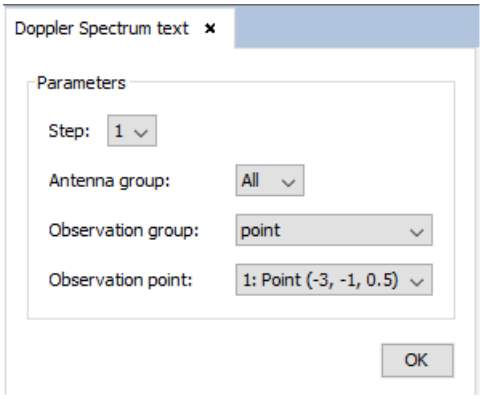


Figure 685: View Text Files panel

In this panel, the user can select the step. Then, the text file is generated and opened with newFASANT text file viewer.

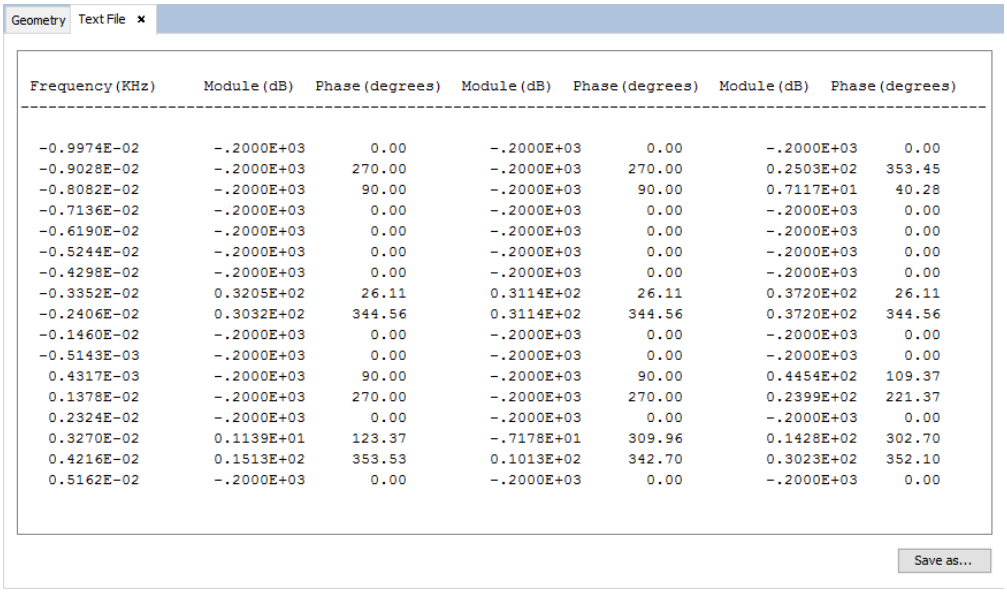


Figure 686: Showing the result text file

The user can export the text file by selecting the "Save as..." button.

7.11.5 Coupling

When the user has executed a **Coupling** simulation, the **Show Results** will look like this:

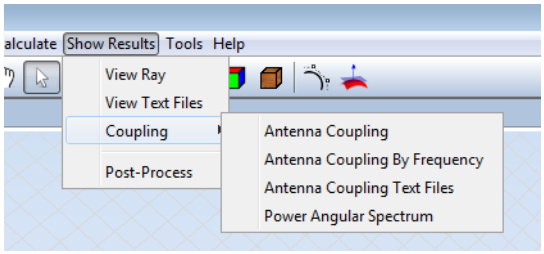


Figure 687: Show Results menu with a coupling simulation

Antenna Coupling

When this option is selected, the following panel will be shown:

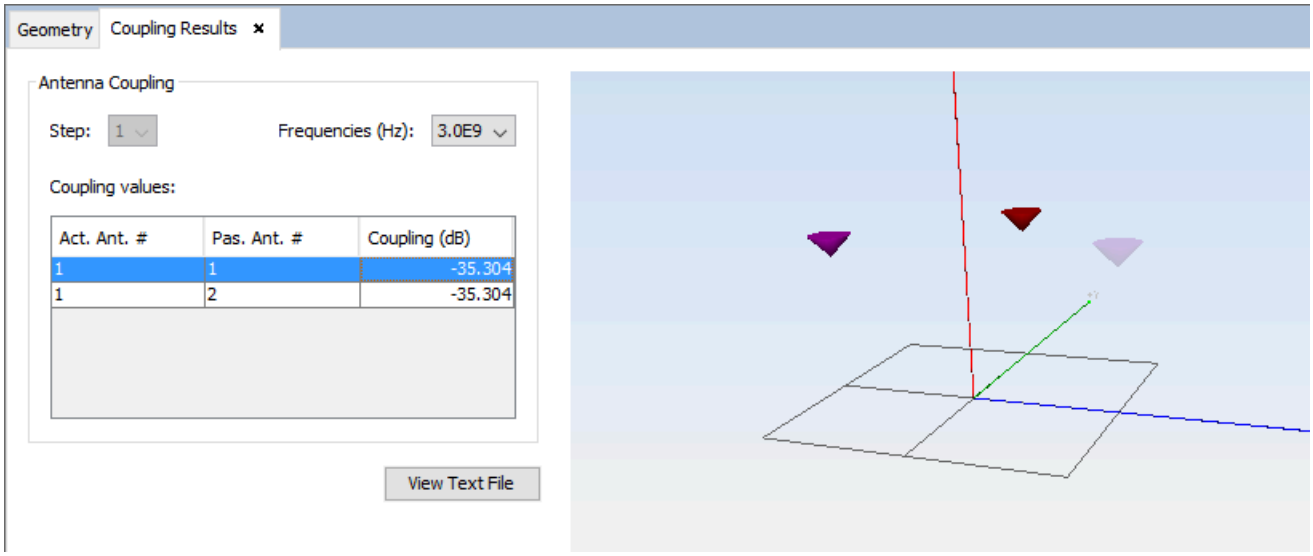


Figure 688: Coupling results panel

This panel allows the user to view the coupling (in dB) between each pair of active antenna group and passive antenna. The user can select a row in the table and the geometry panel will only highlight the antennas corresponding to that coupling value (and rendering the other antennas with low opacity). The user can also select a different step and frequency and the coupling values will be updated immediately.

The user can also visualize the results in text format by pressing the "View Text File" button.

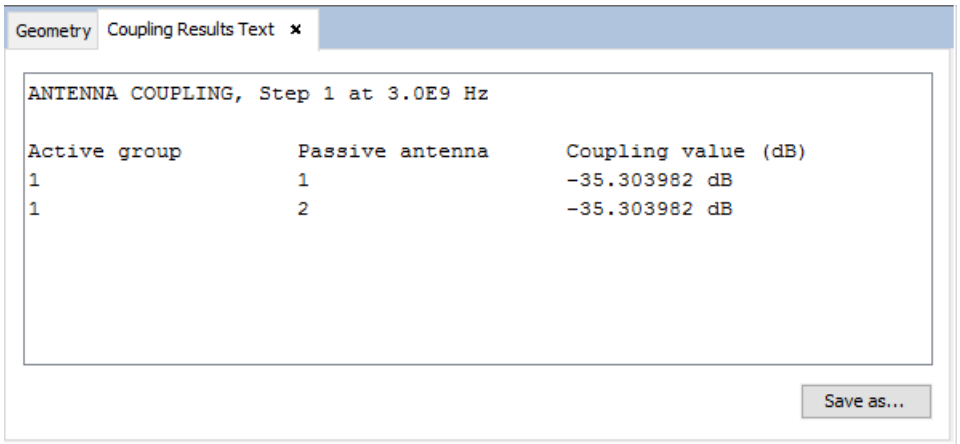


Figure 689: Coupling text file

Antenna Coupling by Frequency

This kind of plot can be used to compare the coupling obtained by the simulation process against a frequency sweep. This option is useful when the user has selected a frequency sweep in the Simulation Parameters, in order to inspect how the coupling value change through a frequency sweep.

When this option is chosen from the Show Results > Coupling > Antenna Coupling By Frequency menu, the panel presented in the figure below will appear.

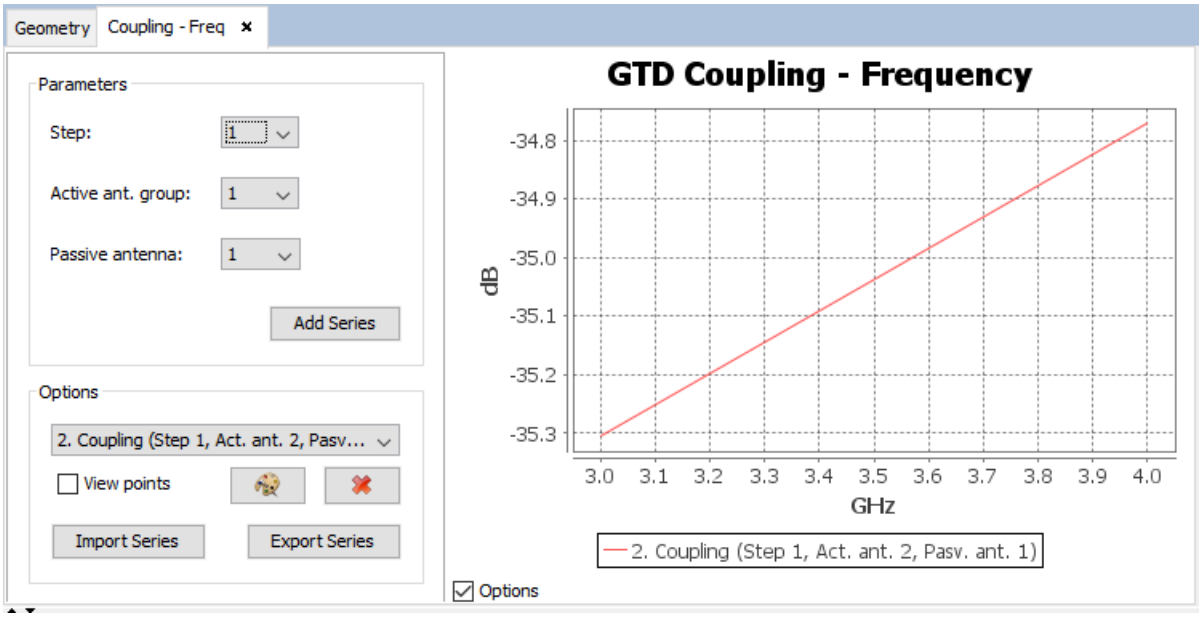


Figure 690: Coupling by frequency

This panel displays a plot with the results. More series can be added by selecting the step in the control panel that appears on the left side of the plot.

- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Active ant. group**: the individual antenna or antenna array that acts as an active antenna.
- **Passive antenna**: the antenna that acts as passive antenna.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look of a series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**.

Series can be imported and exported. To save a series click on the **Export series** button. It is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

Antenna Coupling by Step

This kind of plot can be used to compare the coupling obtained by the simulation process against a frequency sweep. This option is useful when the user has selected a frequency sweep in the Simulation Parameters, in order to inspect how the coupling value changes through a frequency sweep.

When this option is chosen from the Show Results > Coupling > Antenna Coupling By Frequency menu, the panel displayed in the figure below will appear.

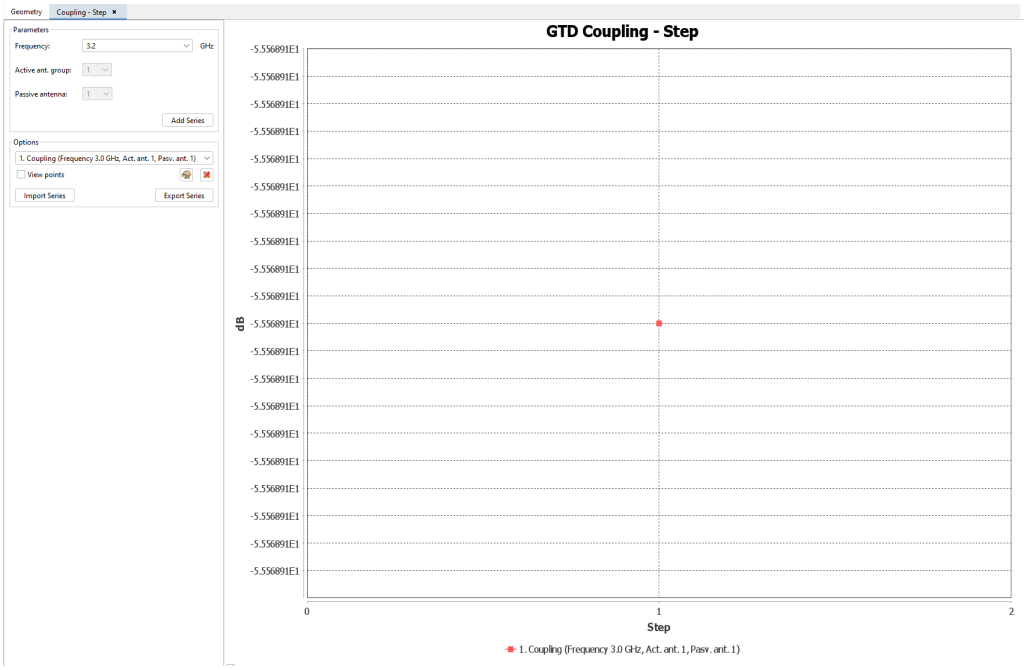


Figure 691: GTD coupling results

This panel displays a plot with the results. More series can be added by selecting the step in the control panel that appears on the left side of the plot.

- Frequency: the user can select a frequency.
- Active ant. group: the individual antenna or antenna array that acts as an active antenna.
- Passive antenna: the antenna that acts as passive antenna.

The selected series can be added to the plot by pressing the Add Series button. It is also possible to modify the display of a series previously added to the chart. You can change the colour of a particular series by selecting the series in the list and pressing the Change Colors button. Series can be imported and exported. To save a series click on the Export series button. It is possible to load previously exported series by pressing the Import Series button and locating the file in which the results were saved in.

Antenna Coupling Text Files

This option shows the coupling results as a text file. First, a panel should appear after selecting this option. Selecting this option will display a panel where the user should select an step. To use this section select a parametric step and click on **OK** to load the data. A table will be presented with the coupling values. To save the result click on the **Save as** button below the table.

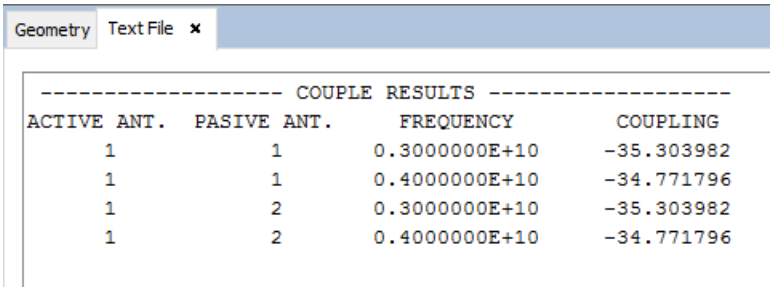


Figure 692: Showing the result text file

Power Angular Spectrum

The Power Angular Spectrum represents the power of the rays that reach the passive antenna according to the arrival direction. The power of each ray that reaches the passive antenna is computed taking into account the angular range in which the ray is received. As shown in the following figure, an imaginary circumference is drawn around the receiver, which is divided into the number of sectors specified by the user. Only the power of the rays received within a specific angular range.

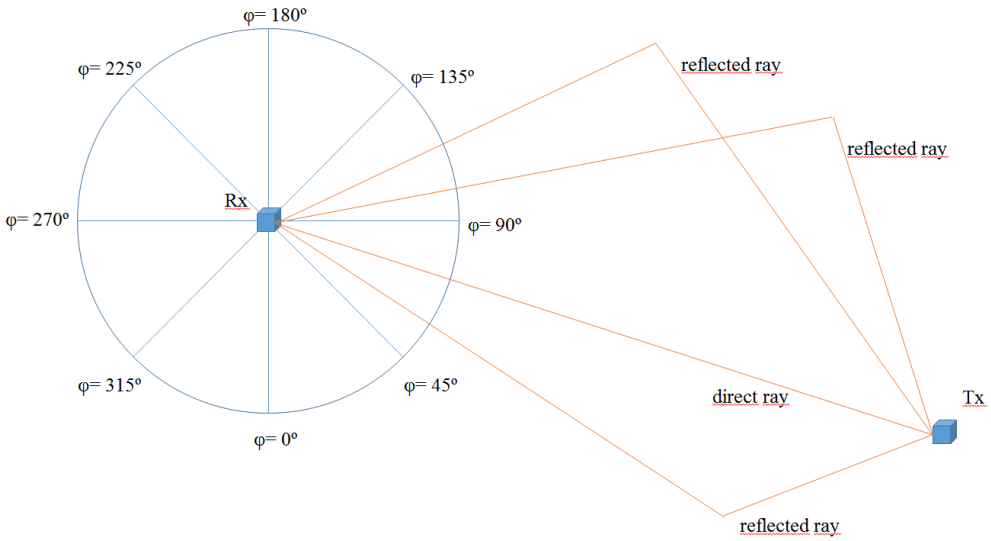


Figure 693: These options are only available when a coupling analysis has been performed. When this option is selected, the following panel is shown:

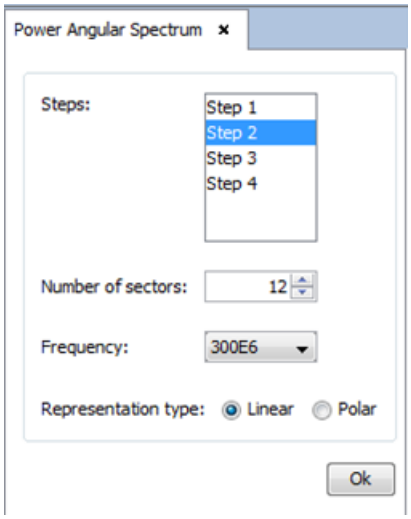


Figure 694: Power angular spectrum panel

In this panel, the user can set:

- Steps: the user can choose the step if a parametric simulation has been performed.
- Number of sectors: number of divisions in which the angular space is divided. For instance, if the number of sectors is 12, 360º is divided by 12, so each angular sector is 30º.
- Frequency: the user can also select the frequency.
- Representation type: the user can also select whether to show a linear graph or a polar one.

When the user presses the **Ok** button, the chart is shown:

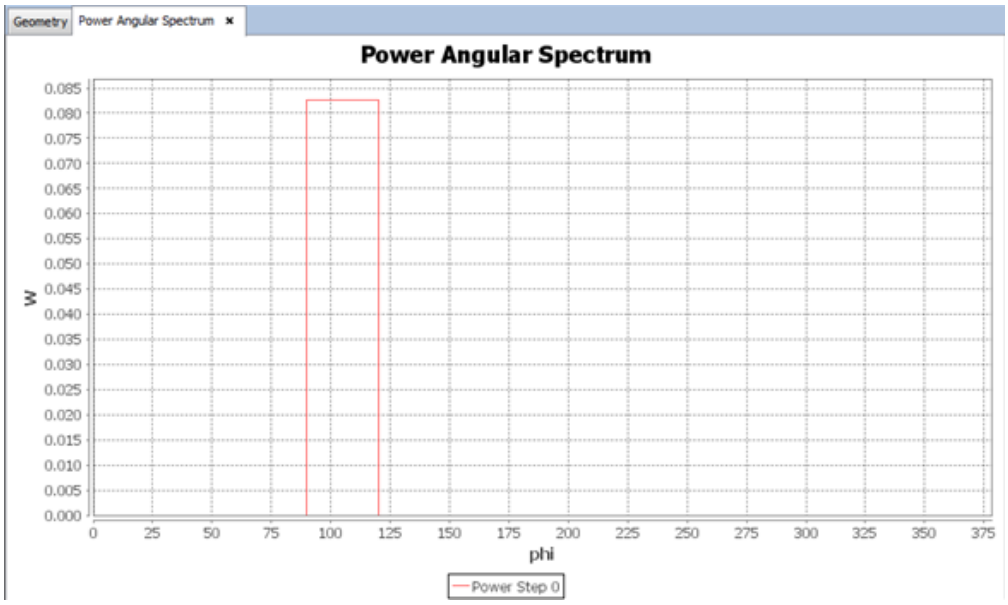


Figure 695: Power angular spectrum chart

7.11.6 MIMO Results

If the user has selected "MIMO" as the simulation type in the Simulation panel, newFASANT will calculate the parameters that characterize the MIMO system defined in the simulation case, in addition to the coupling values.

Channel Transfer Matrix

Pressing the menu item "Show Results -> MIMO Channel -> View Channel Transfer Matrix" will show the following panel:

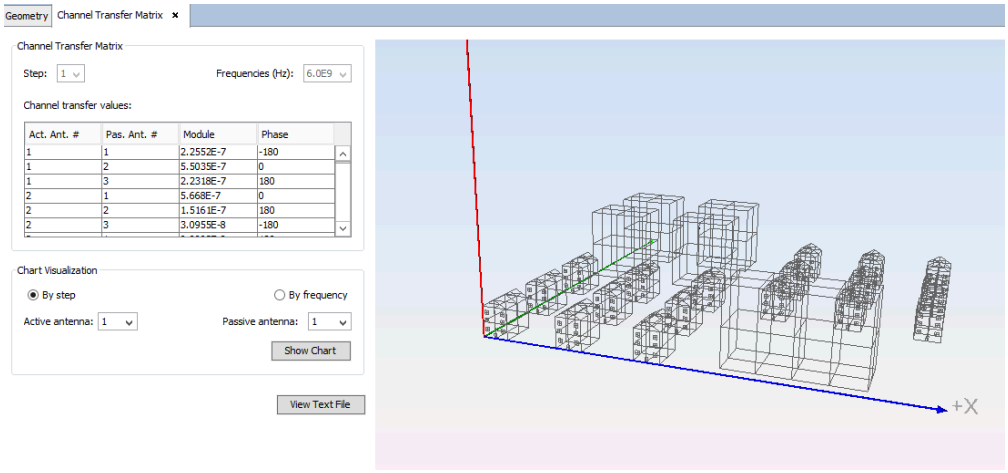


Figure 696: Channel transfer matrix results

If the simulation has more than one step, the user can select the step they want to show the results for, and the same applies for the different simulation frequencies.

The table shows the elements of the channel transfer matrix of the MIMO system. For each value, the table shows the indices of the active antenna and passive antenna pair, as well as the module and phase of the channel transfer matrix element. By selecting any row in the table, the active antenna and passive antenna will be highlighted in the Geometry view, and a line between the two antennas will be drawn. In the geometry panel, transmit antennas are red-colored while receiving antennas are blue-colored.

In this panel, it is also possible to visualize a plot of a certain element of the channel transfer matrix by step or frequency. Pressing the "Show Chart" button will open the Plot Viewer and visualize the selected chart. Refer to the GUI for more details on the Plot Viewer panel.

It is also possible to view a text file with the values of the channel transfer matrix by pressing the "View Text File" button.

Channel Correlation

Selecting the option "Show Results -> MIMO Channel -> View Channel Correlation" will show the following panel:

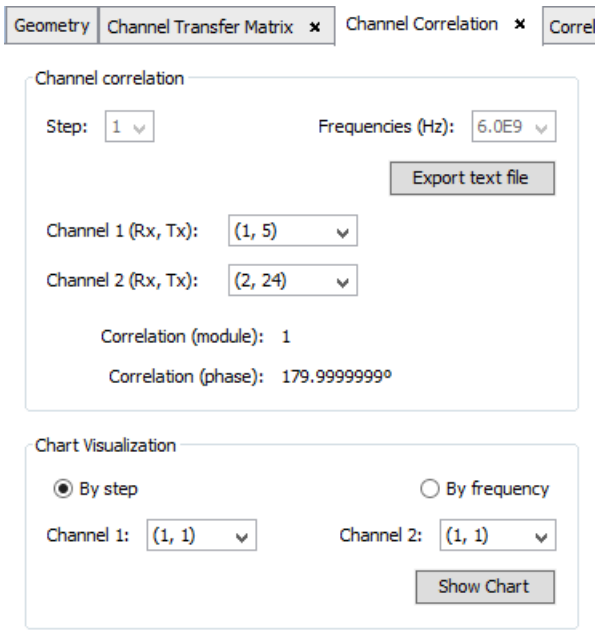


Figure 697: Correlation Matrix results panel

As with the channel transfer matrix panel, the user can select different steps and frequencies to change the shown results (if there is only one step or one frequency in the simulation, the drop-down list will be grayed out).

The "Channel 1 (Tx, Rx)" and "Channel 2 (Tx, Rx)" drop-down lists allow the user to select the two channels for the visualization of their correlation. Each channel is defined by the index of its transmit (Tx) antenna and the index of its receiving (Rx) antenna. If the user selects two different channels, the geometry panel will highlight their antennas and draw a blue line connecting the antennas of the first channel, and a red line connecting the antennas of the second channel. As in the "Channel Transfer Matrix" panel, transmit antennas are red-colored while receiving antennas are purple-colored.

It is also possible to draw a plot of a specific element of the correlation matrix by step or frequency. To do this, the user needs to select whether they want to draw a plot by step or by frequency, select the two channels and press the "Show Chart" button. Doing this will open the Plot Viewer with the selected plot (refer to GUI for more information about Plot Viewer functions).

For each combination of step and frequency, the user can export a text file with the values of the correlation matrix. Pressing the "Export text file" button will show the following window:

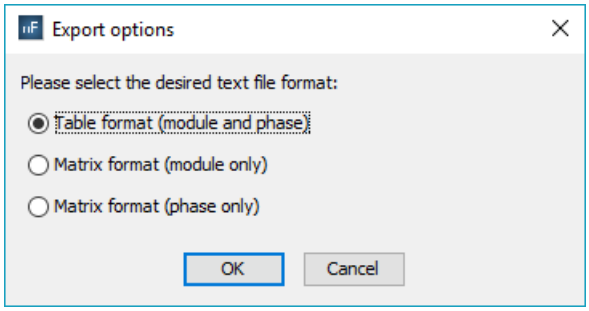


Figure 698: Correlation text file format window

In this window, the user needs to the select the format of the generated text file. There are three different options:

- **Table format (module and phase)**: Generates a text file with its contents arranged in a table. Each row of the table contains the Channel 1 and Channel 2, each one defined by its transmit antenna (Tx) index and receiving antenna (Rx) index; and the module and phase of the correlation for those channels.
- **Matrix format (module only)**: Generates a text file with only the modules of the correlation values. These values are displayed in a matrix format C lines, with each line containing C space-separated values (where C is the total number of channels).
- **Matrix format (phase only)**: The same as the previous format, but the phases of the correlation values are written (instead of the modules).

After selecting the format, the application will show a window in which the user needs to select a location for the text file on their hard drive. The file will be exported to the selected location.

Capacity

Capacity Postprocess

When a MIMO simulation is executed, only the channel transfer matrix and the correlation matrix are calculated. For the user to be able to see the overall capacity of the MIMO system, they first need to specify some parameters.

When the user selects the option "Capacity Postprocess", the following panel is shown:

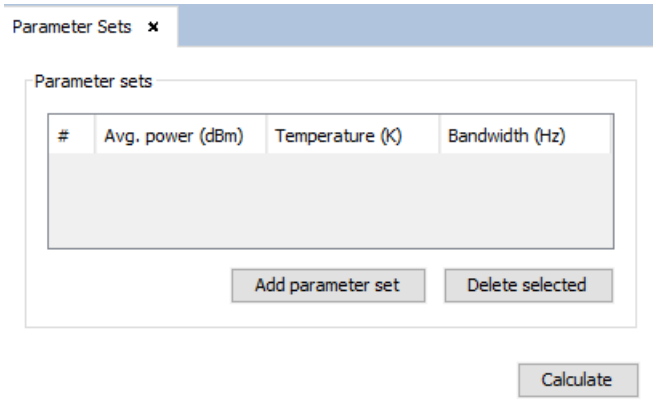


Figure 699: Capacity post-process panel

In this panel, the user can specify multiple sets of the following parameters in the table:

- The **transmitted power per antenna** defines the power, in dBm, that each active antenna transmits.
- The **temperature**, in Kelvin, is the temperature in each of the passive (receiving) antennas. This temperature is used when determining the signal-to-noise ratio in the receivers using thermal noise.
- The **bandwidth** is also used when calculating signal-to-noise ratio at the receiver. It represents the bandwidth over which the thermal noise is calculated.

When the user presses the "Calculate" button, the application will run a post-process to calculate the capacity of the MIMO system for each of the parameter sets configured in the table. The Process Log panel will appear showing the progress of the calculation.

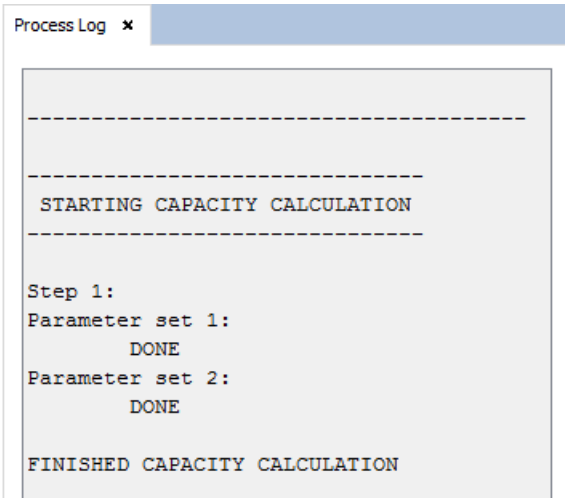


Figure 700: Capacity post-process log

When the post-process has finished running, the user will be able to visualize the results using the options on the "Show Results -> MIMO Capacity" menu.

Viewing Capacity Results

The menu "Show Results -> MIMO Capacity" provides several options to visualize the capacity of the MIMO system:

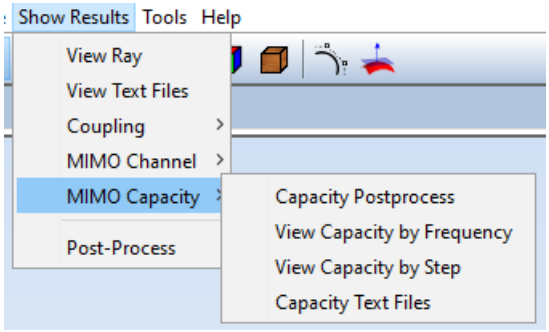


Figure 701: MIMO Capacity menu

The first option, "View Capacity by Frequency", allows the visualization of the capacity in terms of the frequency at a given step. When this option is selected, the following panel will appear:

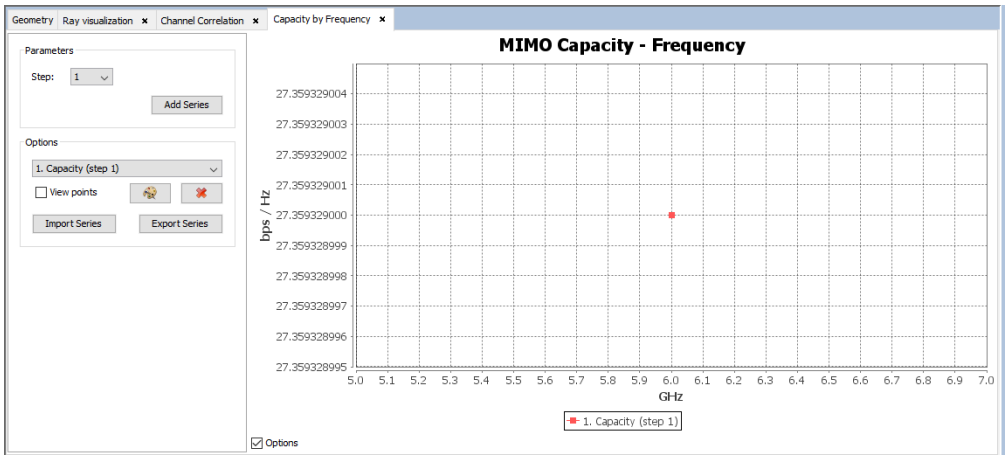


Figure 702: MIMO Capacity by frequency

In this panel, the user can visualize many plots by selecting the step, the parameter set and pressing the "Add Series" button. In addition, it is possible to import existing series from a text file and export a currently shown series to a text file. The user can also change the color of the series and delete them.

Selecting the "View Capacity by Step" menu option will show a similar panel:

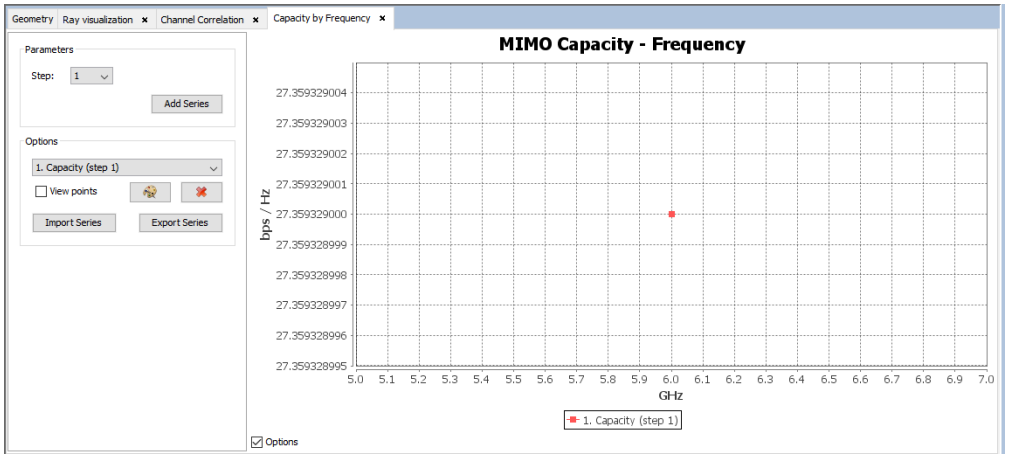


Figure 703: MIMO Capacity by step

In this case, the user can visualize the capacity values in terms of the step for a given frequency value. It is possible to add more plots for different frequency values by selecting the frequency and parameter set and pressing the "Add Series" button. The rest of the functionality is the same as in the "MIMO Capacity by frequency" panel.

If the user selects the "Capacity text files" menu option, a panel will appear (right side of the next figure). This panel will ask the user the step and the parameter set they want to see the results for. When the "OK" button is pressed, the requested text file will be shown.

At the bottom of the "Text File" panel, there is a button labeled as "Save as..." that allows the user to save this text file to a user-given location in their local filesystem.

7.11.7 Post-Process

For information about the Post-Process menu, refer to Post-Process.

7.12 Tools Menu

For information about the Tools menu, refer to Tools Menu.

7.13 Help Menu

For information about the Help menu, refer to Help Menu.

7.14 Training Examples

7.14.1 Example 1: Getting Started

In this section a simple example of a Far Field simulation will be shown. This simulation will be performed considering a cube as the geometry. An antenna will be configured to feed this geometry.

Step 1. The first step consists of creating a new data file where all the parameters of the simulation will be saved. Use the command 'New' from the File menu. Select GTD type.

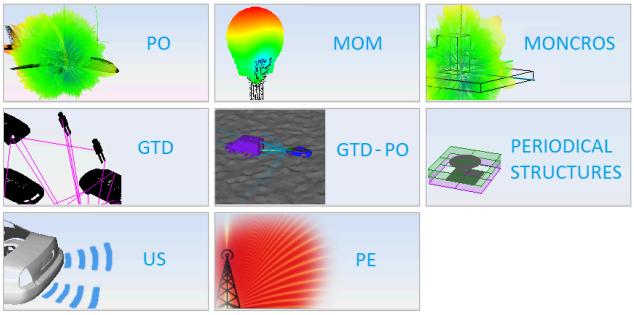


Figure 704: Selection Method dialog box

Step 2. The second step is to generate the box. Select Geometry→ Solid → Box. It will be located at (0,0,0) and the dimensions are 1x1x1 m.

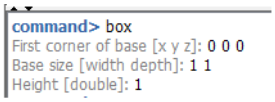


Figure 705: Create box

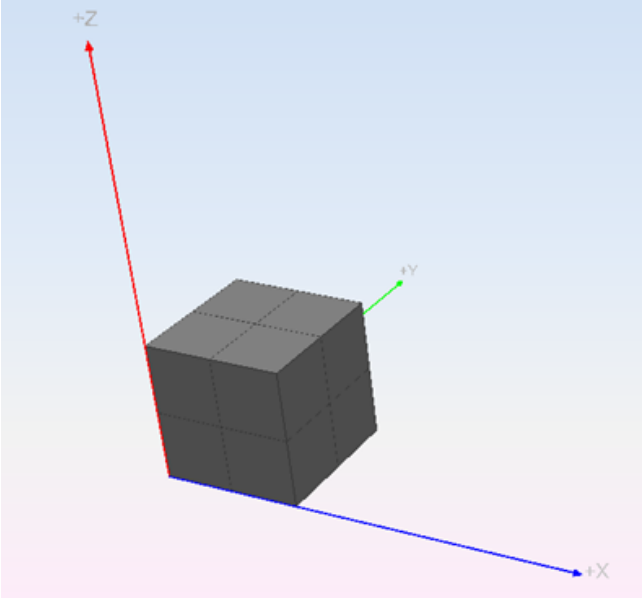


Figure 706: Box situated at (0,0,0)

Step 3. The next step is to set the main parameters of the simulations, which are located in Simulation Parameters Menu → Simulation.

A dialog appears. In this menu, the frequency, number of antennas, kind of simulation, effects to calculate, and other parameters must be defined before running the simulation. In this example, a radiation pattern file in far field is simulated. The user will introduce:

- Far field in Simulation Type.
- The number of frequency points and the frequency: in this example 1 and 2.4 GHz.
- The effect order will be 1.

The rest of the options are the default ones

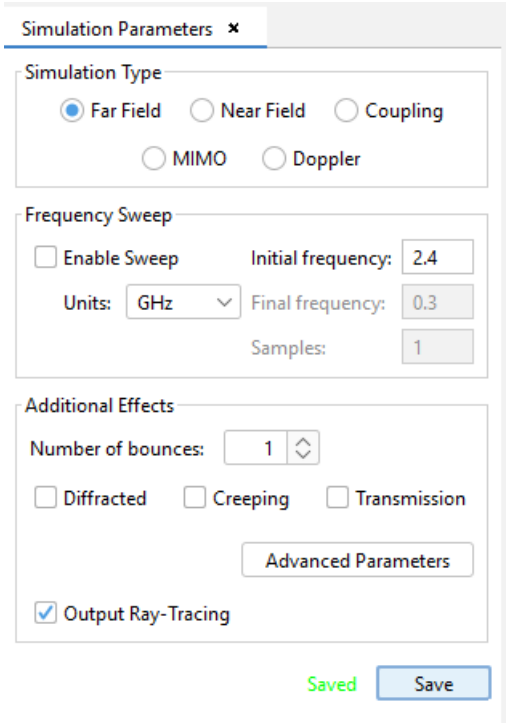


Figure 707: Parameters dialog box

Step 4. Define an antenna over the cube clicking on Antenna → Dipole Antenna. The position of the dipole will be (0.0, 0.0, 1.2).

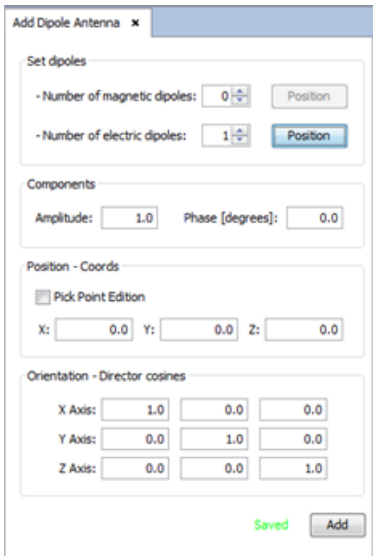


Figure 708: Dipole Antenna dialog box

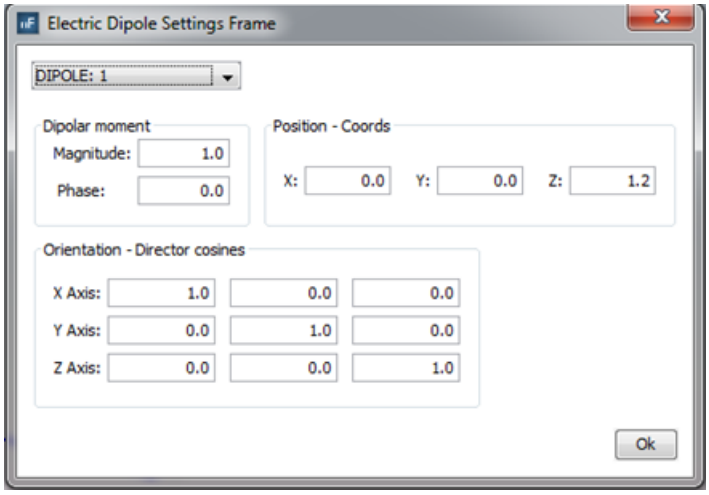


Figure 709: Dipole Antenna dialog box

Step 5.The next step is to situate the Far Field Observation. Select Observation directions from the Output menu. In the Angular Sweep Type dialog select one cut in Phi = 0 degrees and 10 points in theta from 0 degrees to 360 degrees. Select Theta as the Angular Sweeptype.

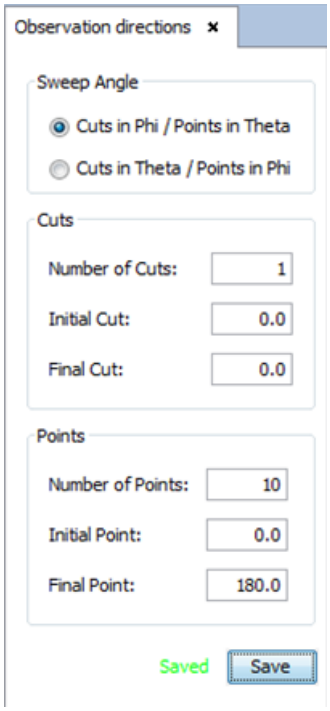


Figure 710: Far Field Observation dialog box

Step 6. To mesh the geometry click on Meshing → Create visibility matrix and select the number of processors.

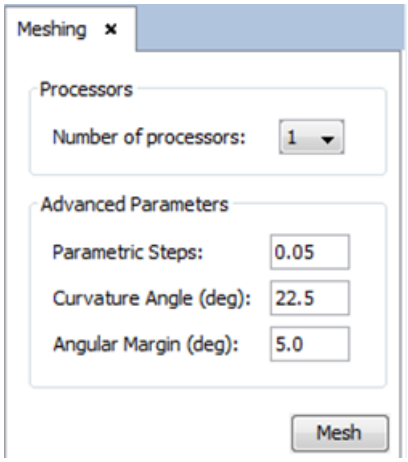


Figure 711: Processors option for meshing

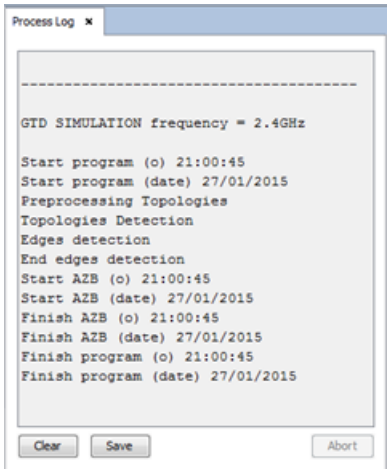


Figure 712: Mesh

Step 7. Execute the simulation by selecting Calculate → Execute and introduce the number of processors to be used.

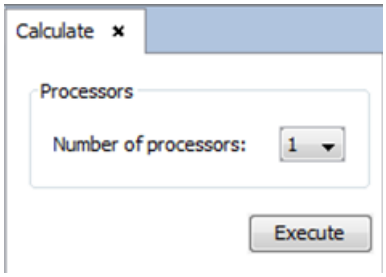


Figure 713: Processors options for Execute

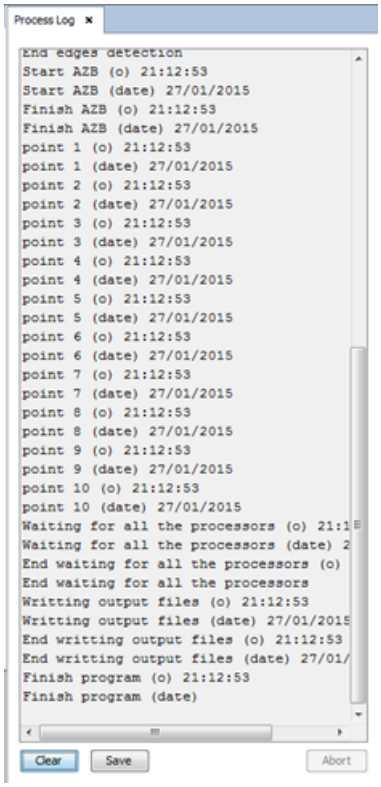


Figure 714: Execute

Step 8. After obtaining the results, choose view ray from show results. It is possible to visualize order 1 rays. It is possible to represent the ray tracing interacting with the geometry.

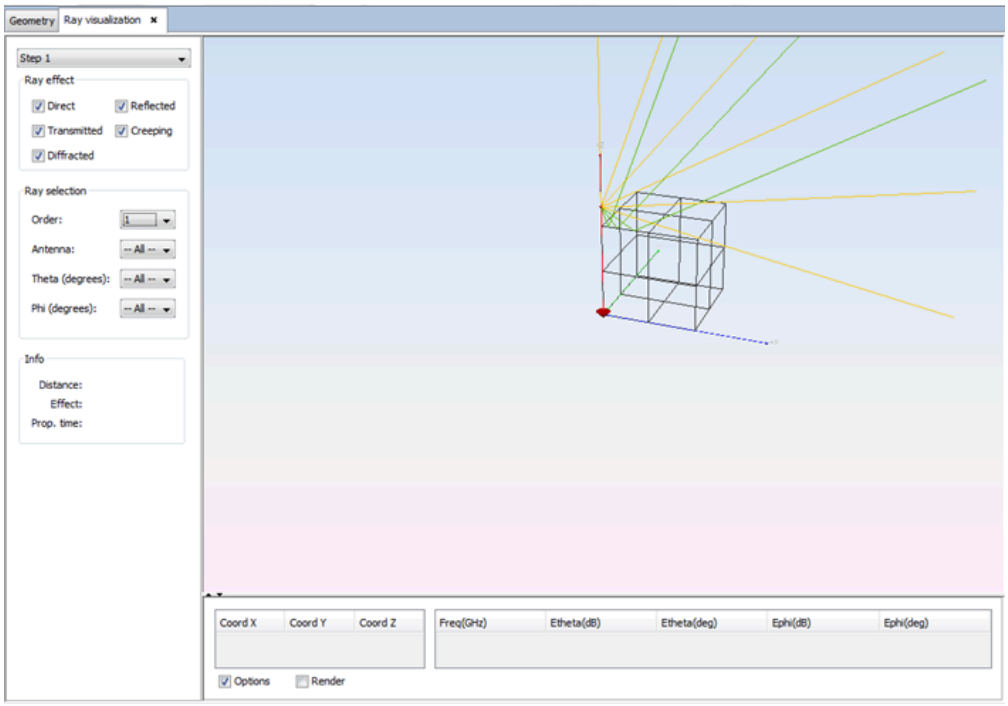


Figure 715: Ray Viewer dialog box

Step 9. It is possible to select a ray to show its information. Clicking in one of the ray will fill its information in the bottom of the panel.

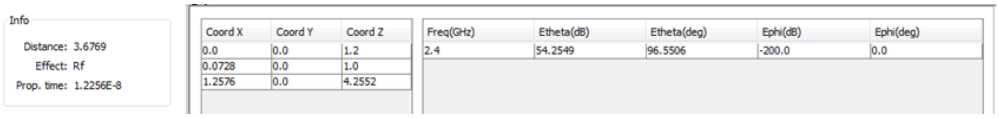


Figure 716: Viewing the ray information

Step 10. To visualize the charts click on Show Results → Far Field → View Cuts, after which the following dialog will be shown:

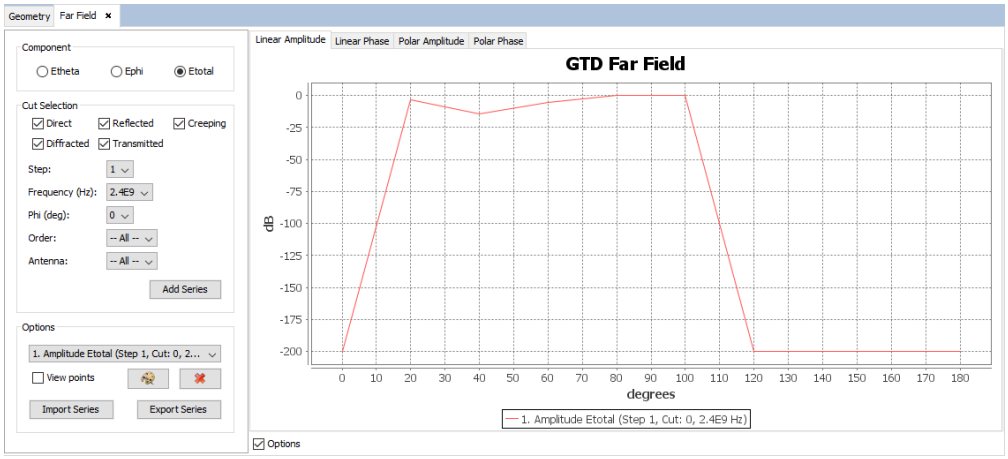


Figure 717: Far field

7.14.2 Example 2: Dynamic Simulation

In this section, we will explain how to compute the Near Field of a dynamic simulation using a simple geometry.

Step 1: Create a newFASANT project for running a GTD simulation. To do this, select the File → New option (or press Ctrl+N) and press the following icon in the panel that appears in order to create a GTD project:

Figure 718: GTD module icon

Step 2: Since we want a dynamic simulation, we need to define some parameters in order to be able to parametrize our geometry. In this simulation, we will create two boxes moving in opposite directions along the Y axis. These boxes will move at a rate of 0.5 meters per step, considering a total of 4 steps. To add a parameter, select the **Geometry** → **Parameters** → **Define Parameters** option and press the **Add parameter**. A new row in the parameters table will be created. Change the name of the parameter to something more descriptive like "yPos1" and change the text in the **Values** column to:

[-0.5, -2.0] 4

This can be read as "This parameter will take four equally spaced values, beginning in-0.5 and ending in -2.0" (so the parameter will take the values -0.5, -1.0, -1.5 and -2.0).

Repeat the same procedure to add a new parameter called "yPos2", which will describe the movement of the second box. Set the content of the **Values** column of this second parameter to the following text:

[-0.5, 1.0] 4

Press the **Save** button to save the parameter settings.

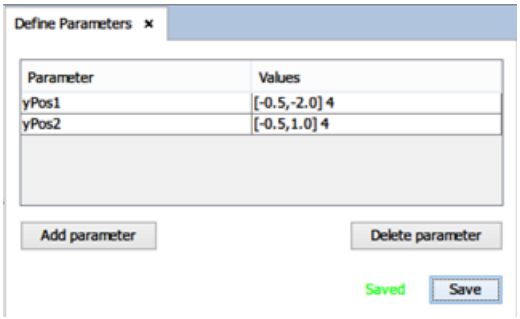


Figure 719: Defining the parameter

Step 3: Now we will create the geometry for the simulation. Let's create both boxes using the command line (the command line is the panel at the bottom side of the window, below the geometry view). The following figure shows the commands to be entered into the command line in order to create the boxes.

```
command> box
First corner of base [x y z]: -1.5 yPos1 0
Base size [width depth]: 1 1
Height [double]: 1
command> box
First corner of base [x y z]: 0.5 yPos2 0
Base size [width depth]: 1 1
Height [double]: 1
```

Figure 720: Commands used to create the geometry

As we can see, we have used the parameters created in the previous step to define a varying Y coordinate in the boxes. The result is shown in the following figure. Note that the following figure depicts the state of the geometry in the first step (i.e. both parameters are equal to the first value in their set of values):

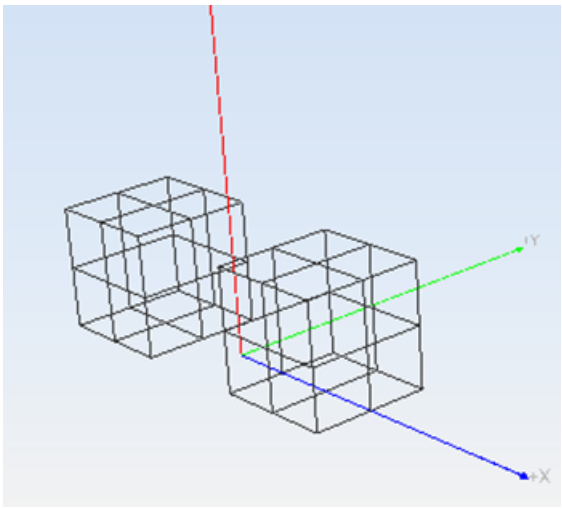


Figure 721: Created boxes

Step 4: Now we are going to set up the parameters of the simulation. Select **Simulation Parameters** → **Simulation**. Enter the following parameters:

- **Near Field**simulation type.
- **Initial frequency** 4.0 GHz.
- Disable **Frequency Sweep.**
- Enable diffraction.
- Keep the number of bounces equal to 1.

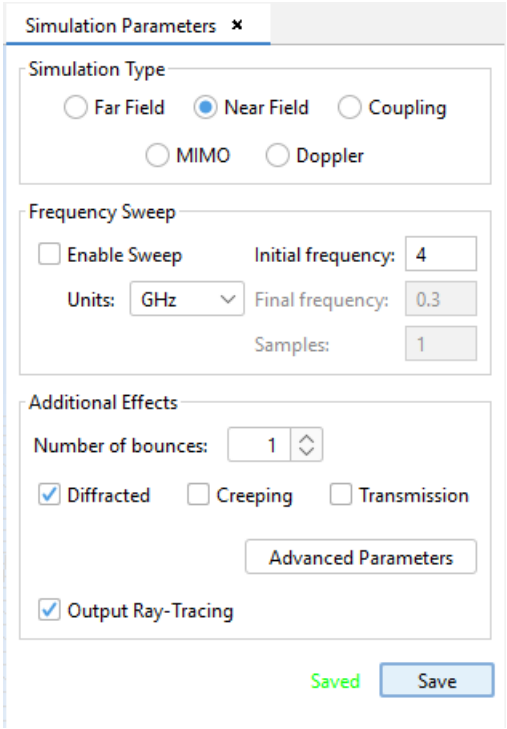


Figure 722: Simulation parameters

Step 5: Define an antenna over the cube by selecting the option **Antennas** → **Dipole Antenna** and setting up the antenna parameters as in the following figure:

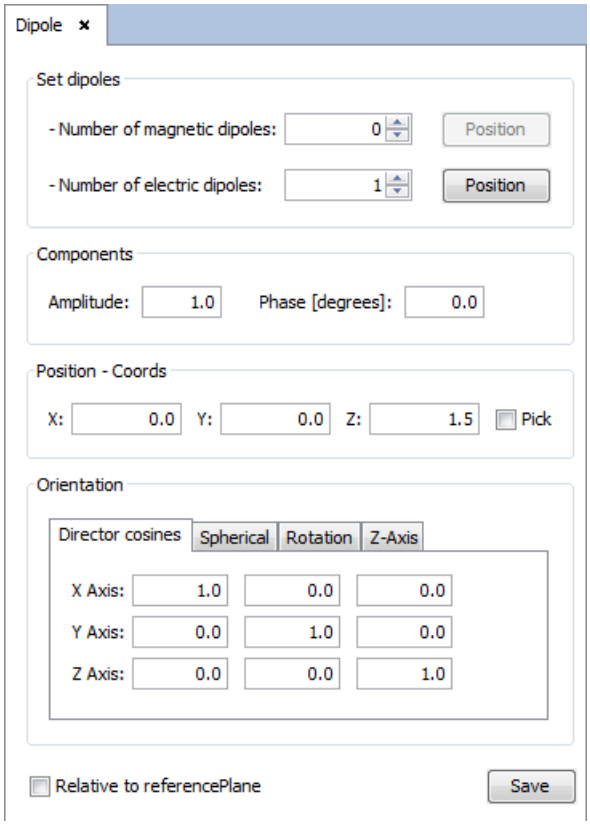


Figure 723: Antenna parameters

Press the **Add** button to add the antenna to the simulation.

Step 6: Now we are going to add a line of observation points. Select the **Output** → **Observation** Points option. In the **Geometry Types** combo box, select **Line** and press the **Add** button. The following dialog will pop up. Set the line parameters as shown in the figure.

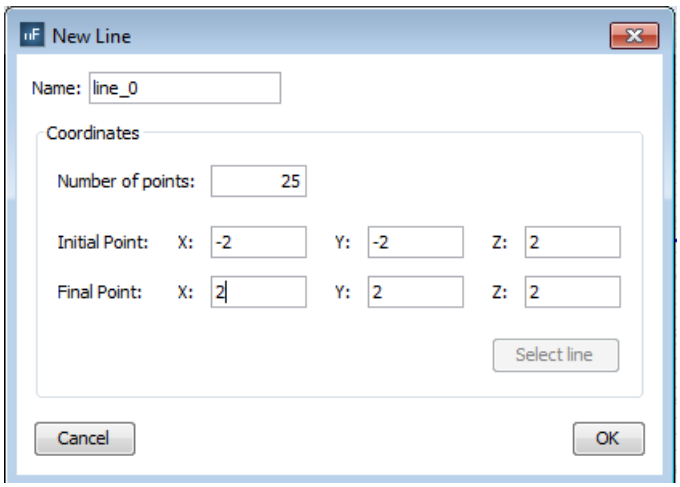


Figure 724: Observation line parameters

Press **OK** button to add the observation line.

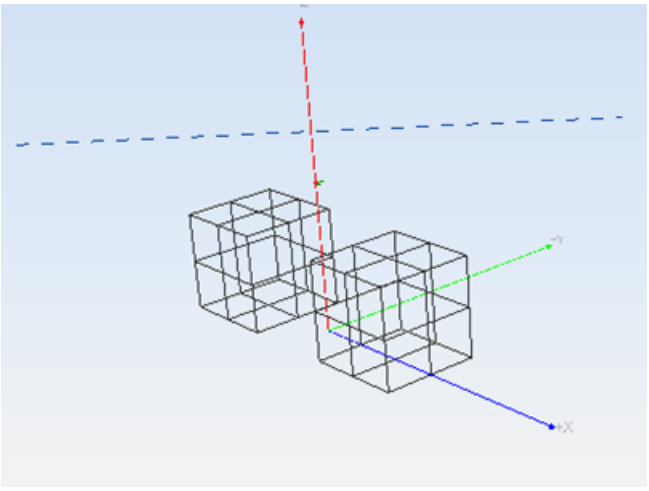


Figure 725: Geometry view showing the observation line

Step 7: Select **Meshing** → **Create Visibility Matrix** to mesh. Select the number of processors to use for the meshing process and press the **Mesh** button.

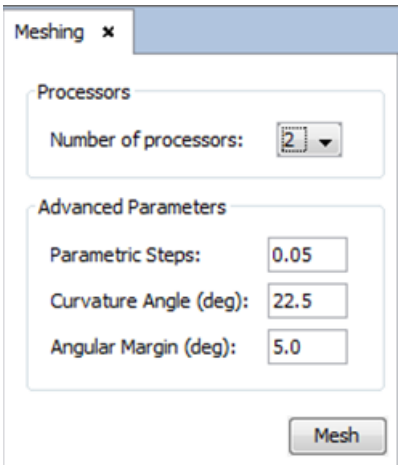


Figure 726: Selecting the number of processors for the meshing

Step 8: Once the meshing process has finished (no more text is being outputted into the process log), we are ready to run the simulation. To do this, select **Calculate** → **Execute**, select the number of processors to use (like we did before meshing) and press the **Execute** button.

Step 9: When the calculation has finished, press **Show Results** → **View Ray**. In the left side of the **Ray Visualization** panel, select Step 2. The result should be the same as in the following figure:

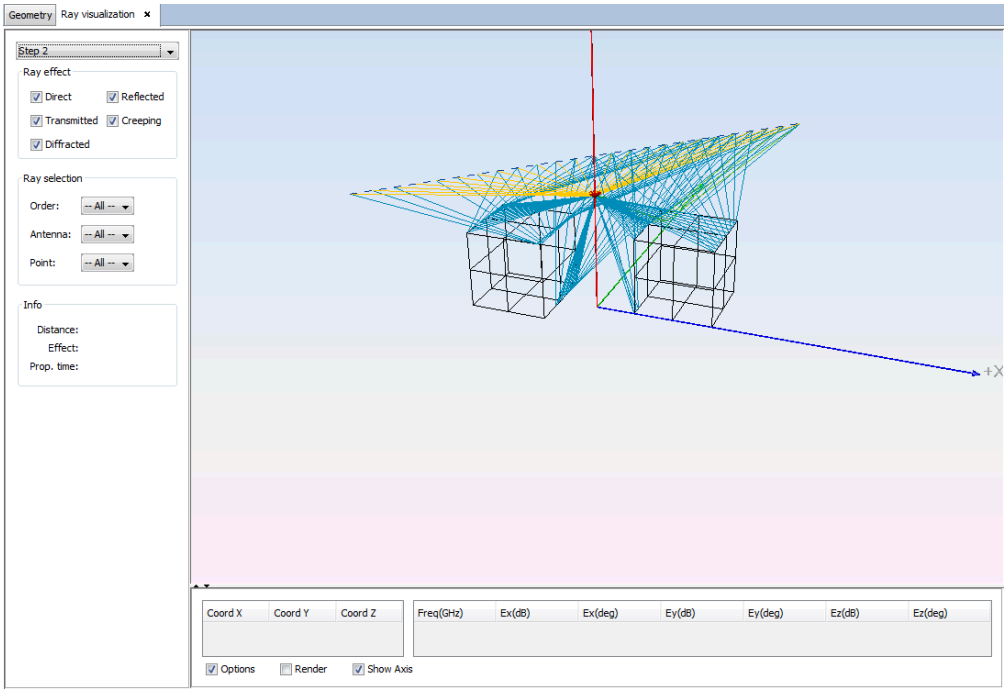
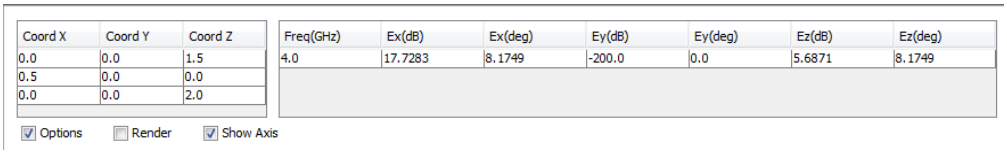


Figure 727: Ray visualization

Step 10: It is possible to select a ray to show its information. Clicking in one of the rays will fill its information in the bottom of the panel.



Coord X	Coord Y	Coord Z
0.0	0.0	1.5
0.5	0.0	0.0
0.0	0.0	2.0

Freq(GHz)	Ex(dB)	Ex(deg)	Ey(dB)	Ey(deg)	Ez(dB)	Ez(deg)
4.0	17.7283	8.1749	-200.0	0.0	5.6871	8.1749

Options Render Show Axis

Figure 728: Information of a selected ray

Step 11: To visualize the angular cuts, click on **Show Results** → **Near Field** → **View Cuts**.

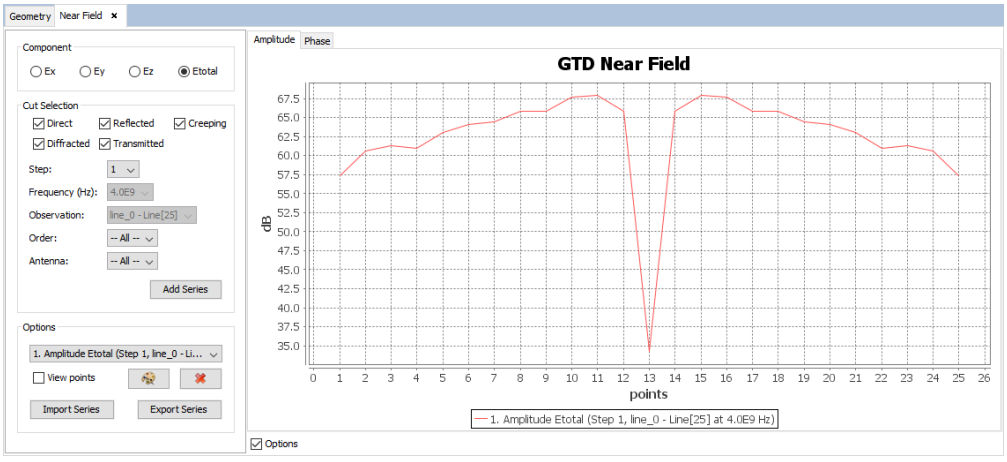


Figure 729: View cuts

Step 12: To visualize the evolution of the near field along the different steps of the simulation, select **Show Results** → **Near Field** → **View Cuts by Step**:

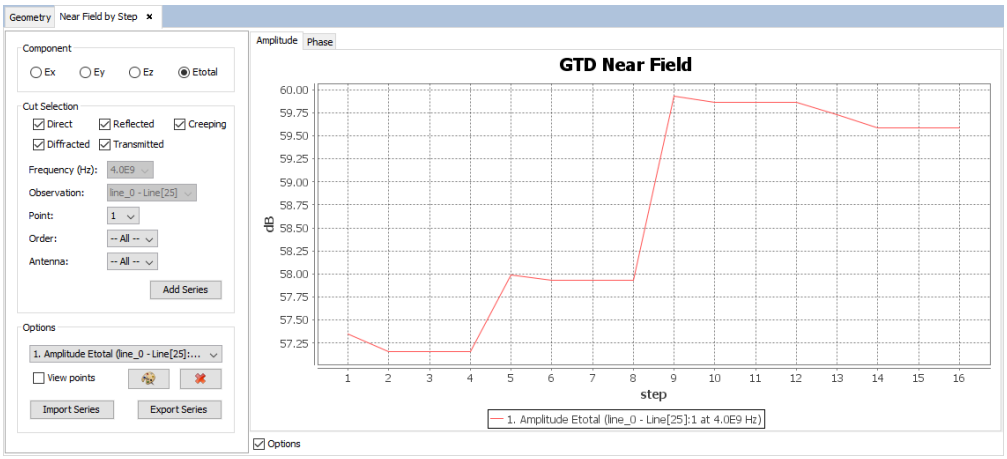


Figure 730: View cuts by step

7.14.3 Example 3: Radiation Pattern File Simulation

In this section, we show how to compute the Near Field using a simple geometry fed by a customized DIA (radiation features definition) file.

Step 1: Create a newFASANT GTD project by selecting the **File** → **New** option (or alternatively pressing Ctrl+N) and click on the following icon in the panel:

Figure 731: GTD module icon

Step 2: To create a new radiation pattern file (.dia), select the **Tools** –> **DIA Files** option.

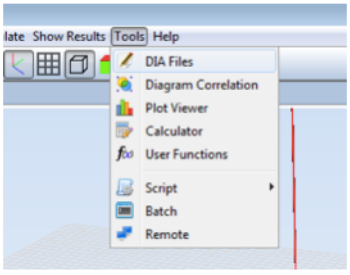


Figure 732: DIA Files option

Step 3: Introduce the following parameters in the **DIA Files Editor**. The formula shown in the first figure has been used to fill the Etheta(dB) column, and the formula is shown in the second figure has been used to fill the Ephi(dB) column.

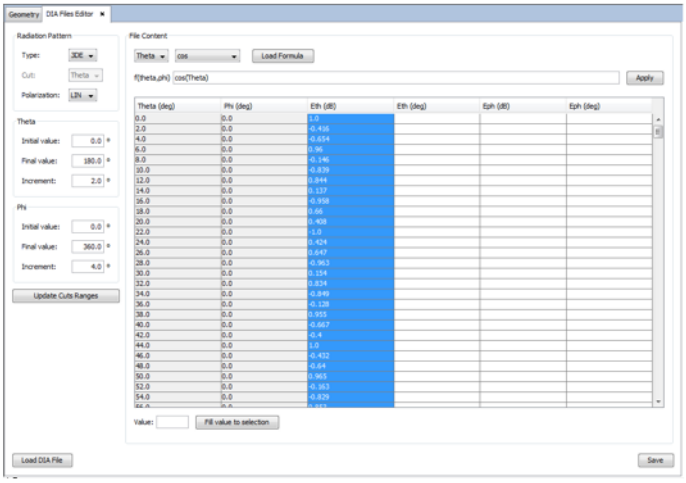


Figure 733: DIA Files Editor

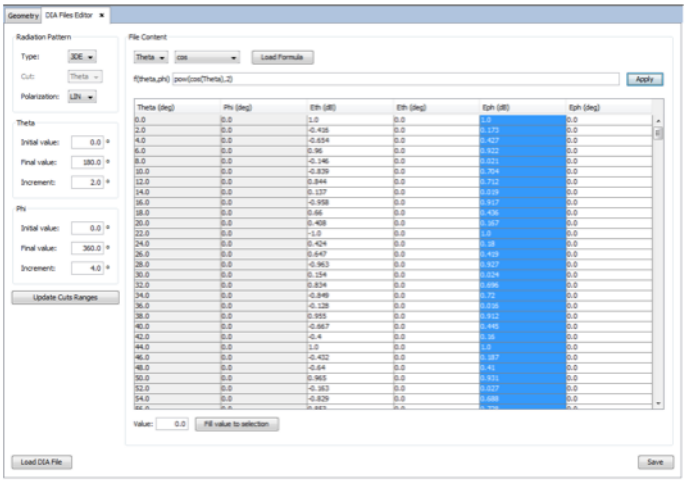
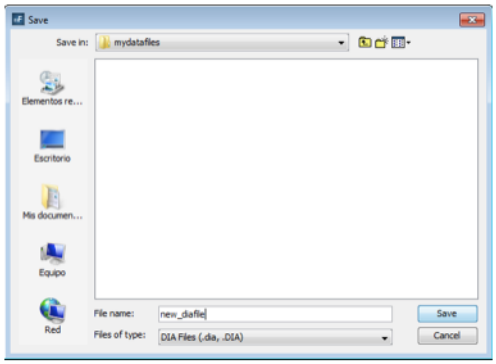


Figure 734: DIA Files Editor

Step 4: Click on the **Save** button to save the DIA file.



Step 5: Now create the geometry for the simulation. Let’s generate a cylinder using the following command:

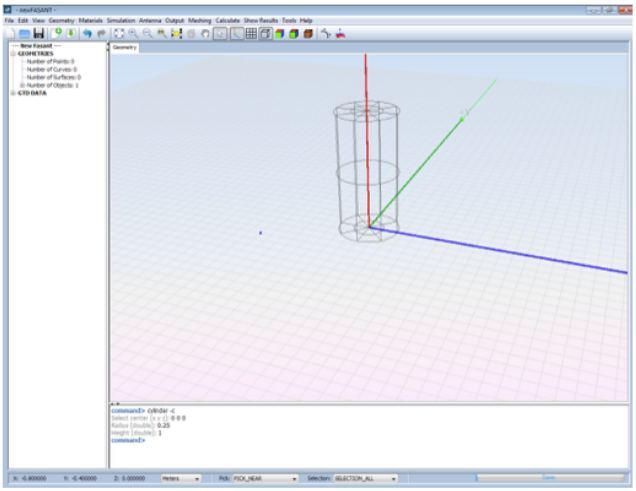


Figure 735: Created cylinder

Step 6: Now set up the parameters of the simulation. Select **Simulation Parameters** → **Simulation**. Enter the following parameters:

- **Near Field** simulation type.
- **Initial frequency** 3.0 GHz. Disable **Frequency Sweep**.
- Enable diffraction.
- Keep the number of bounces equal to 1.

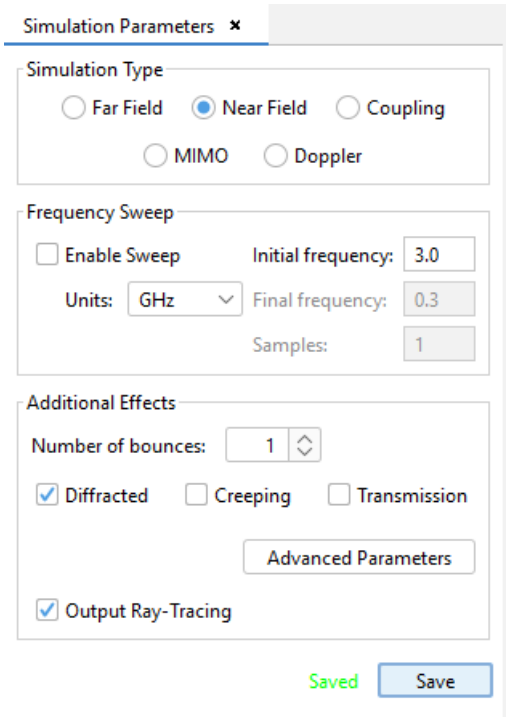


Figure 736: Simulation parameters

Step 7: Antenna -> Pattern File Antenna -> Import DIA File

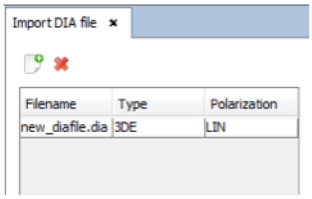


Figure 737: Import DIA file

Step 8: Antenna -> Pattern File Antenna -> Pattern File Antenna

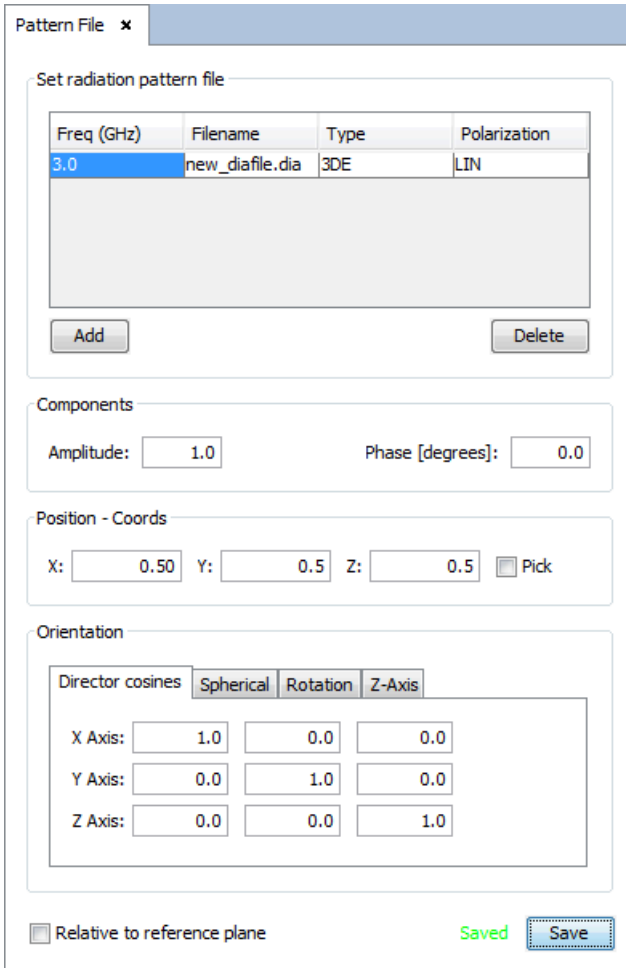


Figure 738: Pattern File parameters

Step 9: Now add a line of observation points. Select the **Output** → **Observation** Points option. In the **Geometry Types** combo box, select **Line** and press the **Add** button. The following dialog will pop up. Set the line parameters as shown in the figure.

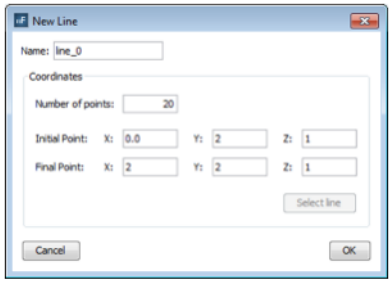


Figure 739: Observation line parameters

Step 10: Select **Meshing** → **Create Visibility Matrix** to mesh. Select the number of processors to use for the meshing process and press the **Mesh** button.

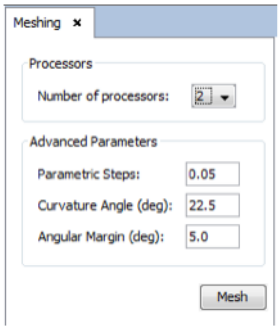


Figure 740: Selecting the number of processors for the meshing

Step 11: Once the meshing process has finished, run the simulation. To do this, select **Calculate** → **Execute**, select the number of processors to use (like we did before meshing) and press the **Execute** button.

Step 12: When the calculation has finished, press **Show Results** → **View Ray**. On the left side of the **Ray Visualization** panel, select Step 2. The result should be the same as in the following figure:

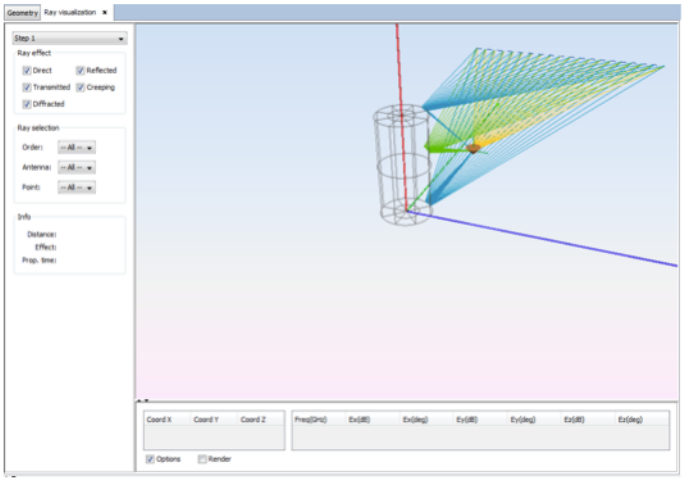


Figure 741: Ray visualization

Step 13: It is possible to select a ray to show its information. Clicking in one of the rays will fill its information in the bottom of the panel.

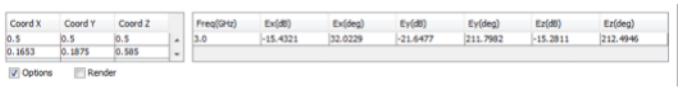


Figure 742: Information of a selected ray

Step 14: To visualize the angular cuts, click on **Show Results** → **Near Field** → **View Cuts**.

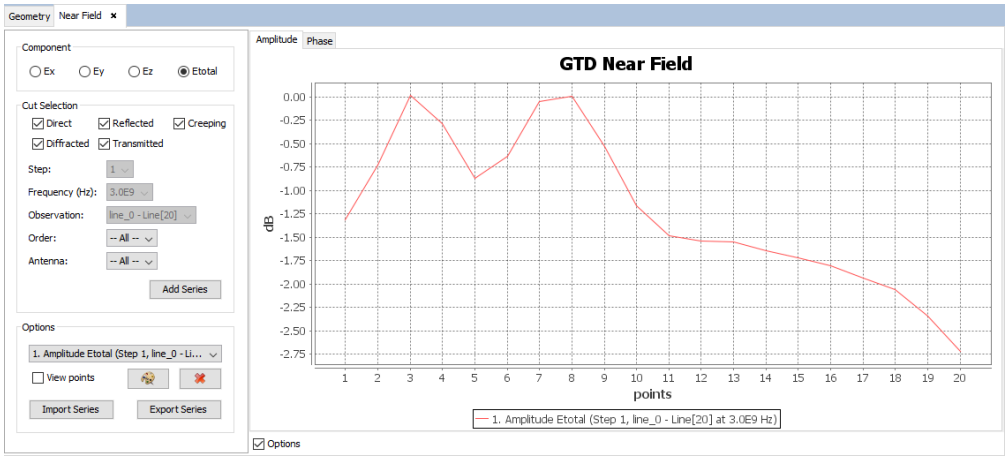


Figure 743: View cuts

7.14.4 MIMO Examples

MIMO Example with 25x25 TX and 3x1RX in Urban Environment

Step 1: Create a new **GTD Project.**

Open newFASANT and select **File - New option.**

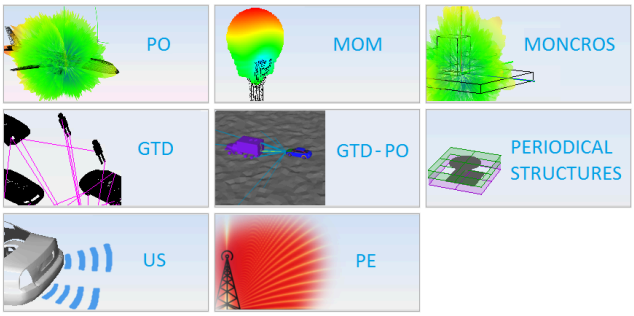
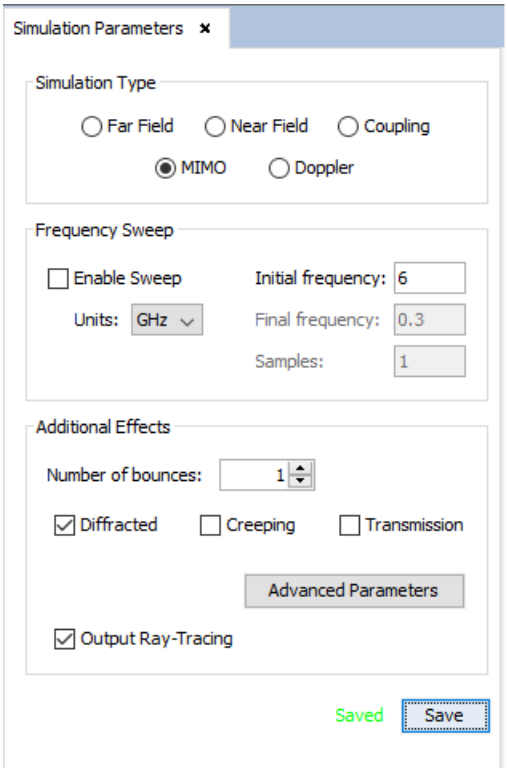


Figure 744: Select GTD option on the previous figure and start to configure the project.

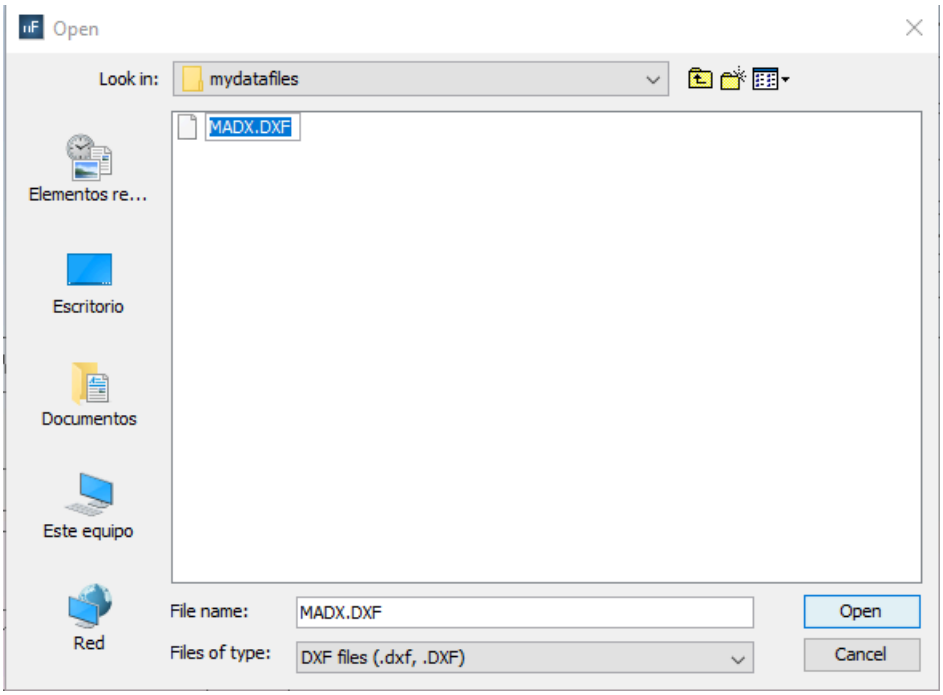
Step 2: Set the **simulation parameters** as shown.

Select **Simulation - Parameters** option, click on the **MIMO** button and set the parameters.



Step 3: Import the geometry.

In order to import the desired geometry, select **Geometry – Import**, select the **.igs** file and open it.



Step 4: Create the transmitter array.

Select **Antenna – Dipole – Dipole Array** and set the parameters.

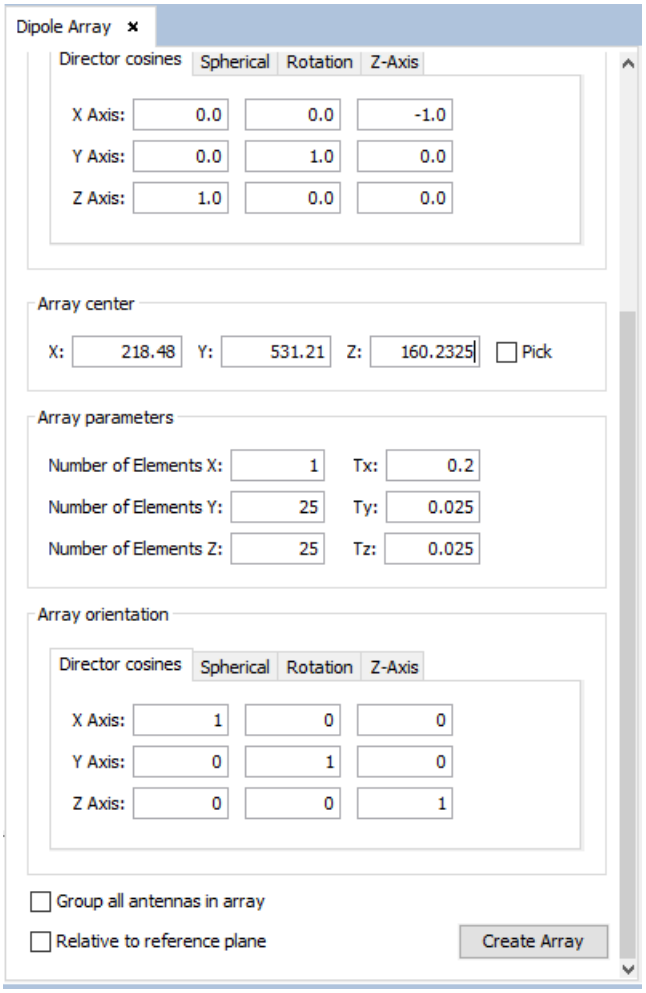
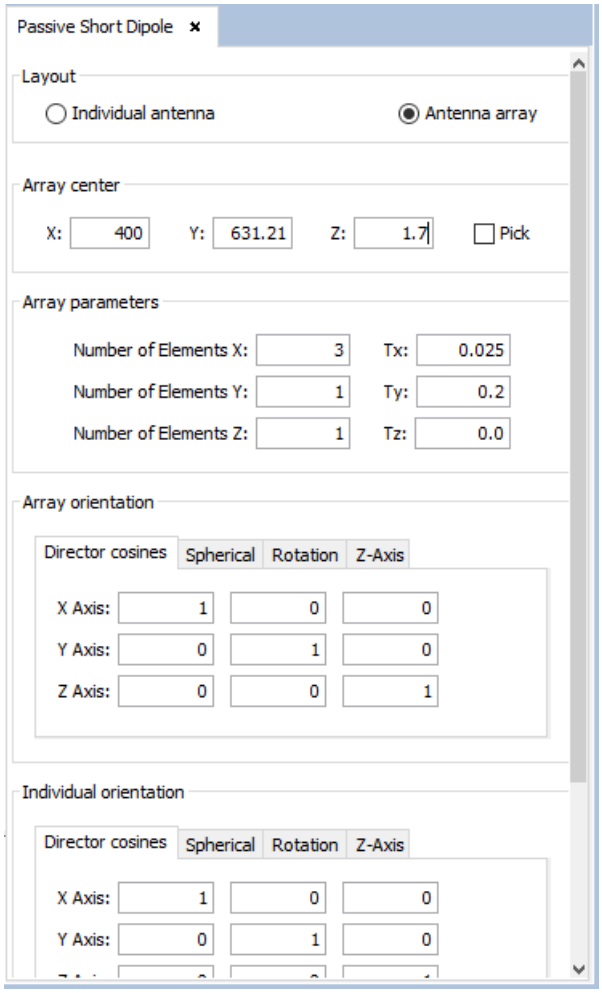


Figure 745: A spacing of 0.025 m. between array elements is chosen because is equivalent to a spacing of 0.5 lambdas at 6 GHz.

It is important that the **Group all antennas in array** option is left **unchecked**.

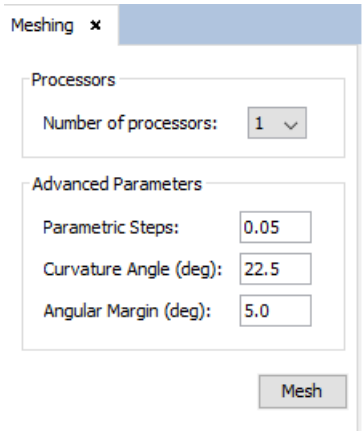
Step 5: Create the receptor array.

Click on **Output – Coupling – Short Dipole** and select the **Antenna array option**. Set the parameters as shown.



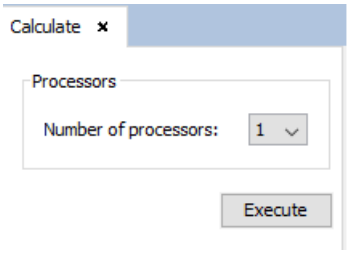
Step 6: Meshing the geometry model.

Select **Meshing - Create Matrix Visibility** to open the meshing configuration panel and then set the parameters as show the next figure.



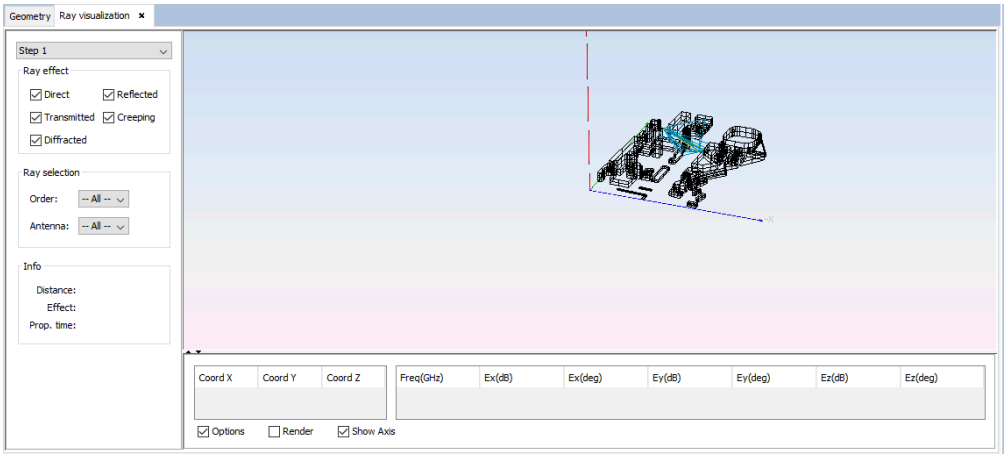
Step 7: Execute the simulation.

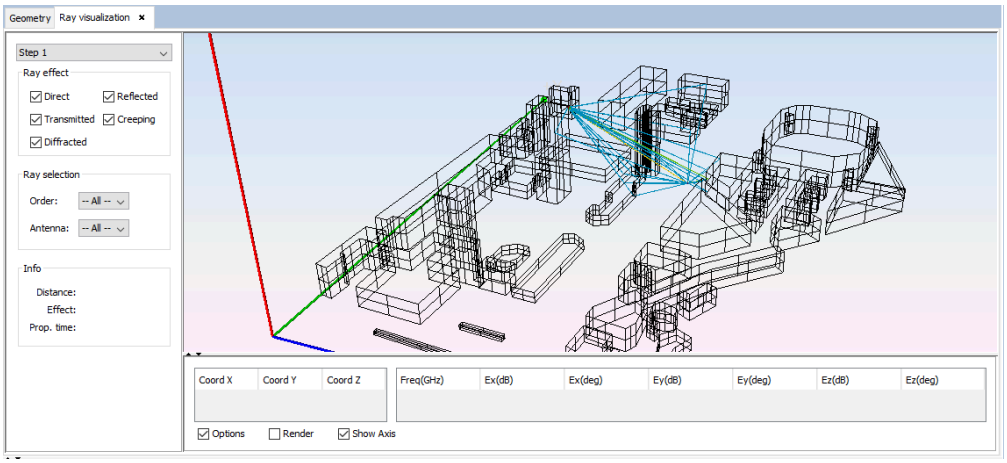
Select **Calculate - Execute** option to open simulation panel.



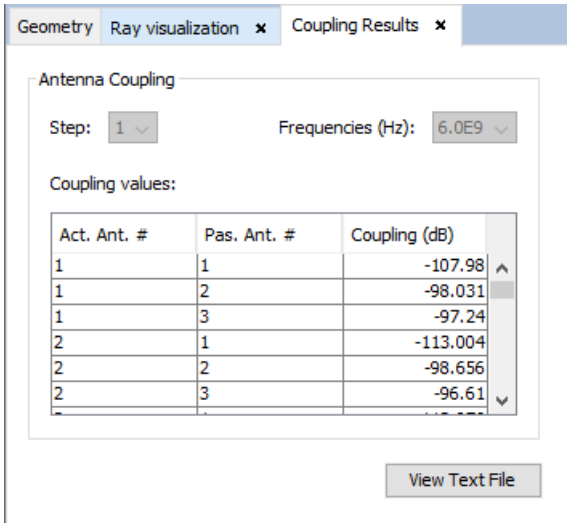
Step 8: Show Results.

The **ray tracing** can be visualized by clicking **Show Results – View Ray**.

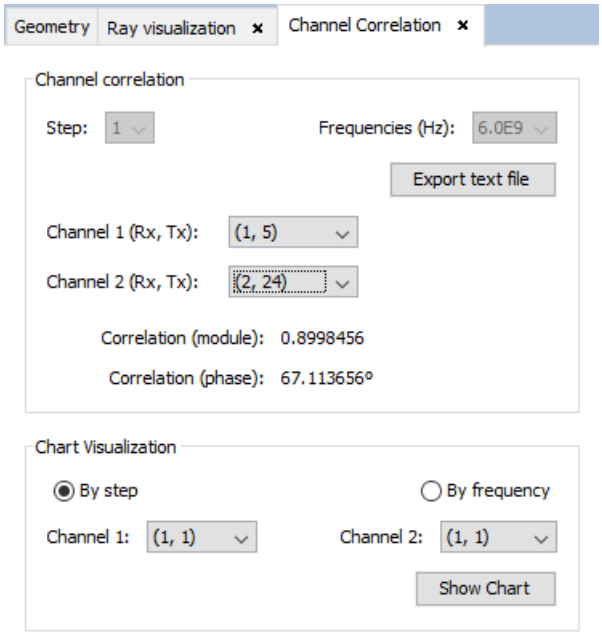




The coupling between the transmitter and the receptor antennas can be visualized by clicking Show Results – Coupling – Antenna Coupling.



The channel correlation can be visualized by clicking Show Results – MIMO Channel – View Channel Correlation.



In order to be able to see the overall capacity of the MIMO system, the user need to specify some parameters.

When the user selects the option **Show Results – MIMO Capacity – Capacity Post-process**, the following panel is shown:

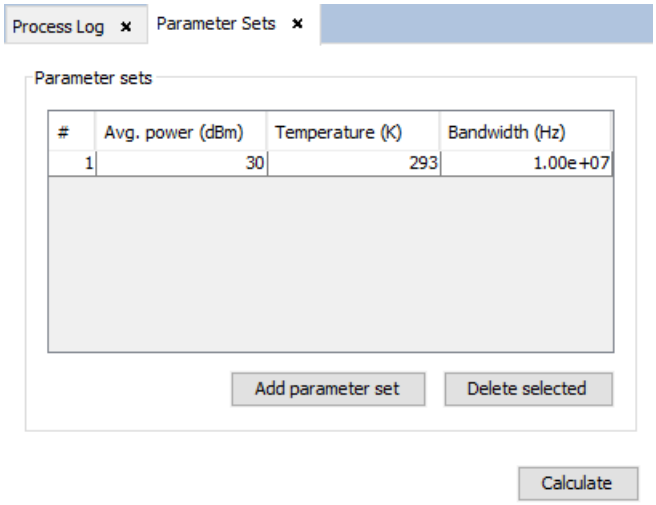
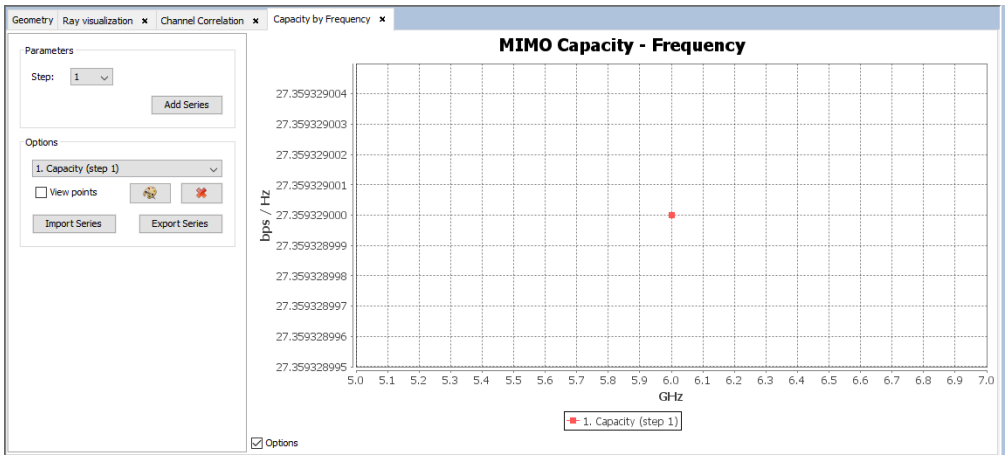


Figure 746: Capacity post-process panel

In this panel, the user has to specify the transmitted power per antenna, the temperature, and the bandwidth. In this example, the user has to set up the values shown in the above figure.

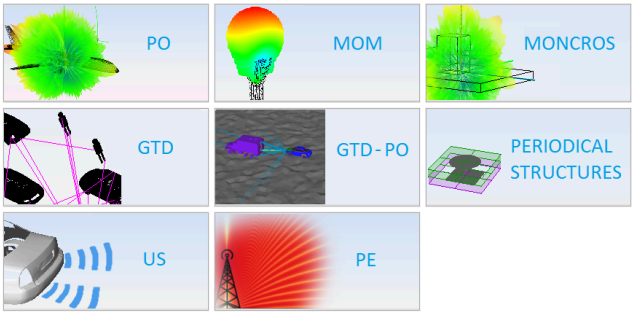
The capacity at the selected frequency can be visualized by clicking **Show Results – MIMO Capacity – View Capacity by Frequency.**



MIMO Example with 25x25 TX and 3x1RX in Urban Environment Considering Multiple Effects

Step 1: Create a new **GTD Project**.

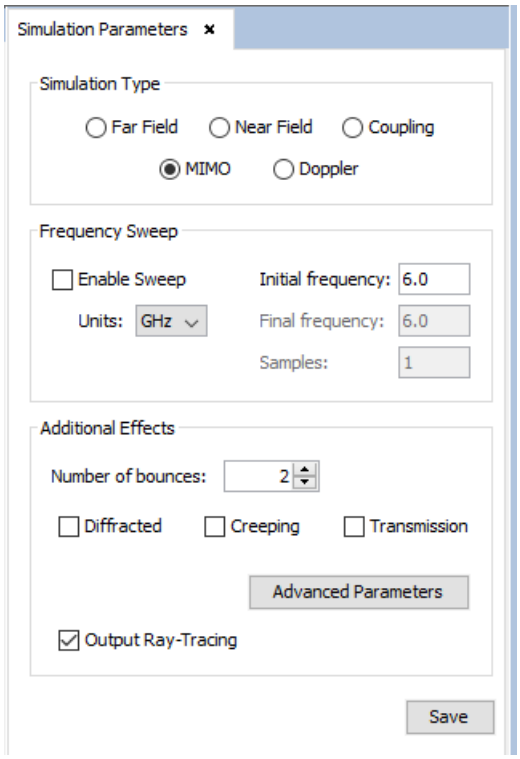
Open newFASANT and select **File - New option**.



Select GTD option on the previous figure and start to configure the project.

Step 2: Set the **simulation parameters** as shown.

Select **Simulation - Parameters** option, click on the **MIMO** button and set the parameters, **increasing the number of bounces**.



Step 3: Import the geometry.

In order to import the desired geometry, select **Geometry – Import**, select the `city.igs` file and open it.

Step 4: Create the **transmitter array**.

Select **Antenna – Dipole – Dipole Array** and set the parameters.

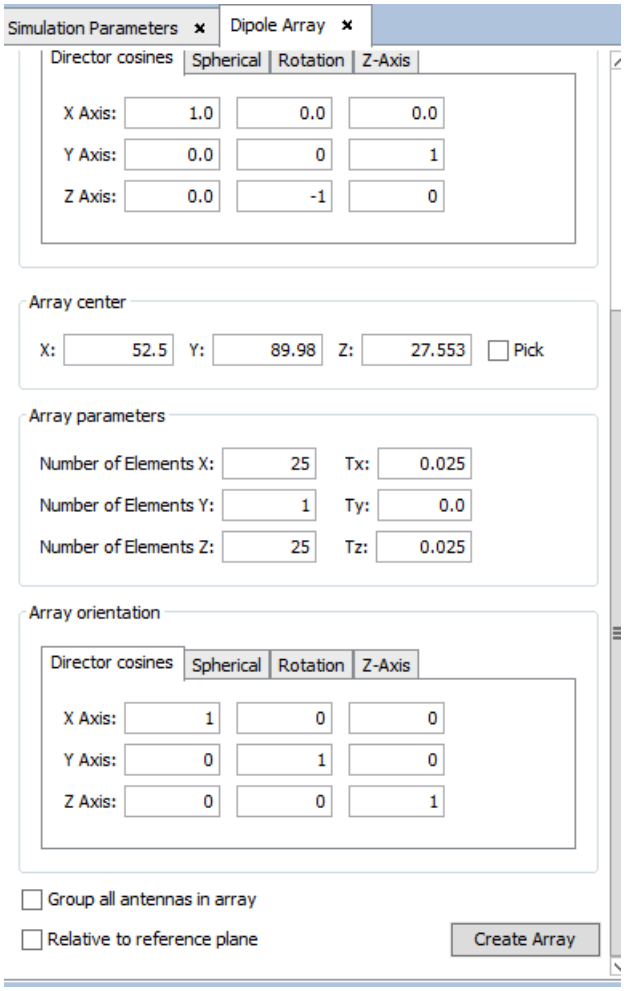
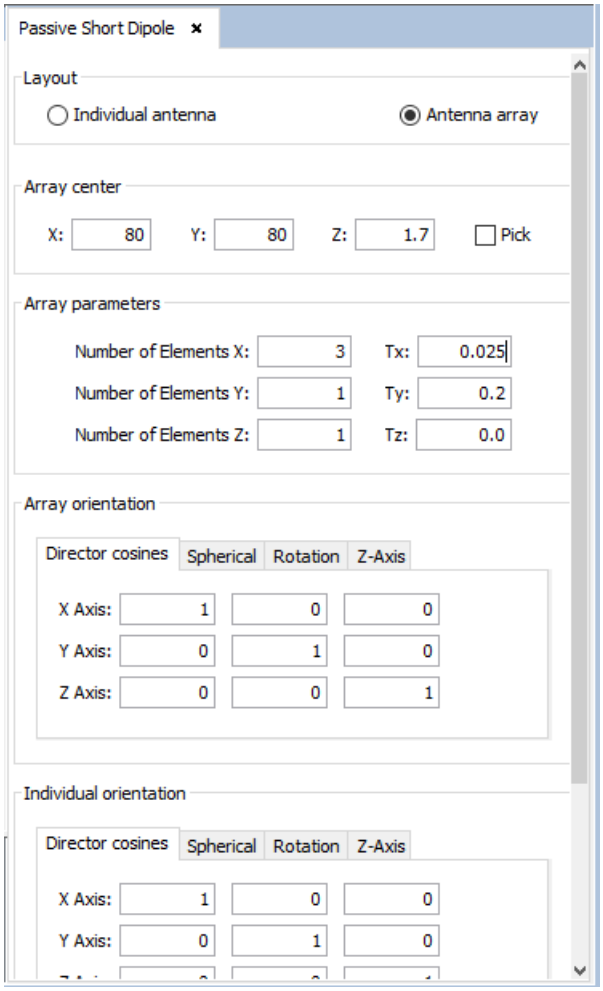


Figure 747: A spacing of 0.025 m. between array elements is chosen because is equivalent to a spacing of 0.5 lambdas at 6 GHz.

It is important that the **Group all antennas in array** option is left **unchecked**.

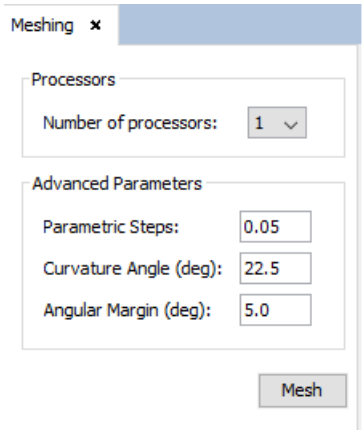
Step 5: Create the receptor array

Click on **Output – Coupling – Short Dipole** and select the **Antenna array option**. Set the parameters as shown.



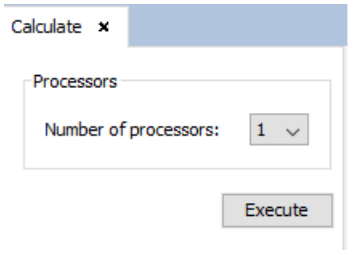
Step 6: Meshing the geometry model.

Select **Meshing - Create Matrix Visibility** to open the meshing configuration panel and then set the parameters as show the next figure.



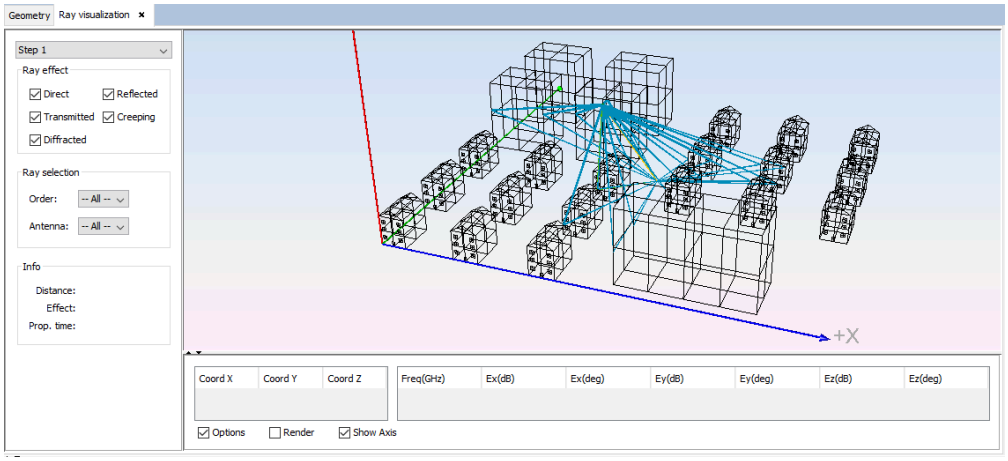
Step 7: Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

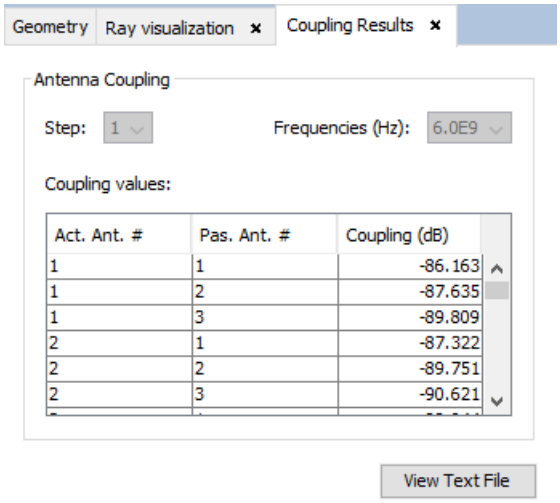


Step 8: Show Results.

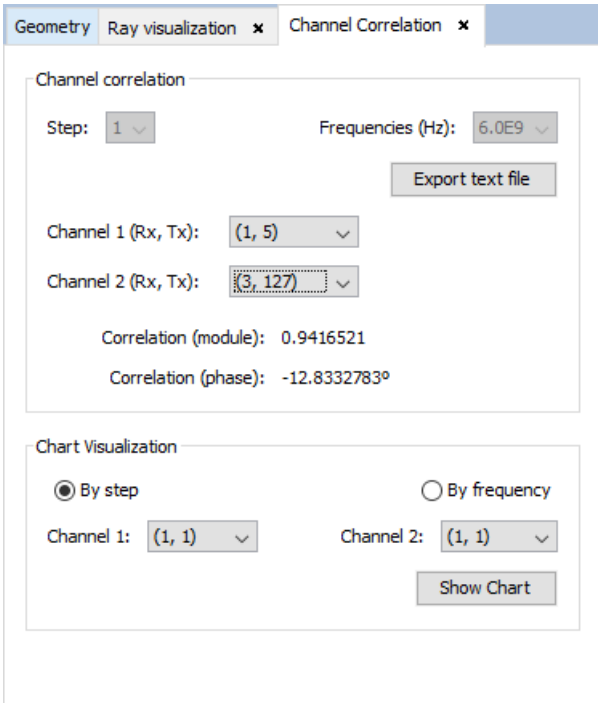
The **ray tracing** can be visualized by clicking **Show Results – View Ray**.



The coupling between the transmitter and the receptor antennas can be visualized by clicking Show Results – Coupling – Antenna Coupling.



The channel correlation can be visualized by clicking Show Results – MIMO Channel – View Channel Correlation.



In order to be able to see the overall capacity of the MIMO system, the user needs to specify some parameters.

When the user selects the option Show Results – MIMO Capacity – Capacity Post-process, the following panel is shown:

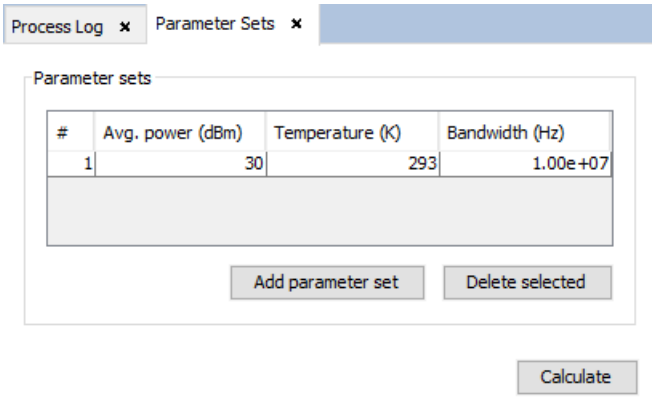
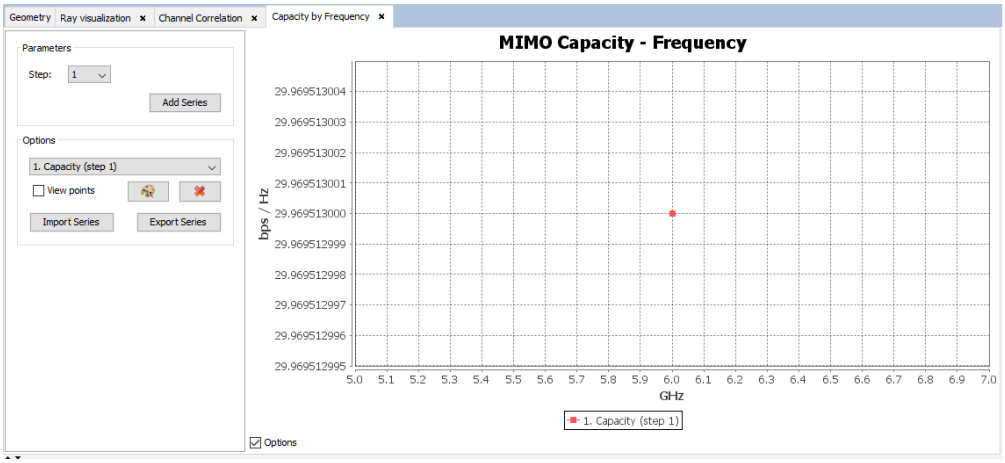


Figure 748: Capacity post-process panel

In this panel, the user has to specify the transmitted power per antenna, the temperature, and the bandwidth. In this example, the user has to set up the values shown in the above figure.

The capacity at the selected frequency can be visualized by clicking **Show Results – MIMO Capacity – View Capacity by Frequency.**



7.14.5 Using Multipole in GTD Module

Examples about how to compute array coupling in Near-Field and Far-Field using multipoles and GTD in newFASANT are presented in this section.

Using Multipole in newFASANT to Compute Mutual Coupling in Near-Field

A training example to compute the array coupling in Near-Field using multipoles and GTD module in newFASANT is presented.

The first step is to calculate the multipole using the MoM module and export it to be used to compute the array coupling in Near-Field. You can find it on the next link:

- Multipole Array Simulation

In order to generate the multipole file, an array close to a sphere is created and it will be used as the passive antenna in this training example.

Note that in version 6.3 only is able to compute the coupling with only one element of the passive array. In new improvements, a more general formulation for the coupling between arrays will be stated. We explode all the array elements except this one selected for coupling calculation in order to get only one antenna in the array (other elements will be considered as passive scatters).

In this case, the exploded array elements are grouped and the simulation is selected as shown.

The size of the multipole is set up to 2*wavelengths, therefore the multipole can be used accurately for distances greater than 2*((2*0.1)**2)/0.1=0.8 m.

Then, the simulation is meshed, calculated and saved the multipole generated.

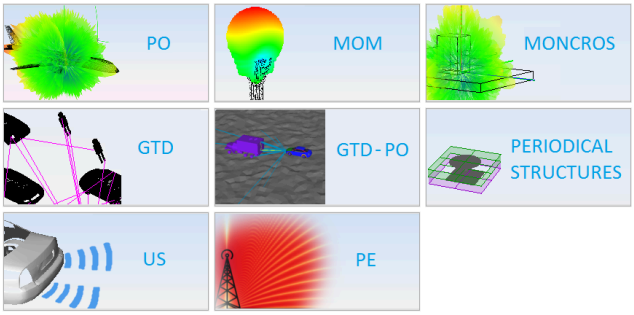
More information about how to do a multipole simulation and how to export this file are in the next links:

- Multipole Simulation
- Export Multipole File

Once the multipole has been created, the GTD project can be started.

Step 1: Create a new GTD Project.

Open newFASANT and select 'File' --> 'New' option.



Select 'GTD' option on the previous figure and start to set up the project.

Step 2: Create the geometry model. To obtain more information about geometries generation see Geometry Menu.

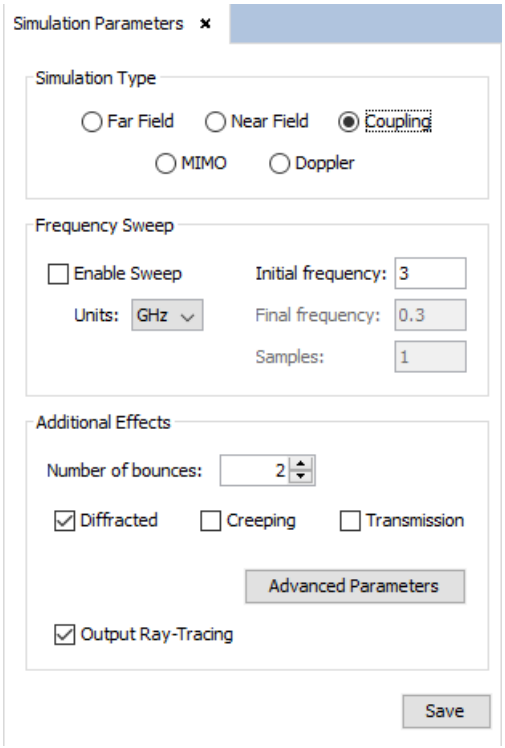
Execute 'plane' command writing it on the command line and set up the parameters as the next figure shown when the command line asks for it.

Execute 'box' command writing it on the command line and set up the parameters as the next figure shown when the command line asks for it.

```
command> plane
First corner of plane [x y z]: -6 -6 -1
Plane size [width depth]: 12 12
command> box
First corner of base [x y z]: 3 -1 -0.5
Base size [width depth]: 2 2
Height [double]: 2
```

Step 3: Set up Simulation Parameters.

Select 'Simulation' --> Parameters option on the menu bar and the following panel appears. Set up the parameters as the next figure shows and save it.

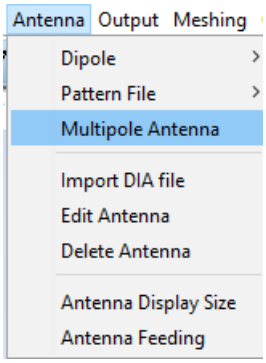


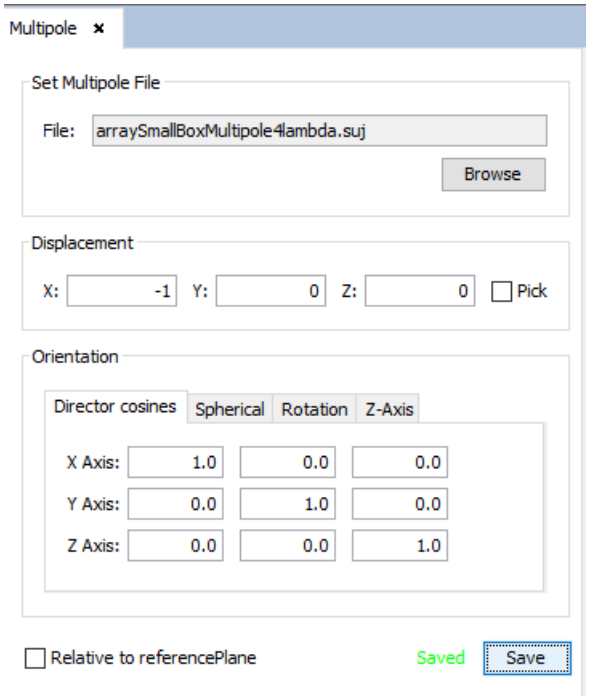
Note: It is important to select 'Output Ray-Tracing' to be able to visualize the ray-tracing.

Step 4: Set up the active antenna.

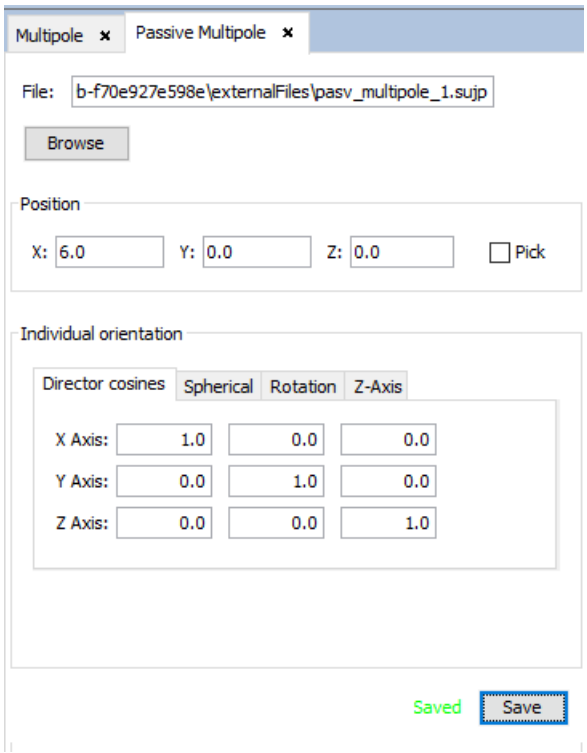
Select 'Antenna' --> 'Multipole Antenna' and select the arraySmallBoxMultipole4lambda.suj file and locate the antenna at (-1.0, 0.0, 0.0).

Note the file arraySmallBoxMultipole4lambda.suj is the multipole file obtained in the Multipole Array Simulation.



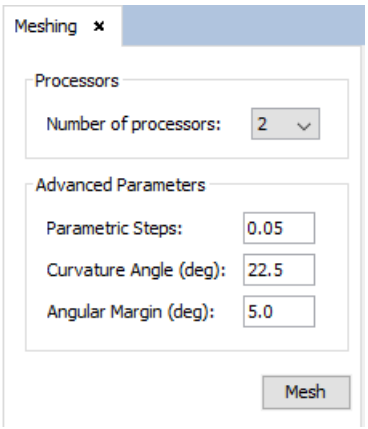


Step 5: In the output menu, the multipole option is selected, input the multipole of the array near sphere of the previous example (only one element is active). Select the array2oneDipole.suj file and locate it at (6.0, 0.0, 0.0).

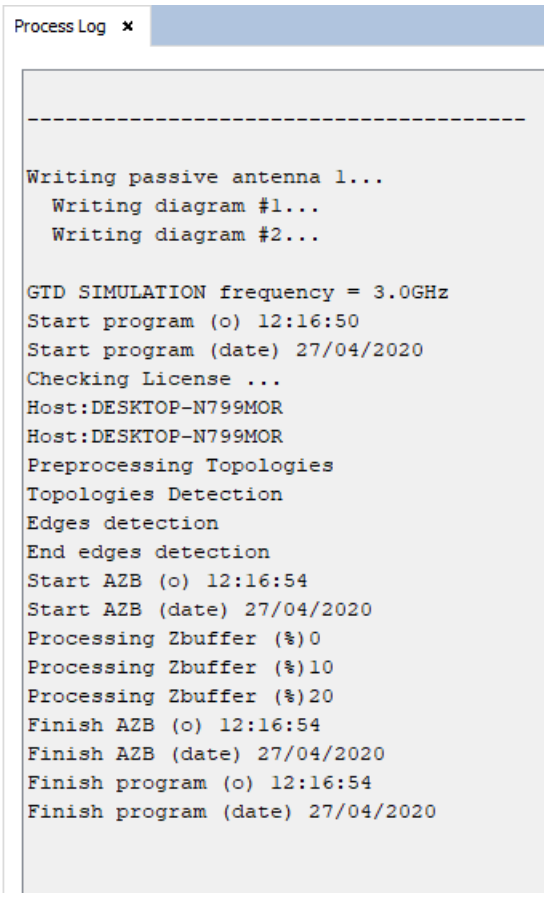


Step 6: Meshing the geometry model.

Select 'Meshing --> 'Create visibility matrix' and select the number of processors.



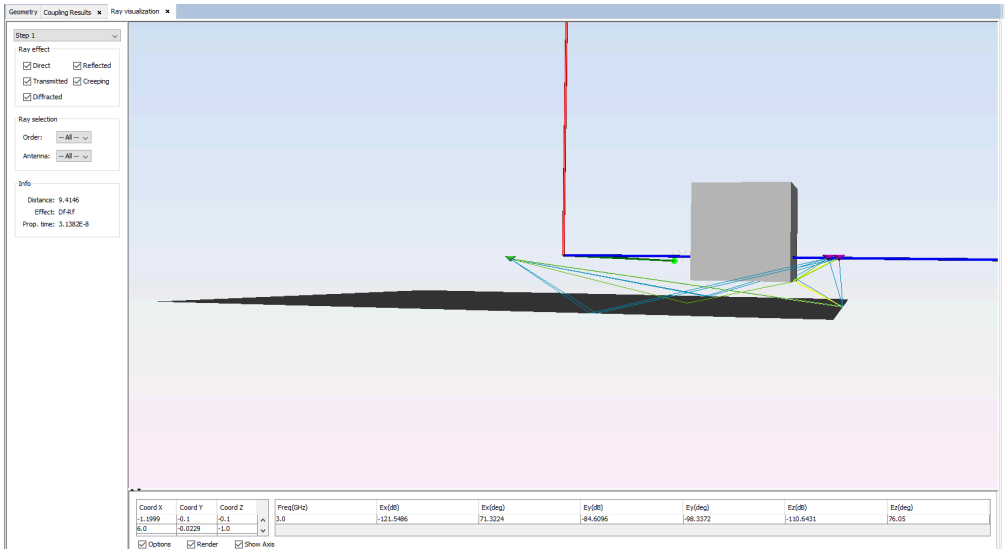
Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.



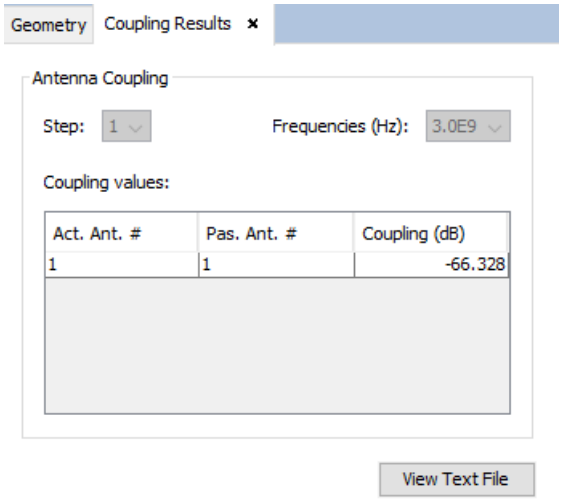
Step 7: Execute the simulation.

Select 'Calculate --> 'Execute' and introduce the number of processors to be used.

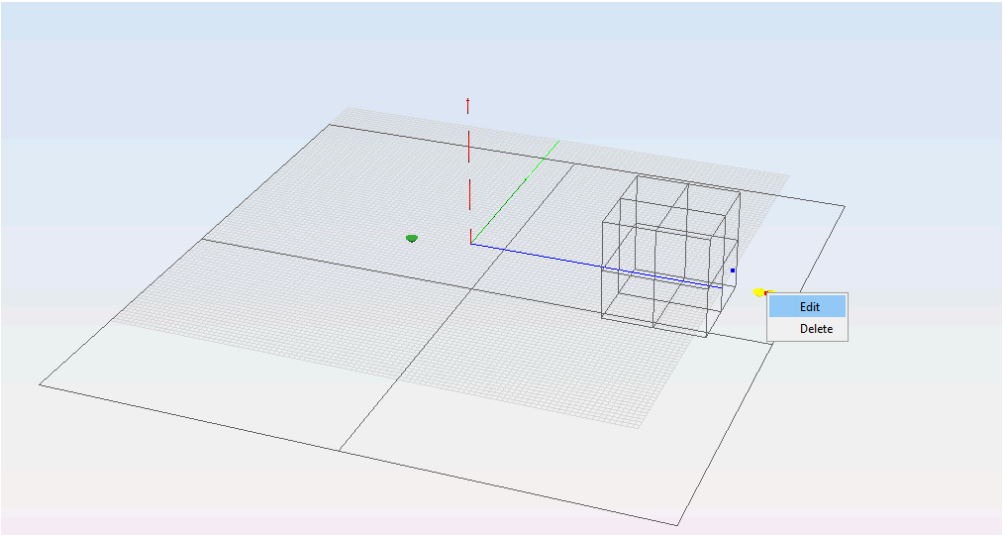
Step 8: After obtaining the results, choose view ray from show results. It is possible to visualize the order 1 ray. It is possible to represent the ray-tracing interacting with the geometry.



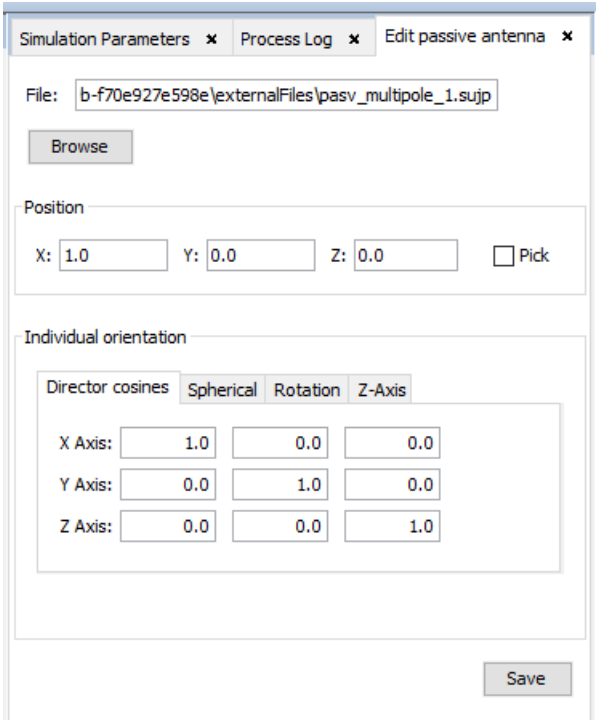
Step 9: Coupling.



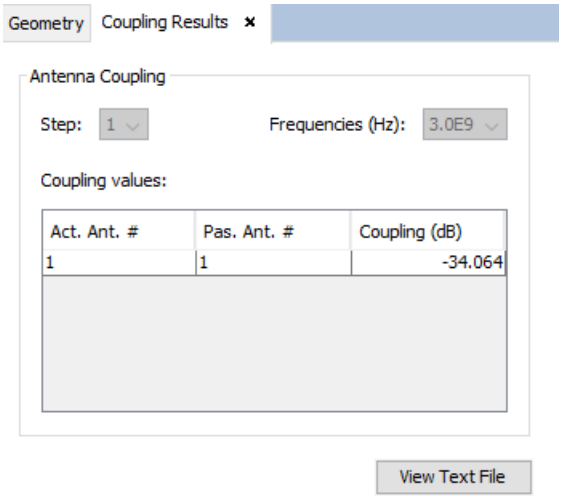
Step 10: In order to modify the location of the passive multipole, the user has to click on the cones with the right button of the mouse, and then the click on 'Edit', as is shown in the next figure.

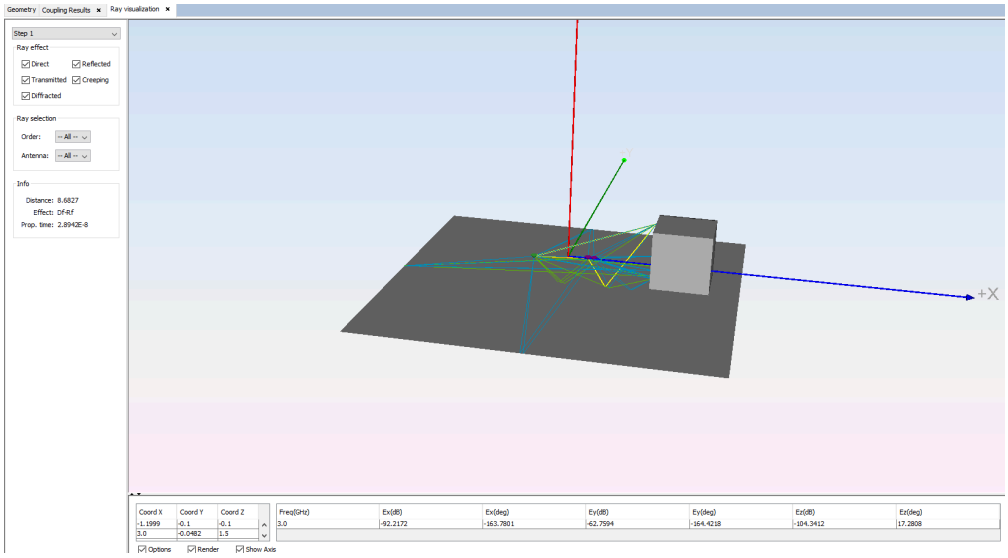


Step 11: Now, the passive multipole is located at (1.0, 0.0, 0.0).



Step 12: A new coupling value and a new ray-tracing between the active array and the dipole in the passive array are obtained.


Geometry
Coupling Results
Antenna Coupling
Step: 1
Frequencies (Hz): 3.0E9
Coupling values:
Act. Ant. #
Pas. Ant. #
Coupling (dB)
1
1
-34.064
View Text File


Geometry
Coupling Results
Ray visualization
Step 1
Ray effect
Direct
Reflected
Transmitted
Creeping
Diffracted
Ray selection
Order:
Antenna:
Info
+X

PO

8

PO is used for radar cross section analysis of electrically large and complex targets using physical optics (PO).

This chapter covers the following:

- 8.1 File Menu (p. 778)
- 8.2 Edit Menu (p. 779)
- 8.3 View Menu (p. 780)
- 8.4 Geometry Menu (p. 781)
- 8.5 Materials Menu (p. 782)
- 8.6 Simulation Menu (p. 783)
- 8.7 RCS Menu (p. 789)
- 8.8 Output Menu (Alt +O) (p. 790)
- 8.9 Meshing Menu (p. 792)
- 8.10 Calculate Menu (Alt + C) (p. 796)
- 8.11 Show Results Menu (p. 798)
- 8.12 Tools Menu (p. 804)
- 8.13 Help Menu (p. 805)
- 8.14 Training Examples (p. 806)

8.1 File Menu

For information about the File menu, refer to File Menu.

8.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

8.3 View Menu

For information about the View menu, refer to View Menu.

8.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

8.5 Materials Menu

For information about the Materials menu, refer to Materials.

8.6 Simulation Menu

This menu has several options that let the user change the simulation settings for the current project.

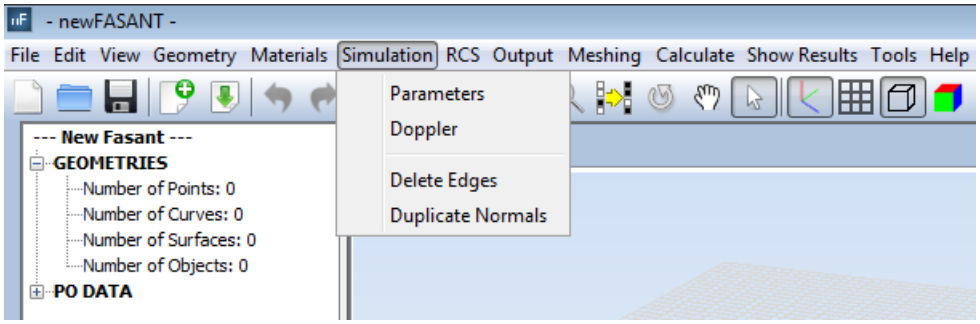


Figure 749: Simulation Menu

The following options are available:

- Parameters this option is used to modify simulation frequency, bounces, and the ground plane.
- Doppler this option is used to apply Doppler shifts to objects that are part of the simulation.
- Delete Edges this option is used to discard selected edges during the simulation.
- Duplicate Normals this option tells the simulation process to consider both sides of selected surfaces.

8.6.1 Parameters Menu

Parameters are defined by the user for the simulation on the right-side of the window where the user can select the parameters to be defined. The user may enable a single frequency simulation, or enable a swept simulation by left-clicking the Enable Sweep box. The user may then define the Initial Frequency, Final Frequency, and the number of frequency points (Samples) for the simulation.

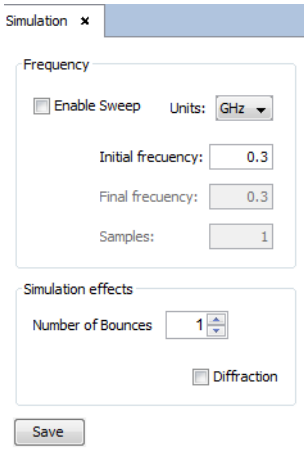


Figure 750: Simulation Parameters panel

- **Frequency**. To set the simulation frequency. The following parameters may be modified:

- *Enable Sweep*
  select this option to simulate a range of frequencies.

  Units
  to set the magnitude of the frequency. Hz, kHz, MHz, and GHz are available.

  Initial frequency
  the first step in the frequency range, in the selected **Units**. If the frequency **Sweep** is disabled, this value coincides with the simulation frequency.

  Final frequency
  the last step in the frequency range, in the selected **Units**. This parameter may be only modified when the frequency **Sweep** is enabled.

  Samples
  number of frequency samples to be analyzed on the frequency **Sweep**. This parameter may be only modified when the frequency **Sweep** is enabled.

Simulation effects.
In this section, the user can select the order of the effects to take into account in the simulation for the computation of the fields. The order of the effects relates to the combination of simpler effects. For example, order 1 contains only direct, reflected and the diffracted field (if the user has checked the diffraction check box). Order 2 contains the combination of the simple effects reflected-reflected, reflected-diffracted, diffracted-reflected, etc.

NOTICE: Click on **Save** button before closing this window to confirm the changes.

8.6.2 Doppler

This option is used to consider the Doppler shift in the computation of the RCS. On the right-side of the window the user can define the parameters, as shown in the next Figure:

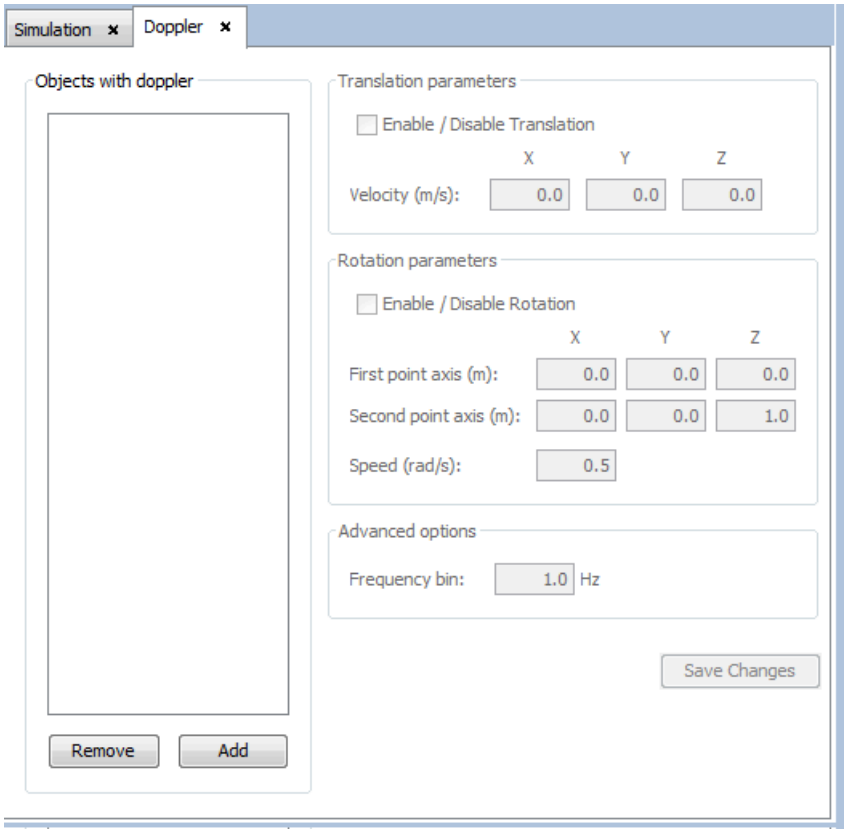


Figure 751: Doppler Parameters panel

In this panel, we can add Doppler effect to objects in the geometry. To do this, select an object from the geometry view and press the **Add** button. The selected object will be added to the list of objects. It is possible to remove the Doppler effect from an object by selecting it from the list and pressing the **Remove** button.

- **Translation Parameters**. Any object in the list could have a translation movement. Using this panel the user can define the linear velocity of the object. The Enabled/Disabled Translation check box must be selected.
- **Rotation Parameters**. Any object in the list could have a rotation movement. The Enabled/ Disabled Rotation check box must be selected. Using this panel the user can define:
- ◦ **First point axis** cords of the first point axis.
  - ◦ **Second point axis** cords of the second point axis.
  - ◦ **Speed** rotation speed
- **Advanced Options.** The user can also define the source velocity, the observation velocity, and the frequency bin.

Once the parameters of the movement of one object have been defined, it is necessary to click on **Save Changes** button in order to save this information and to assign it to the object selected.

NOTICE: Click on **Save Changes** button before closing this window to confirm the changes.

8.6.3 Delete Edges

This option is used to avoid considering the diffraction effect due to edges contained in the simulation. The selection of edges to be discarded from the simulation can be performed in different ways, as shown in the next Figure:

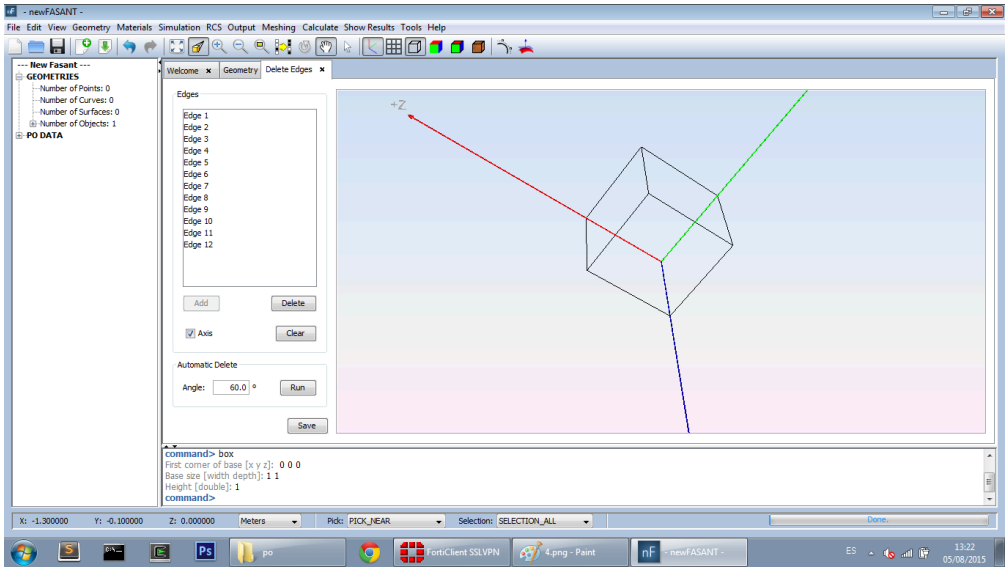


Figure 752: Delete Edges

Automatic Define the angle of the edge to delete and left click on run. All of the edges with an angle smaller than the specified threshold angle are selected and shown in the window. For example, in the previously created box object, the edges represented in the next Figure are detected when an angle of 90° is defined in the Automatic Edges Deletion window.

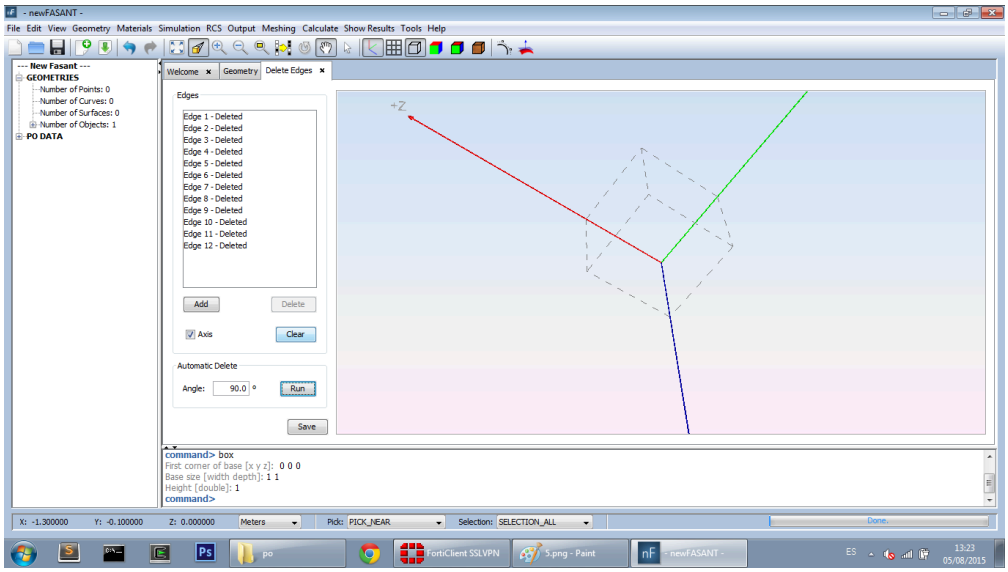


Figure 753: Delete edges automatic

Manual: Select edges from the list and left-click on delete. All the edges detected in the geometry are shown in the Selector Point window. Edges can be selected to not be considered for the simulation by

left-clicking on them and pressing the delete button. The user can confirm the deletion with the Save changes button. In the next Figure, we can see how the deleted edges are represented by dashed lines.

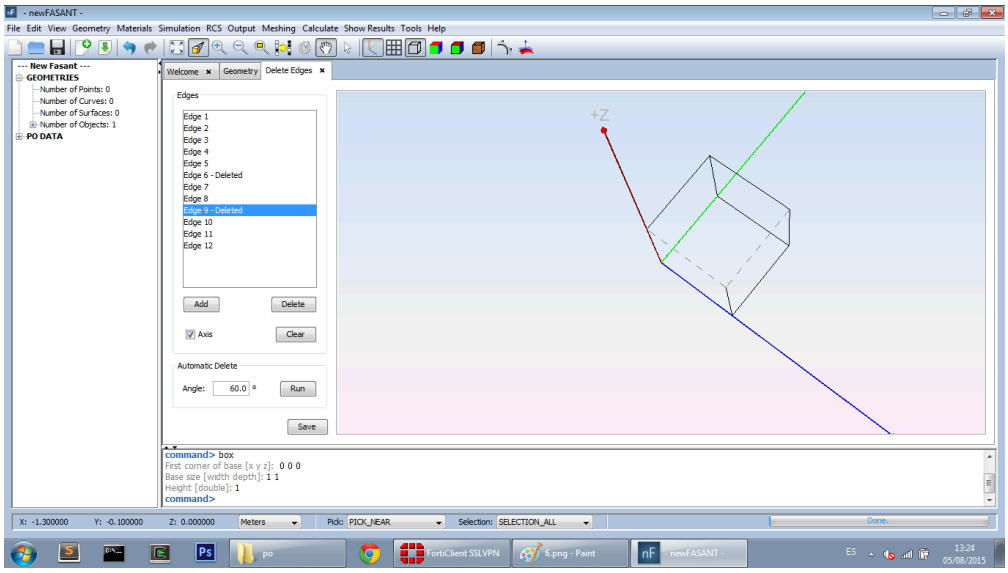


Figure 754: Delete Edges manual

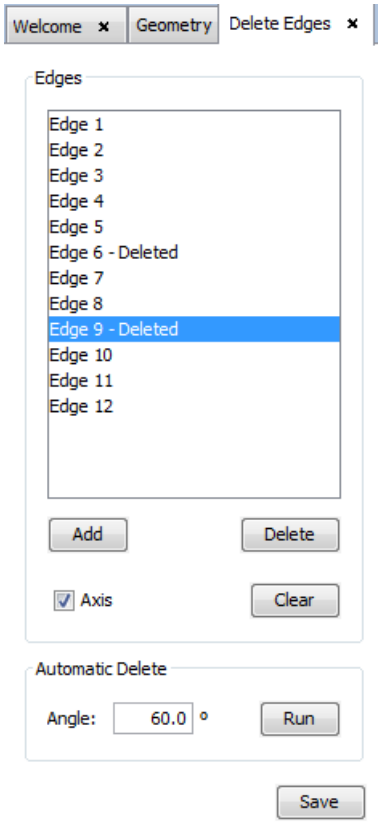


Figure 755: Delete Edges options panel

NOTICE: Click on **Save** button before closing this window to confirm the changes.

8.6.4 Duplicate Normals

In the PO module, the surfaces are only considered in the region pointed by its normal vectors. This option is used to consider both sides (according to with the normal vector directions) of the selected objects. It is recommended for use when the user wants to calculate the RCS of open surfaces in the two normal vector possible directions. As shown in the figure below:

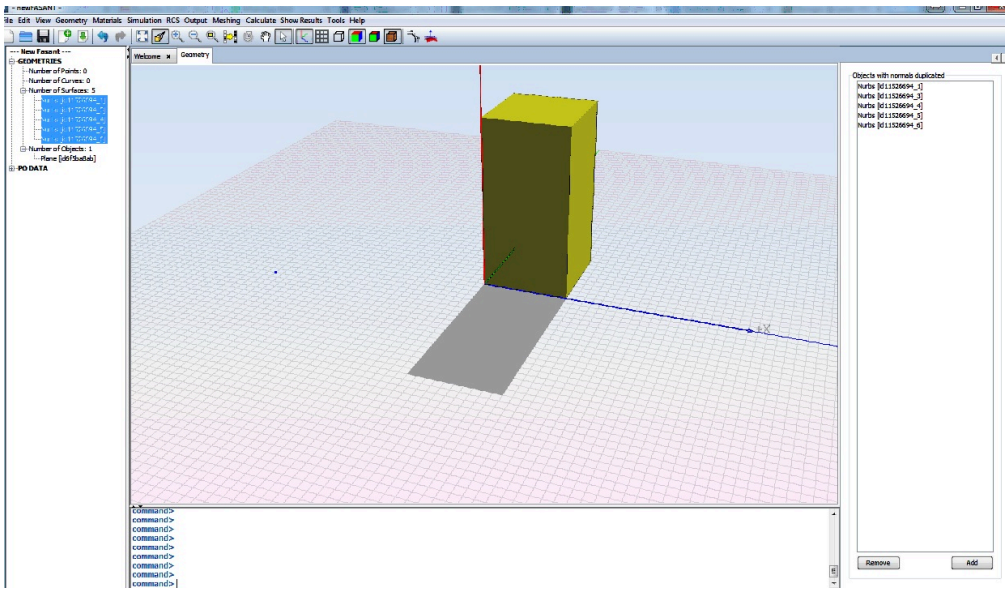


Figure 756: Duplicate Normals

8.7 RCS Menu

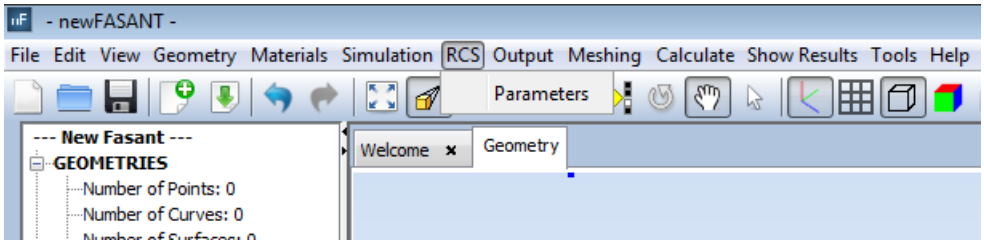


Figure 757: RCS menu

8.7.1 Parameters

In this window, the user selects the type of RCS calculation to be performed.

The following setting is available:

- **Monostatic RCS** in order to compute the Monostatic RCS, in this case, the user can enable **speed-up**. Speed-up option will reduce the analysis time for problems that have several incident angles.
- **Bistatic RCS** in this case, the user must define the angle of incidence (by default theta and phi are 0.0º).

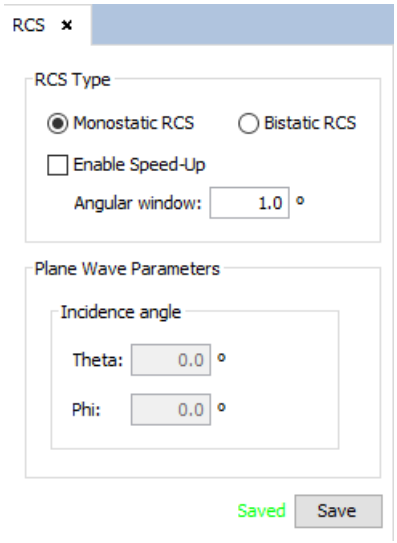


Figure 758: RCS panel

Click on **Save** button before closing this window to confirm the changes.

8.8 Output Menu (Alt +O)

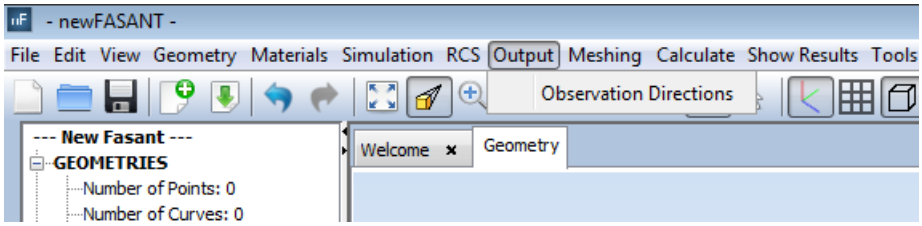


Figure 759: Output Menu

8.8.1 Observation Directions

The user needs to configure the directions of observation where the electrical far field is to be calculated. The user selects a combination of angular sweeps in Theta or Phi, choosing in each case the number of samples for the variable angular component and the number of cuts to be simulated between the initial and final cuts, for the fixed angular value defined by the user. All the values for Theta and Phi are entered in degrees.

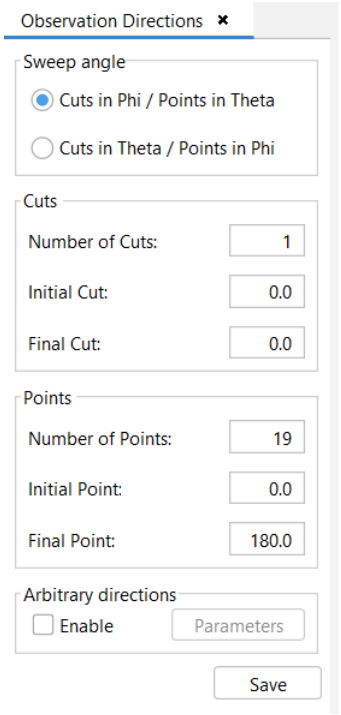


Figure 760: Observation Directions panel

The following settings are available:

- **Cuts in Phi / Points in Theta**: the user selects to compute cuts in Phi and angular sweeps in Theta.

- **Cuts in Theta / Points in Phi**: the user selects to compute cuts in Theta and angular sweeps in Phi.
- **Number of cuts**: defines the number of samples in the angular cuts where the user wants to compute the RCS.
- **Initial Cut**: define the cut initial value (in degrees).
- **Final Cut**: define the cut final value (in degrees).
- **Number of Points**: defines the number of samples in the angular sweep where the user wants to compute the RCS.
- **Initial Point**: define the angular initial value (in degrees).
- **Final Point**: define the angular final value (in degrees).

Additionally, the user can select *Enable Arbitrary Directions* and click on *Parameters* button, then in the menu shown in Figure 761:

- the user can import the information from a text file and show/edit the arbitrary directions (theta/phi angles) where the RCS computation wants to be computed
- the user can edit the arbitrary directions (theta/phi angles) where the RCS computation wants to be computed and export them to a text file.

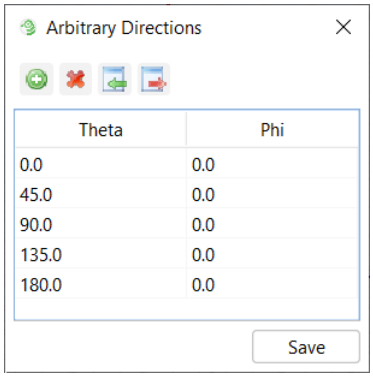


Figure 761: Arbitrary Directions panel

Note: Click **Save** before closing this window to confirm the changes.

8.9 Meshing Menu

In this section, all the features available in the Meshing menu of the PO module are explained. The options available in the Meshing menu are shown in the next Figure.

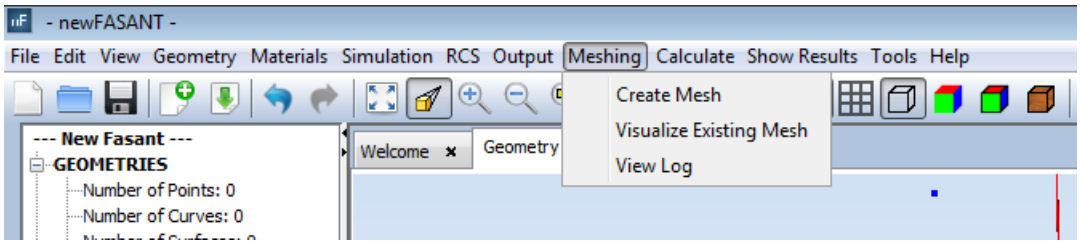


Figure 762: Meshing Menu

In the following sections, the options for the Meshing Menu are described.

8.9.1 Create Mesh

In the PO module, the mesh generation is a required step that is performed immediately before simulating. In the Meshing Parameters window, represented in the next Figure, the following options are available:

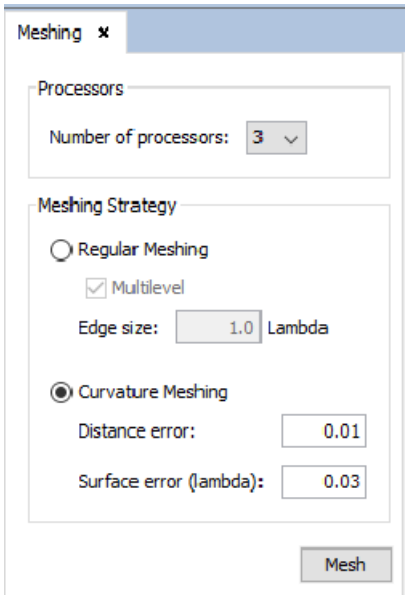


Figure 763: Meshing Parameters panel

In the Meshing Options panel the following options are available:

- **Number of processors**: To set the number of processors for the meshing process.
- **Meshing Strategy**
  - **Regular Meshing**: The size of the generated elements is selected by the user. Regular meshing is the common algorithm used in other modules such as MOM. It provides a mesh of

triangles of similar sizes with a distribution as uniform as possible. It enables the user to set the exact size edge of the triangles to be generated, given in lambda units. However, a huge number of mesh elements may results when the geometry contains large surfaces.

- **Multilevel**: Use Multilevel to generate the mesh automatically in several sequenced steps. This option is usually more efficient (in terms of runtime) than the common mesh generation option, so it is selected by default. However, there may be minor differences between the meshes obtained with and without the Multilevel mode.
- **Edge size**: The desired size for the edge of the elements, in wavelengths.

◦ **Curvature Meshing**:

Using curvature meshing we get the meshed geometry defined in terms of triangles. This geometry should be very similar to the original geometry. We can force that the meshed geometry be close to the original one using the surface error (in wavelengths). In the mesher we have default values for this magnitude but the user can change it. The mesher tries to minimize the number of triangles of a meshed geometry such that the distance from any point of this geometry to the original one is smaller than the "selected surface error".

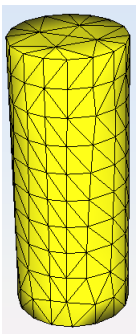
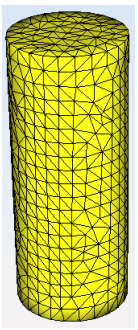
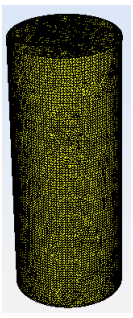
If we have a cube, say the original geometry is defined by six flat quadrangles the meshed geometry is composed by twelve triangles. In this case the distance of any point of the meshed geometry to the original one is null, say both geometries are identical for every frequency.

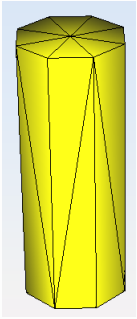
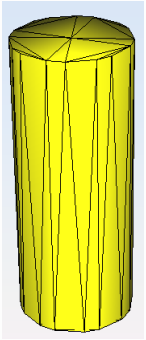
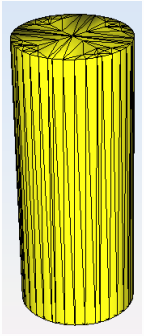
If an sphere of 1 m of radius at 300 MHz the "selected value" using the defaults values is 0.08*wavelength = 8.0 cm. The mesh is composed by 104 triangles. If you increase the frequency we obtain the same mesh until about 0.7 GHz. Due to topological restriction the number of elements of a mesh of triangle can not increases continuously, say we can not go from 104 triangles to 105 triangles for the sphere (next for the sphere can be 174 triangles).

The **Distance error** error is other parameter of the meshing by curvature. It only applies to flat facets. The longer edge of the planar surface is split by this parameter. For instance if you set 10 for the cube case you get $12*(10^2) = 1200$ triangles. By default we have 0.01, that is, if this parameter is lower than 1.0 not division of flat facets is consider.

The table below shows a comparison of meshing criteria with a different configuration for meshing a cylinder with a radius of 0.25 m. and height of 1 m. at 3 GHz.

Table 7: Comparison of meshing criteria.

	1 lambda	0.5 lambda	0.01 lambda
Regular			

	0.2 surface error	0.1 surface error	0.01 surface error
Curvature			

8.9.2 Visualize Existing Mesh

It is possible to visualize an existing mesh already generated by searching for mesh files (msh file extension), as shown in the following Figure. This option is only useful to visualize meshes, but not for simulations, unless the visualized mesh matches the one used in the current simulation project.

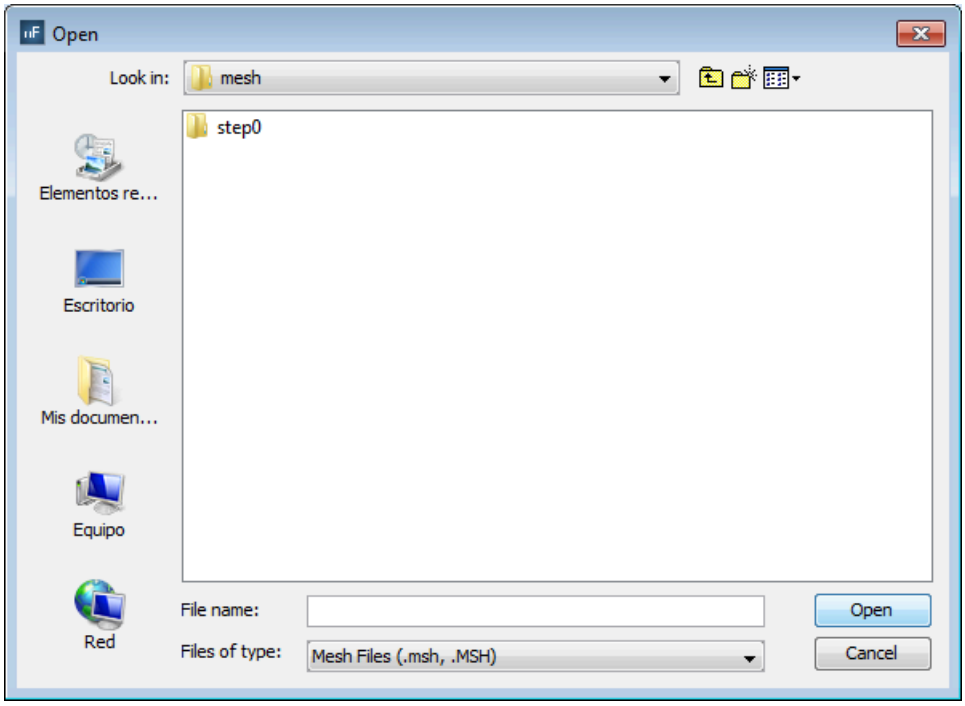


Figure 764: Visualize Existing Mesh

8.9.3 View Log

When some special operation has been performed on the input geometry, some information about that process is written into a log file (`mesh_log.txt`) that can be seen by left-clicking on View Log button.

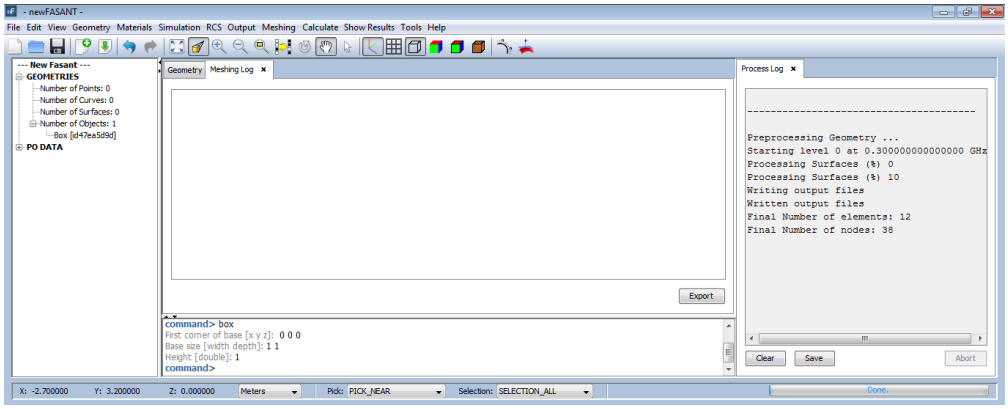


Figure 765: View Log window

8.10 Calculate Menu (Alt + C)

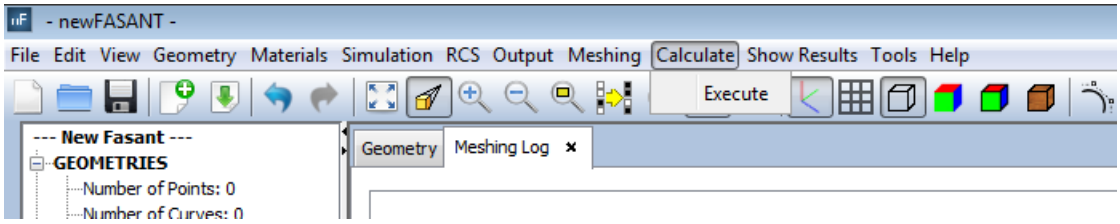


Figure 766: Calculate Menu

Before executing the simulation, the user needs to make sure that every simulation parameter has been entered by the user. If something is wrong with a user defined parameter, then a warning message is displayed to let the user know that it is necessary to correct one or more parameters for this simulation.

8.10.1 Execute

This option runs the simulation. The user can define the number of processors for solving this electromagnetic problem using the parallel version of newFASANT, where the user can choose between two parallelization models:

- **MPI** (Messages Passing Interface)
- **OpenMP** (Multi-Processing).

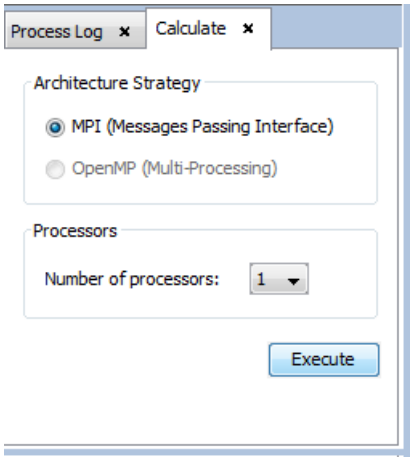


Figure 767: Calculate Menu panel

When the simulation starts, a new screen is shown:

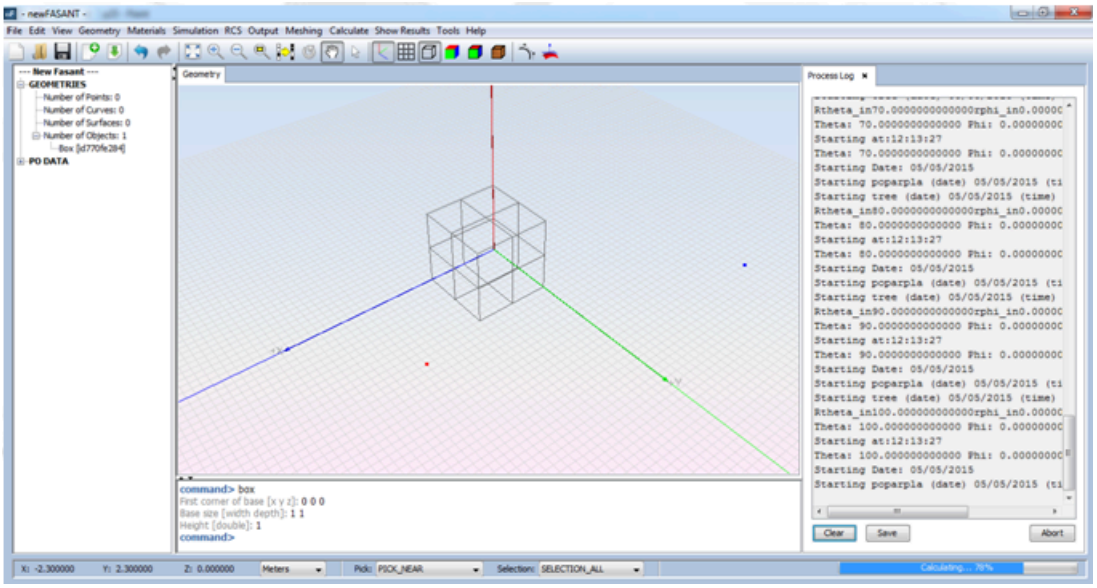


Figure 768: Processing screen

During a solution process, the screen reports the status of each phase of the calculation process date, number of iterations, CPU time for every phase of the solution process, etc.

Click on the **Save** button to save it, as shows the next Figure.

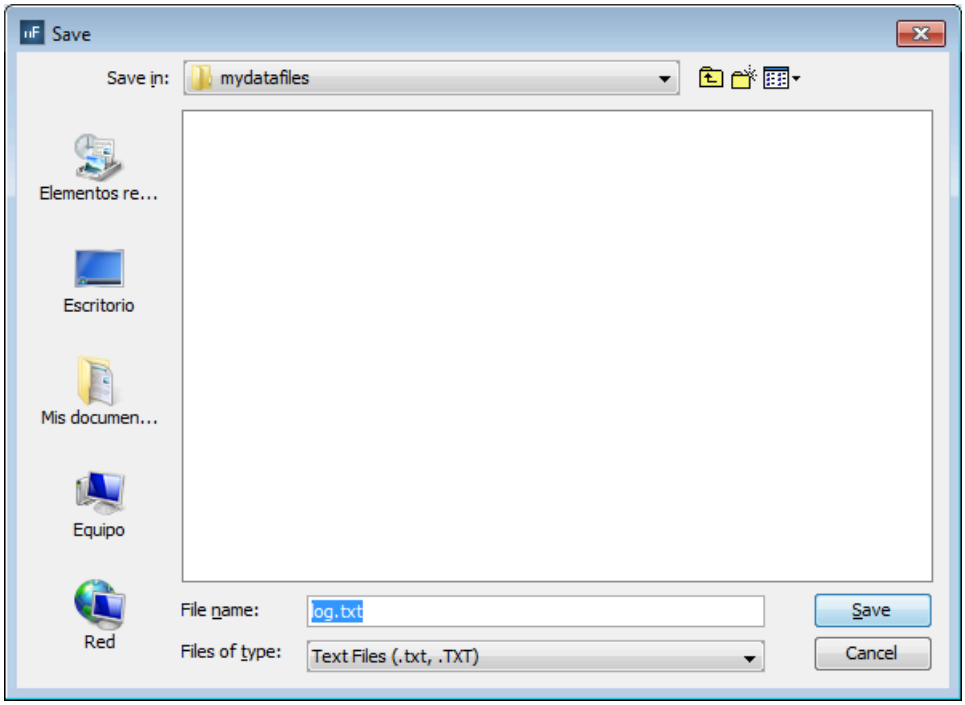


Figure 769: Processing Calculate

8.11 Show Results Menu

This menu allows the user to visualize the results of the simulation. The results are stored in external files that the user can access to obtain the information corresponding to that simulation.

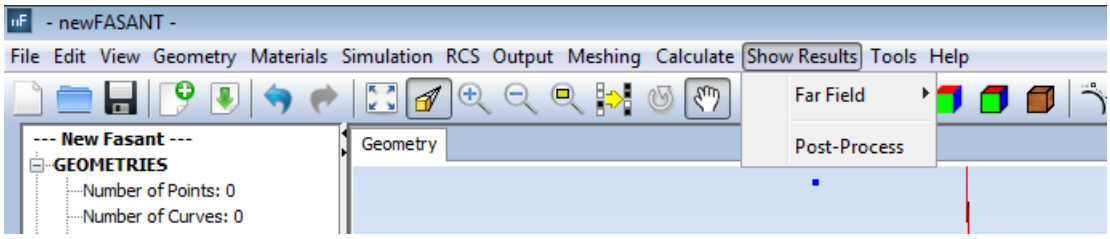


Figure 770: Show Result menu

Most of the plots show 2-D curves of a result field versus an input parameter. This type of graphics share additional features of zoom, axis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

8.11.1 Far field

Using this menu it is possible to analyze the amplitudes in the Far Field through the observation directions previously defined in the Output section.

The following options are available:

- View Cuts display a plot that relates the amplitude against a Theta cut or a Phi cut.
- View Cuts by Frequency displays a plot that relates the amplitude against a frequency sweep.
- View Cuts by Step displays a plot that relates the amplitude against a parametric step.
- View Text Files displays the numeric results obtained in the simulation process.

View Cuts

This command plots the amplitude or the phase of the RCS fields computed for each angular cut of the simulation geometry. Several cuts and frequencies can be plotted in the same graphic. To plot a graph, the user can select the RCS component, the frequency, and the cut. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a serie that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a data file written as a text file.

This panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against every possible frequency that has been simulated. More series can be added, by selecting component, step and spherical coordinates in the control panel that appears leftside to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Theta cut / Phi cut:** select one of the theta cuts or phi cuts that were previously added to the project.
- **Point:** select a particular observation point in the cut selected before.

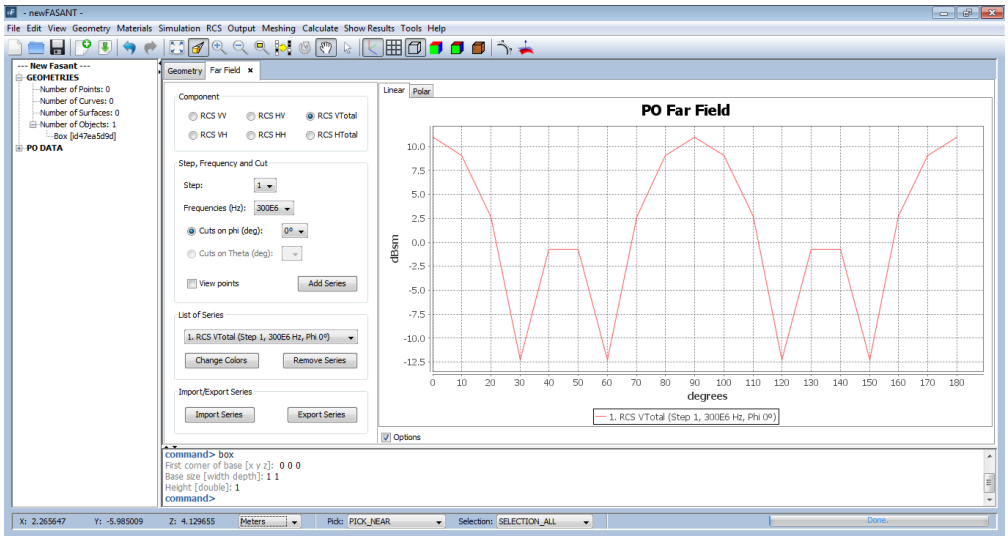


Figure 771: View Cuts Linear window

The user can also draw a polar plot with the same options used in the previous window.

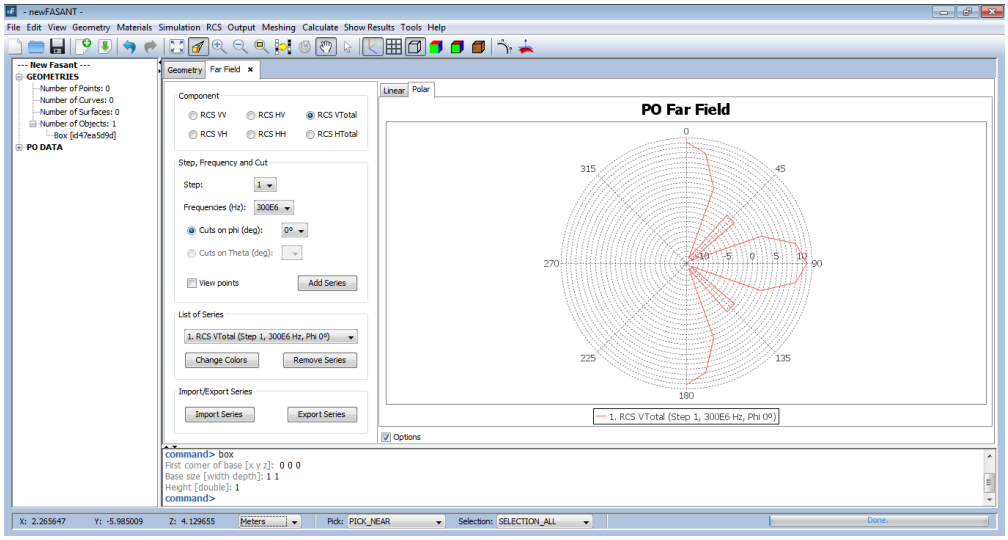


Figure 772: View Cuts Polar window

View Cuts By Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, cut and point in the sweep, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation Settings, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the Show Results > Far Field > View Cuts By Frequency menu, the panel presented in the figure below will appear.

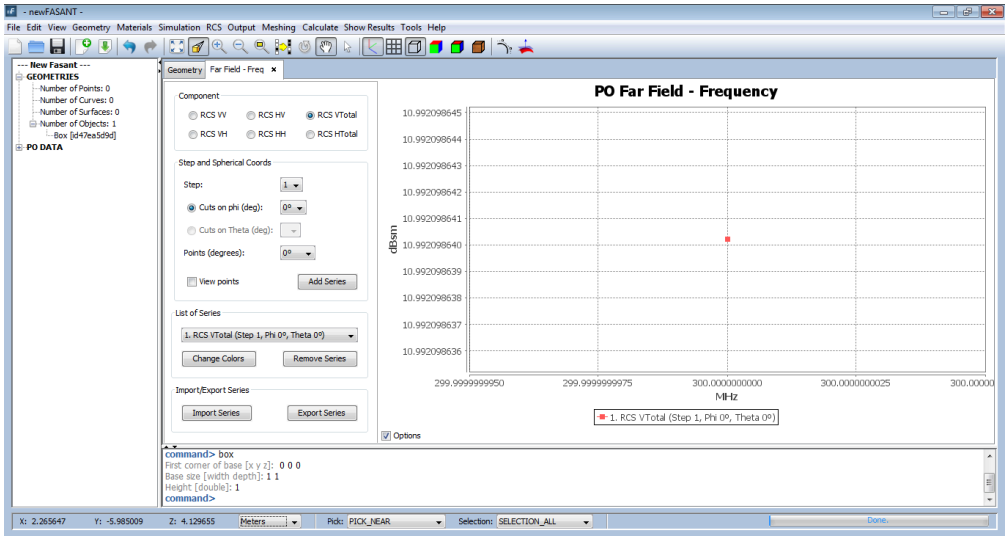


Figure 773: View Cuts By Frequency window

Similar to the behaviour of the standard Far Field chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against every possible frequency that has been simulated. More series can be added, by selecting component, step and spherical coordinates in the control panel that appears leftside to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Theta cut / Phi cut:** select one of the theta cuts or phi cuts that were previously added to the project.
- **Point:** select a particular observation point in the cut selected before.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Cuts By Step

This kind of plot lets the user analyze RCS measurements made by the simulation provess for a particular component, frequency, theta or phi cut and observation point against the parametric steps defined in the project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the Show Results > Far Field > View Cuts by Steps menu, the panel presented below will appear:

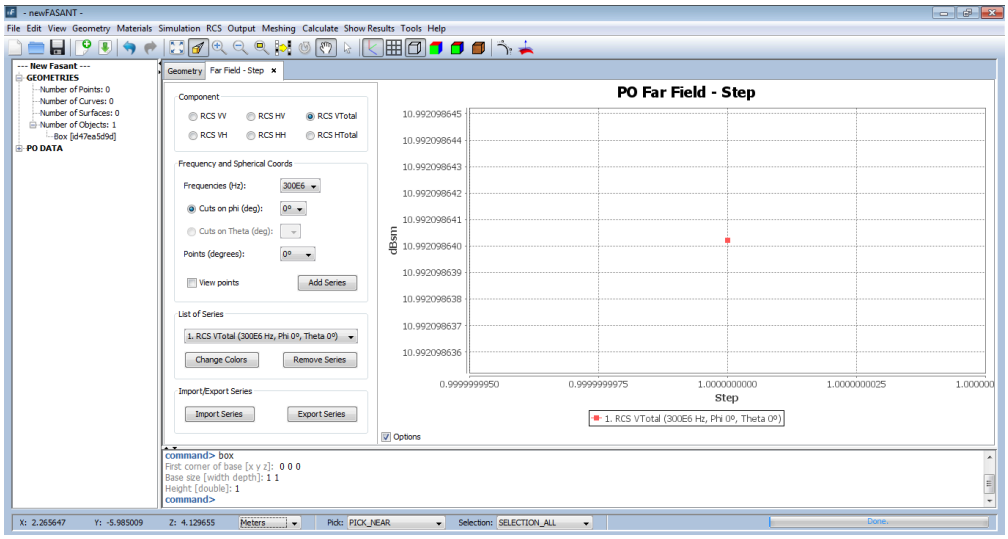


Figure 774: View Cuts By Step window

Similar to the behaviour of the standard Far Field chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against every possible frequency that has been simulated. More series can be added, by selecting component, step and spherical coordinates in the control panel that appears leftside to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Theta cut / Phi cut:** select one of the theta cuts or phi cuts that were previously added to the project.
- **Point:** select a particular observation point in the cut selected before.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Text Files

This option shows the simulation results as a text file. First,a panel should appear after selecting this option. Selecting this option will display a panel where the user should select a step and a frequency. To use this section, select a parametric step from the list, and select the frequency to get the results from and click on **OK** to load the data. A table will be prsented with the values for this frequency and step (see the next figure). To save the result click on the **Export** button below the table.

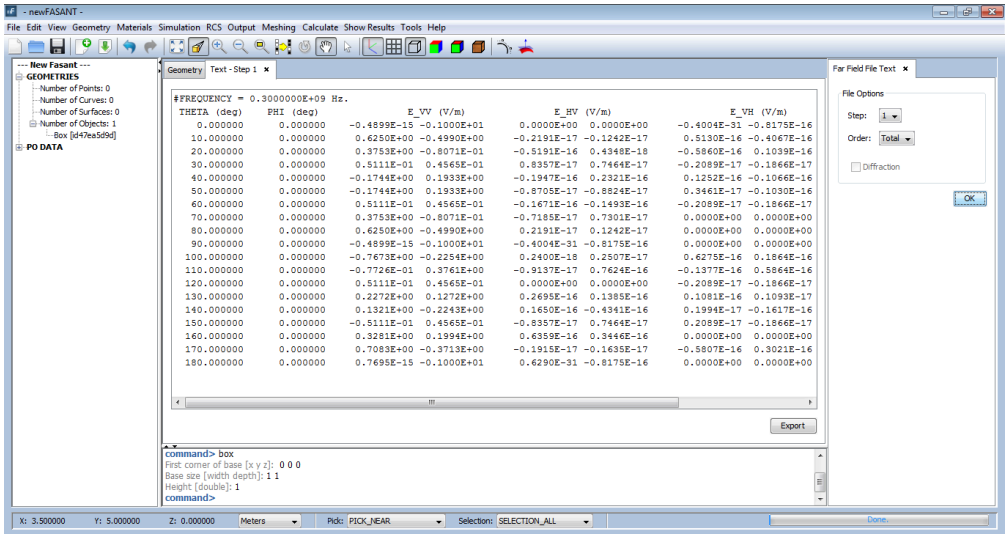


Figure 775: View Text Files window

8.11.2 Post-Process

For information about the Post-Process menu, refer to Post-Process.

8.12 Tools Menu

For information about the Tools menu, refer to Tools.

8.13 Help Menu

For information about the Help menu, refer to Help Menu.

8.14 Training Examples

8.14.1 Example 1

In this example, the Monostatic RCS of a plane is calculated. The normal vectors of the geometry are duplicated for this simulation.

Step 1

Start newFASANT.

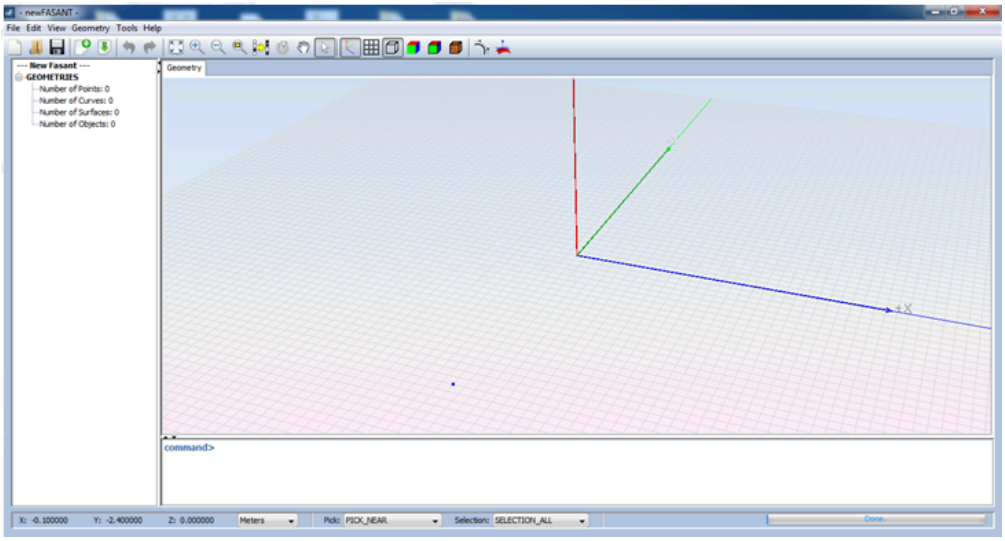


Figure 776: Start newFASANT window

Step 2

Select File and click on New.

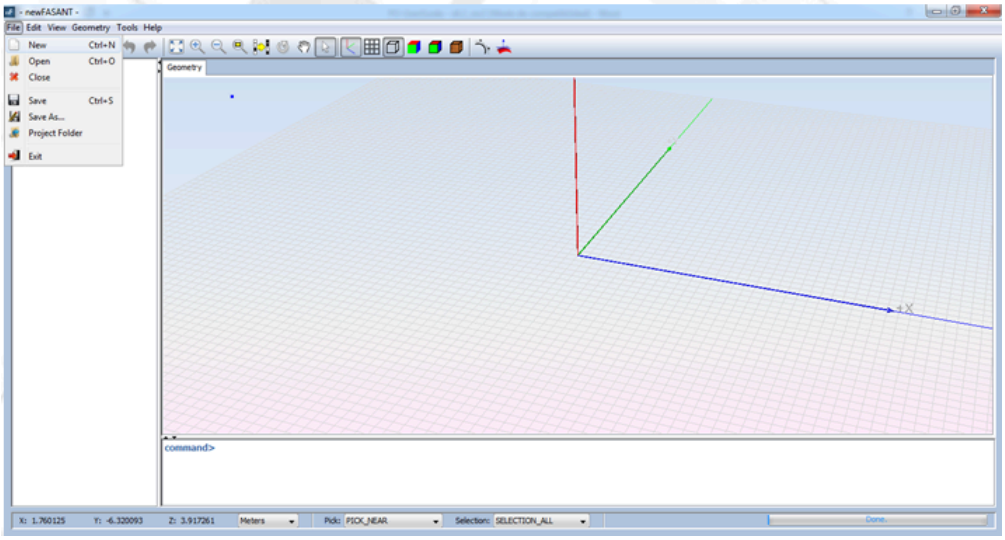


Figure 777: New File

Step 3

Select PO.

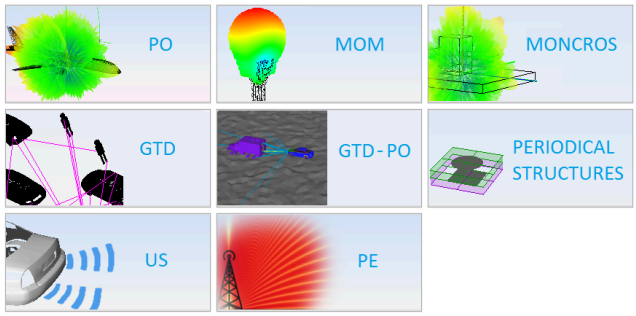


Figure 778: Method Type selection

Step 4

Click on Geometry → Surface → Plane, which introduces the first and the second point dialog as shown in the next Figure.

In this example the user enters the following parameters into the command line:

- First corner of plane: -0.5 -0.5 0
- Plane size [width depth]: 1 1

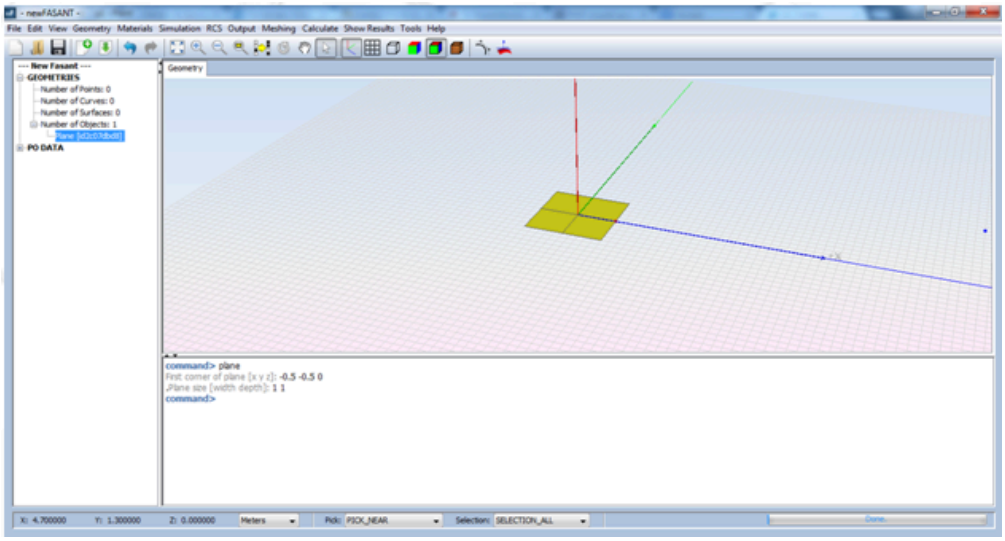


Figure 779: Create plane command

Step 5

Click on Simulation → Parameters.

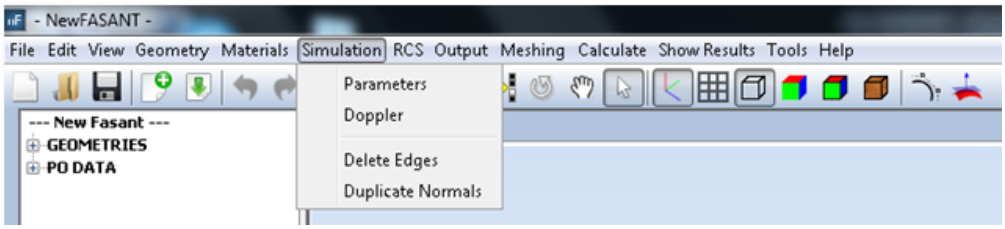


Figure 780: Simulation Parameters Menu

Step 6

Select 1 bounce (simple reflection) and a frequency of 0.3 GHz as shown and left-click the Save button.

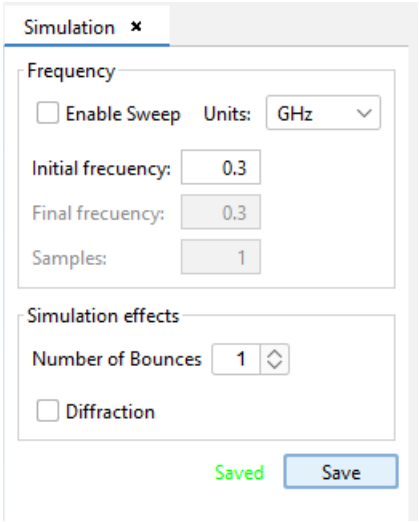


Figure 781: Simulation Parameters panel

Step 7

Select RCS → Parameters.

This command appears in the top left side of the newFASANT window as shown in the next Figure.

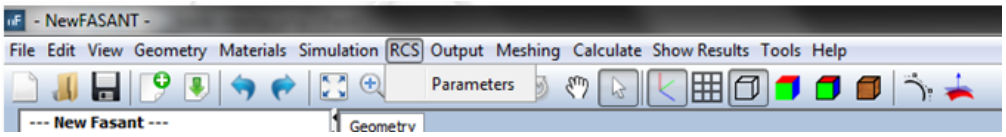


Figure 782: RCS Menu

Step 8

Select Monostatic RCS only, and left-click on the Save button.

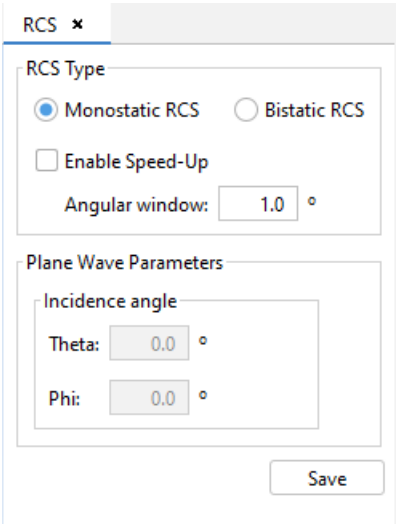


Figure 783: RCS Parameters panel

Step 9

Select Output and left-click on Observation Directions.

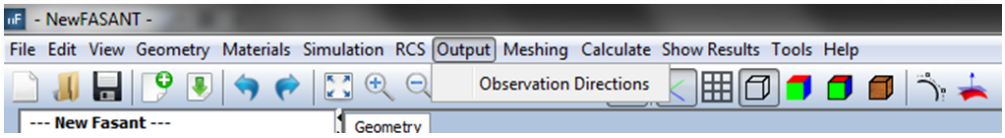


Figure 784: Observation Directions Menu

Step 10

Introduce the far-field observations (cuts, points and sweep angles), as shown in the next Figure and left-click on the Save button.

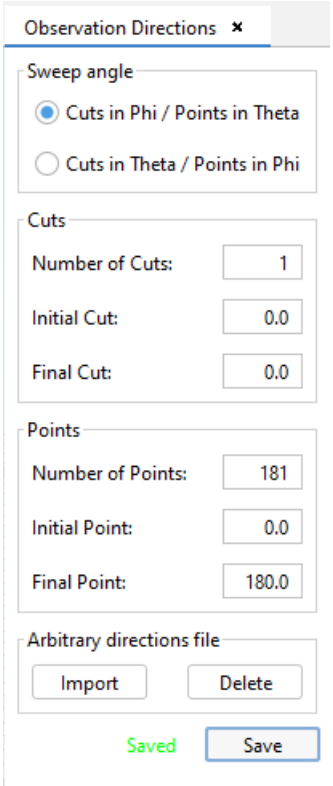


Figure 785: Observation Directions panel

Step 11

Before running the case, select Meshing → Create Mesh.

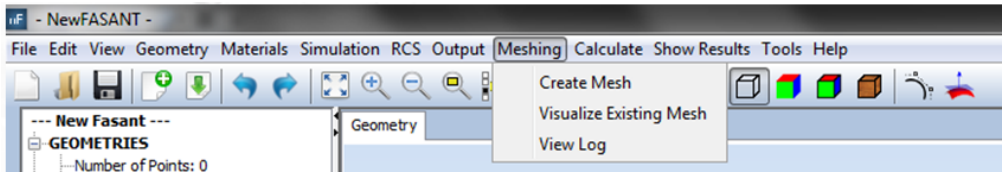


Figure 786: Meshing Parameters Menu

Step 12

Select 1 processor and define the curvature mesh option with a distance error of 10 and surface error of 0.08 and left-click on Mesh.

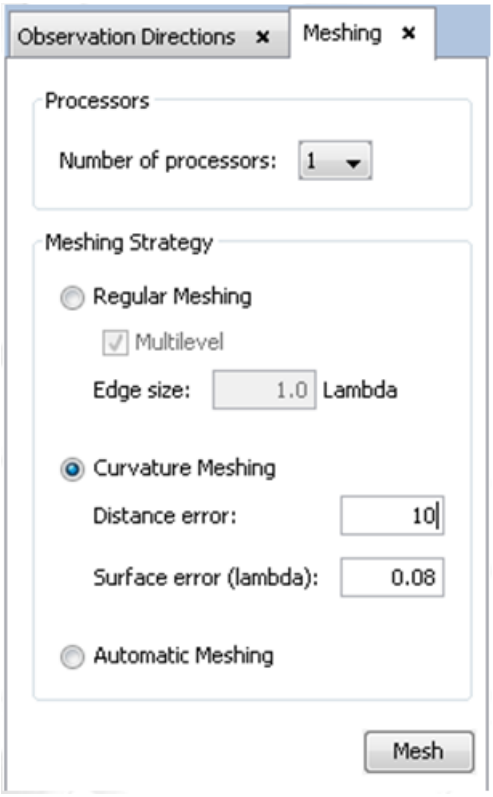


Figure 787: Meshing Parameters panel

Left-clicking on Mesh enables the meshing engine as shown in the next Figure:

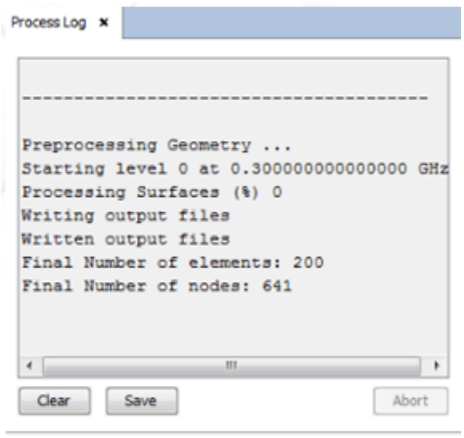


Figure 788: Mesh process

In order to visualize the mesh, then Meshing → Visualize Existing Meshing and select the `.msh` file.

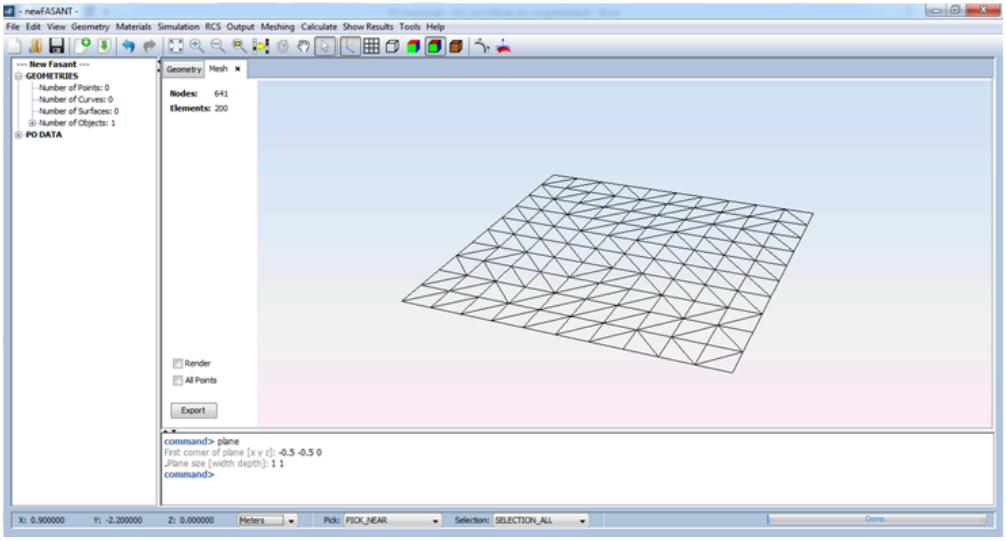


Figure 789: Mesh visualization

Step 13

Select Calculate → Execute and then indicate the a number of processors available to simulate this case.

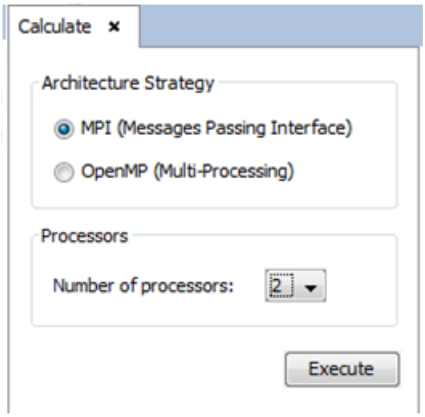


Figure 790: Execute panel

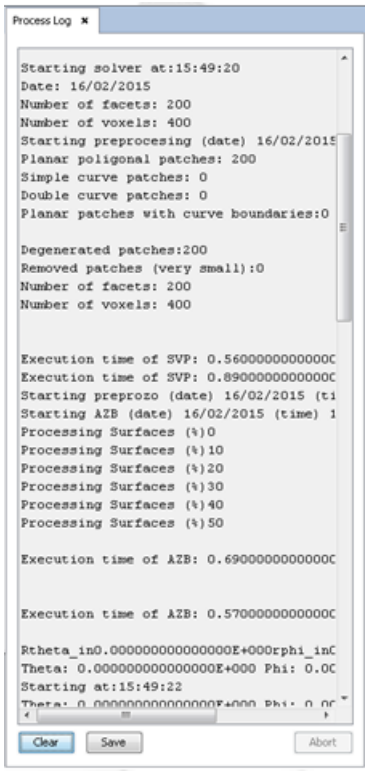


Figure 791: Execute information

Step 14

When the simulation finishes, we can visualize the simulation results. Click on Show Results → Far Field → View Cuts, which allows the user to show the RCS graphic (in the next figure).

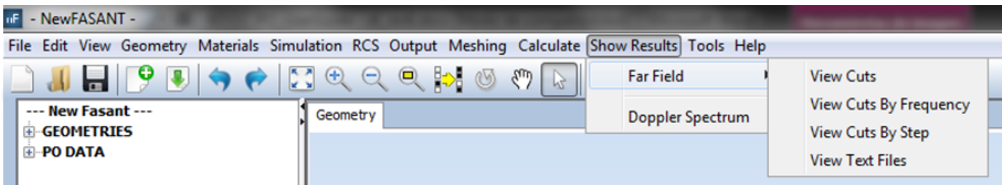


Figure 792: Far Field View Cuts Menu

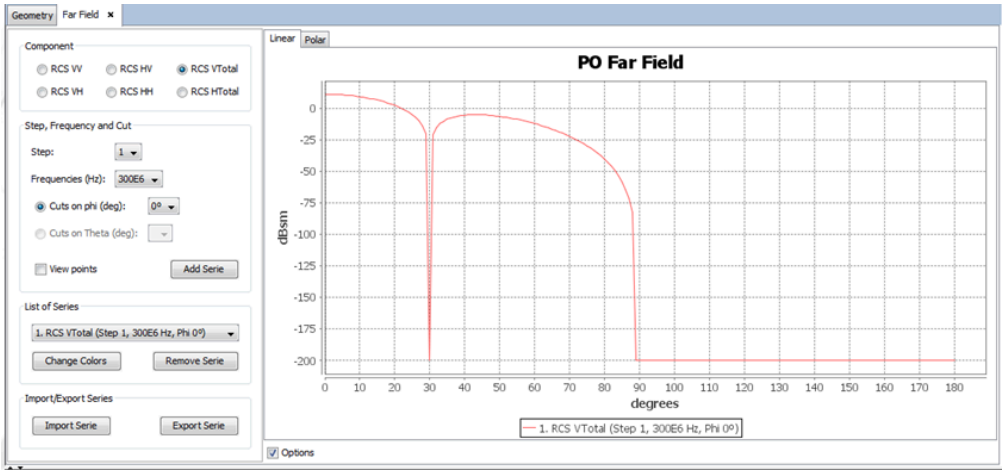


Figure 793: RCS Graphic

Step 15

Click on Show Results → Far Field → View Text Files. Then select the Steps and the order and press OK to show the RCS data file.

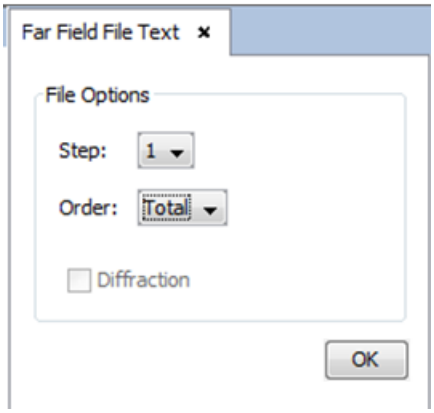


Figure 794: Far Field View Text File panel

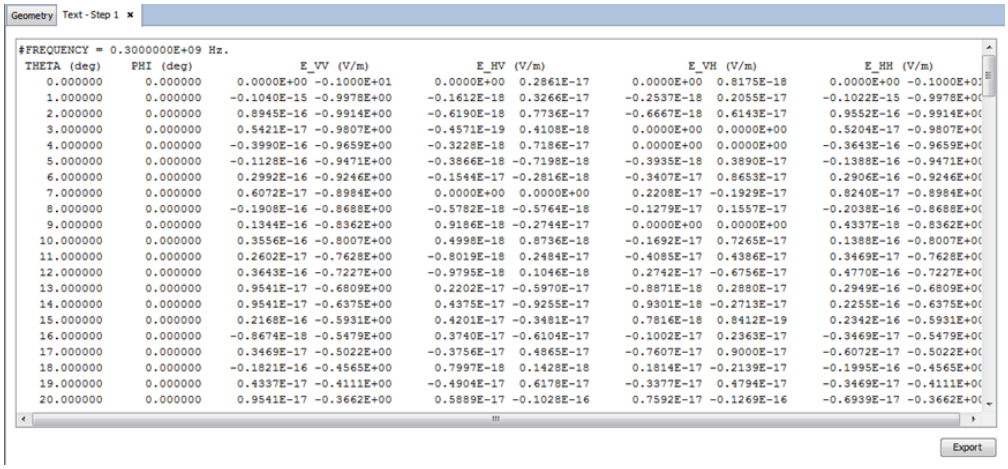


Figure 795: Result Text File visualization

This example has been executed using a plane whose normal vector is along the Z-axis. The user can duplicate this vector and simulate the problem again using *duplicate normal vectors*. After the simulation, the user can visualize the results to see the differences between one normal vector and two duplicated normal vectors. To view the normal vectors of this geometry, click the icon, as shown in the next Figure.

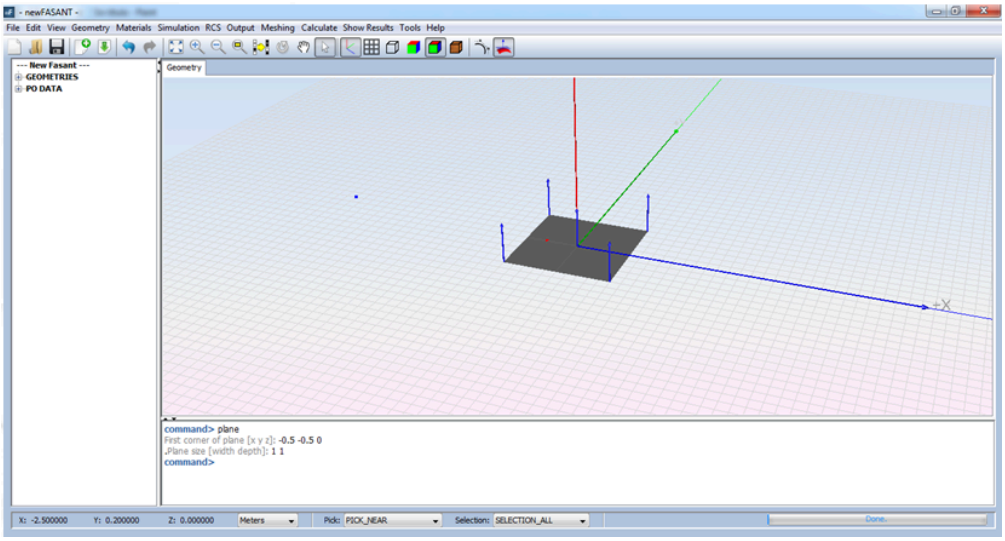


Figure 796: Appearance Normals visualization

Step 16

To duplicate the normal vectors click on Simulation → Duplicate Normals.

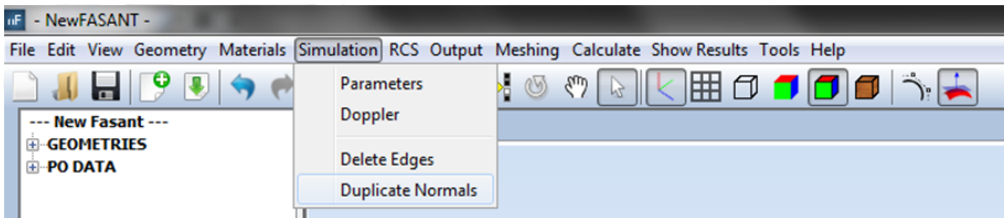


Figure 797: Duplicate Normals Menu

Then select the object to duplicate the normal vector and left-click the Add button in the right panel that has appeared.

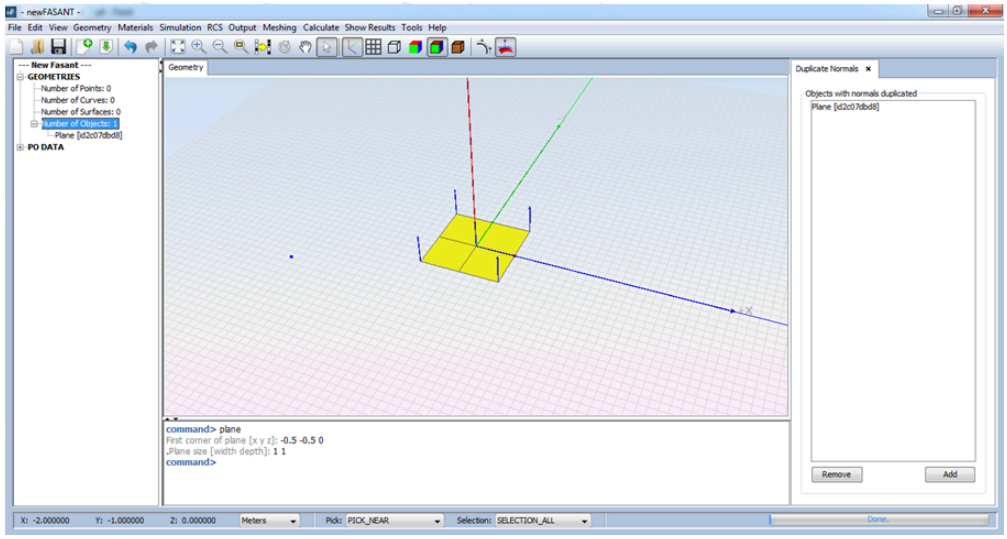


Figure 798: Duplicated Normals visualization

Step 17

Re-mesh the geometry and start the simulation again, like in step 12 and 13. The process is simply the same.

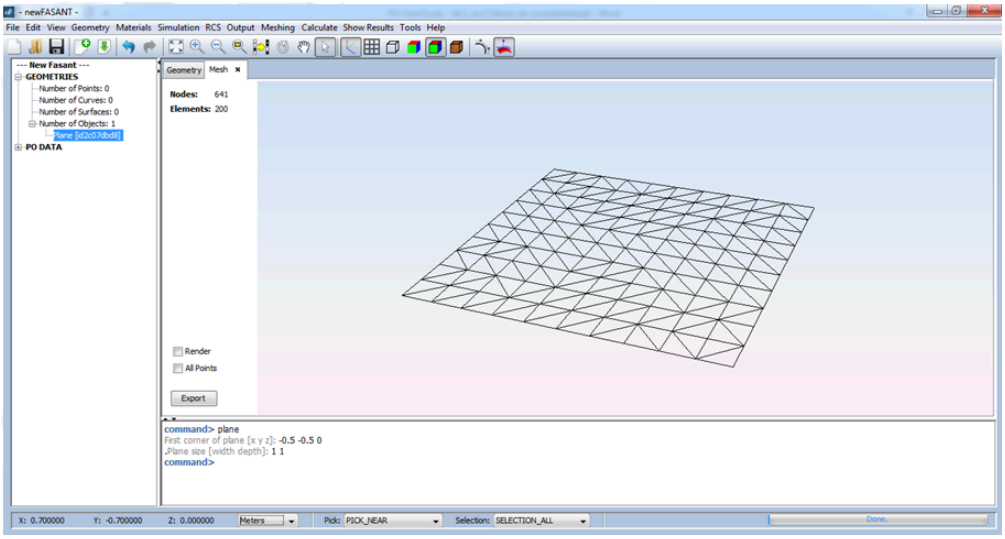


Figure 799: Mesh window

Step 18

Click on Show Results → Far Field → View Cuts, to show the RCS graphic like in step 14. In this case, we can notice that the result is different.

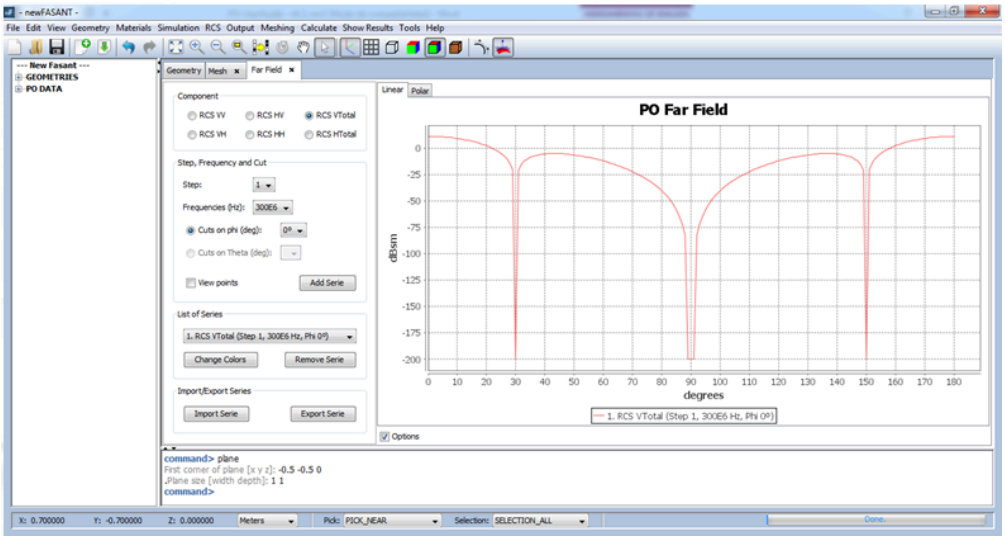


Figure 800: RCS Graphic

8.14.2 Example 2

In this example, the bistatic RCS of a flare is calculated using a frequency sweep from 0.4 GHz to 0.6 GHz (400 MHz to 600 MHz).

Step 1

Create a new project using Step 1 to Step 5 discussed in Example 1.

Step 2

Click on Geometry → Solid → Flare.

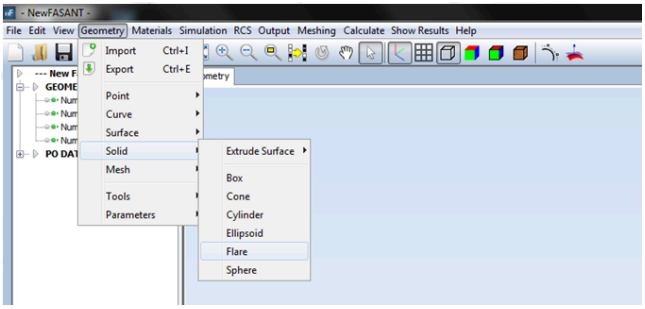


Figure 801: Create flare command

Step 3

Enter the following parameters to define the flare in the command line as shown in the next Figure:

· First corner of base [x y z] -0.5 -1 0

· Base size [width depth] 1 2

· Height 2

· First corner of top [x y] -0.25 -0.25

· Top size [width depth] 0.5 0.5

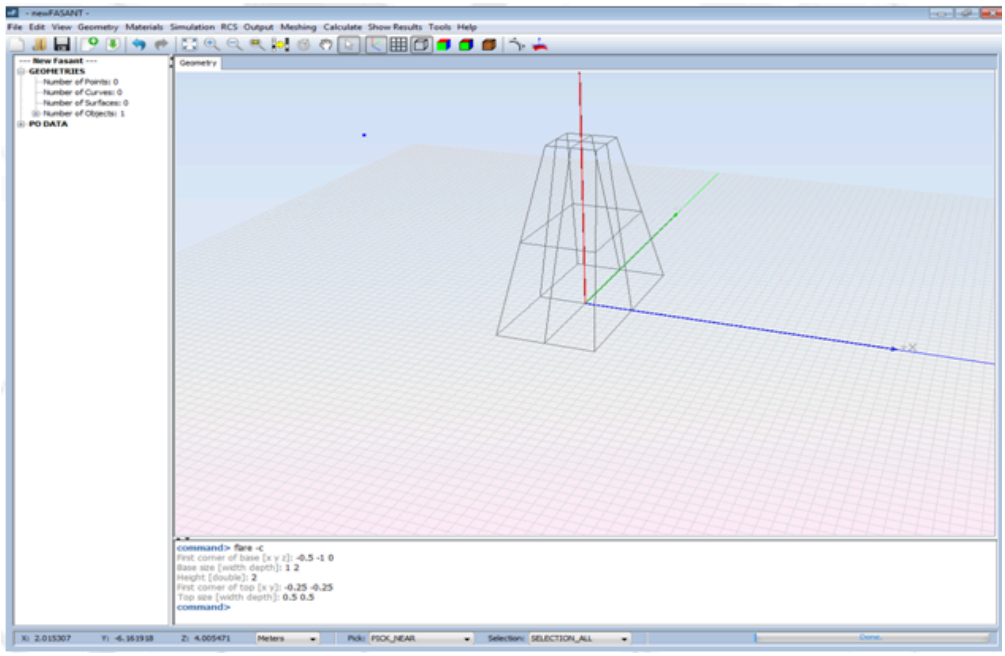


Figure 802: Flare Parameters box

Step 4

Select Simulation → Parameters. Select 1 bounce (simple reflection), and define the diffraction and a frequency sweep from 0.4E9 Hz to 0.6E9 Hz (400 MHz to 600 MHz) as shown in the next Figure, and left-click on Save button.

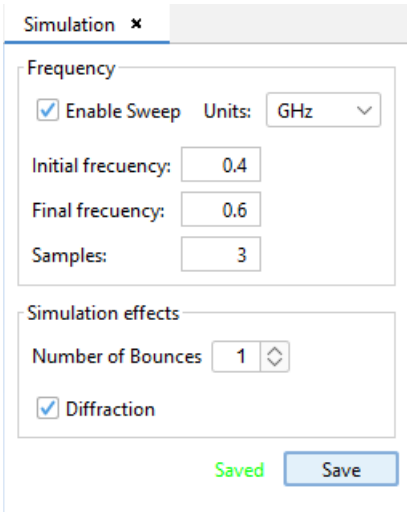


Figure 803: Simulation Parameters panel

Step 5

Select RCS → Parameters. Define the parameters as shown in the following Figure, and left-click the Save button.

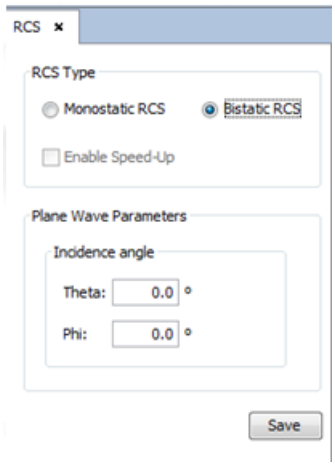


Figure 804: RCS Parameters panel

Step 6

Select Output and left-click on Observation directions. Define the parameters as shown in the next Figure, and left-click the Save button.

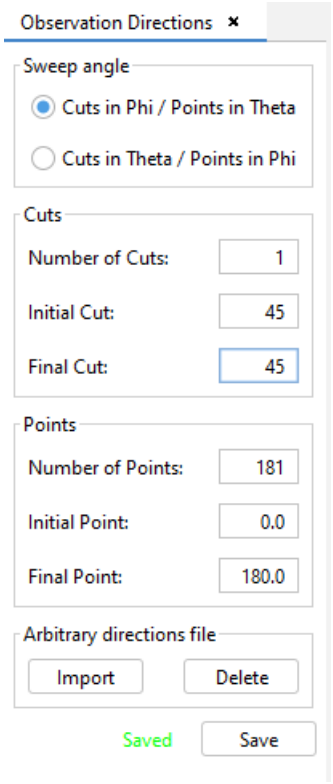


Figure 805: Observation Directions panel

Step 7

Select Meshing → Create Mesh. Define the parameters as shown in the next Figure, and left-click the Mesh button.

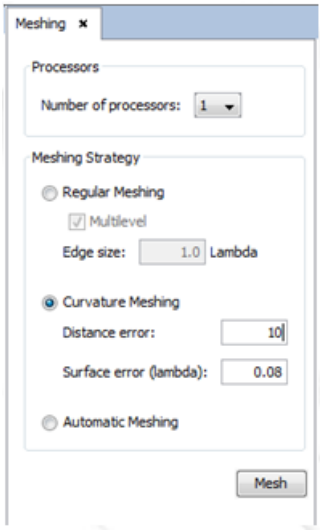


Figure 806: Mesh process

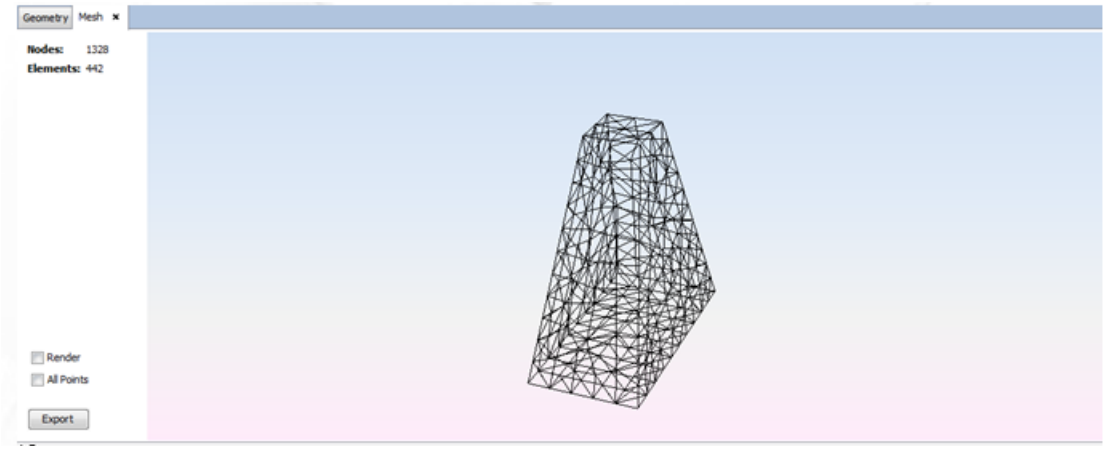


Figure 807: Mesh process

Step 8

Then select Calculate → Execute and enter the number of processors to be used in this simulation.

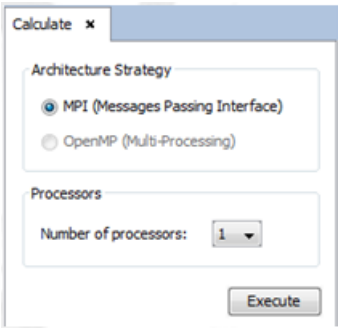


Figure 808: Execute Parameters panel

Step 9

Select Show Results → Far Field → View Cuts.

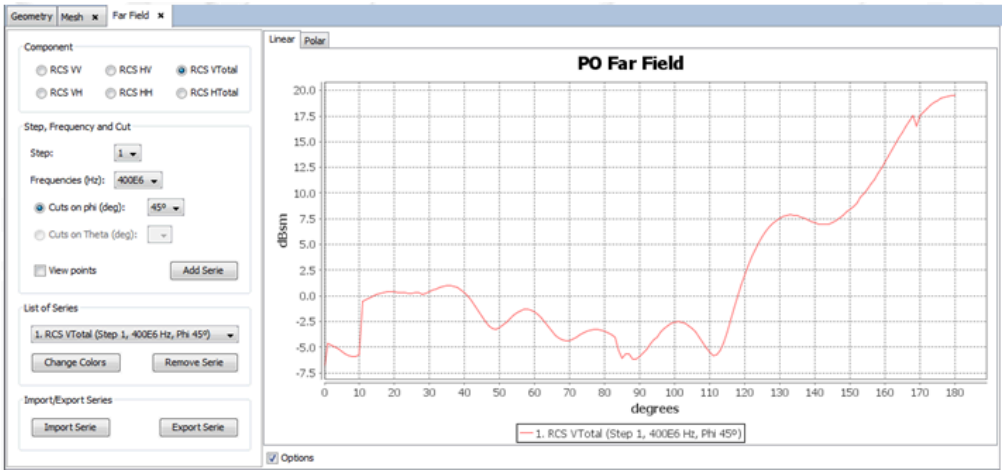


Figure 809: RCS Graphic

Step 10

Select Show Results → Far Field → View Text Files.

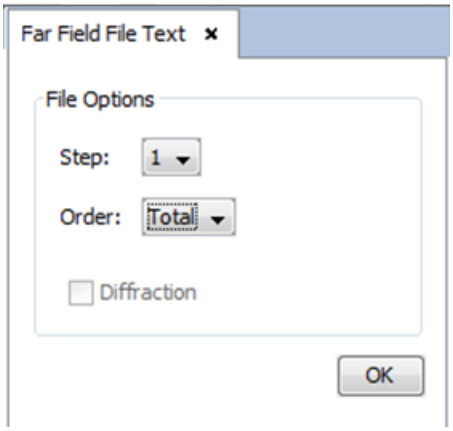


Figure 810: Text Files panel

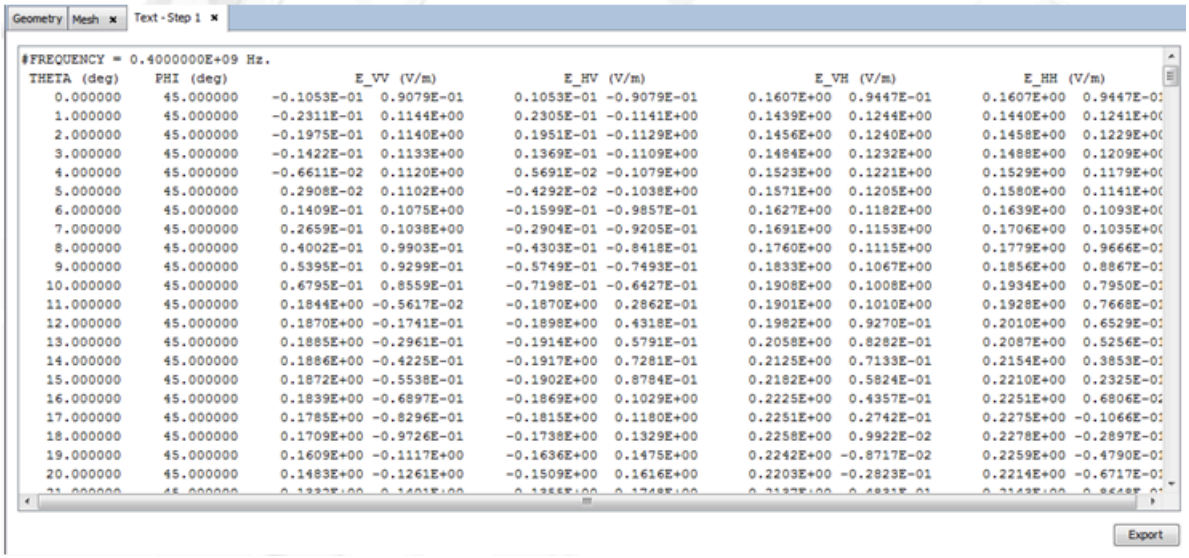


Figure 811: Result Text File visualization

Step 11

Select Materials → Add to add a new material to the simulation in the new panel shown at the right side of the window.

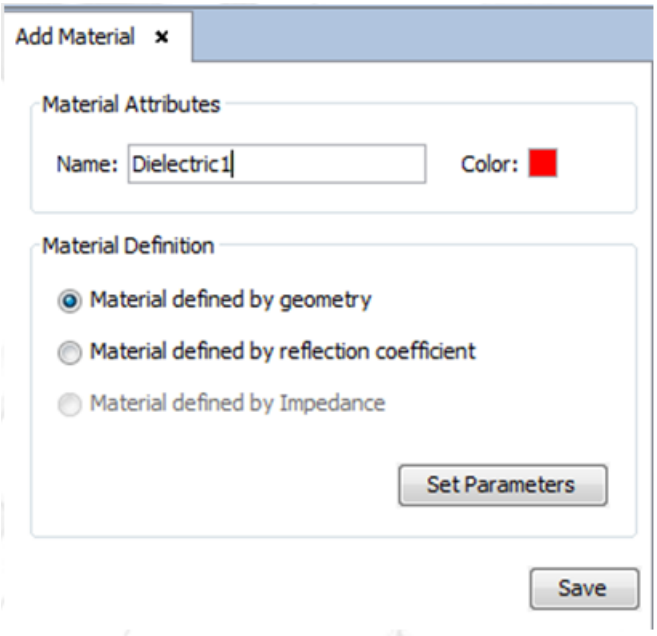


Figure 812: Add Material Parameters box

Then select the "Material defined by Geometry" option and also left-click the "Set Parameters" button to define the material properties as shown in the next Figure. Press Save.

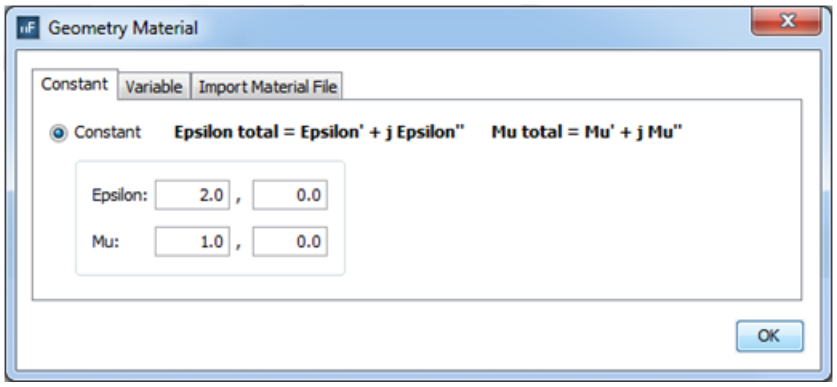


Figure 813: Material defined by geometry Properties box

Step 12

Select Material → Assign to assign the new material properties to the geometry. Select the object and left-click the "Assign" button as shown in the next Figure.

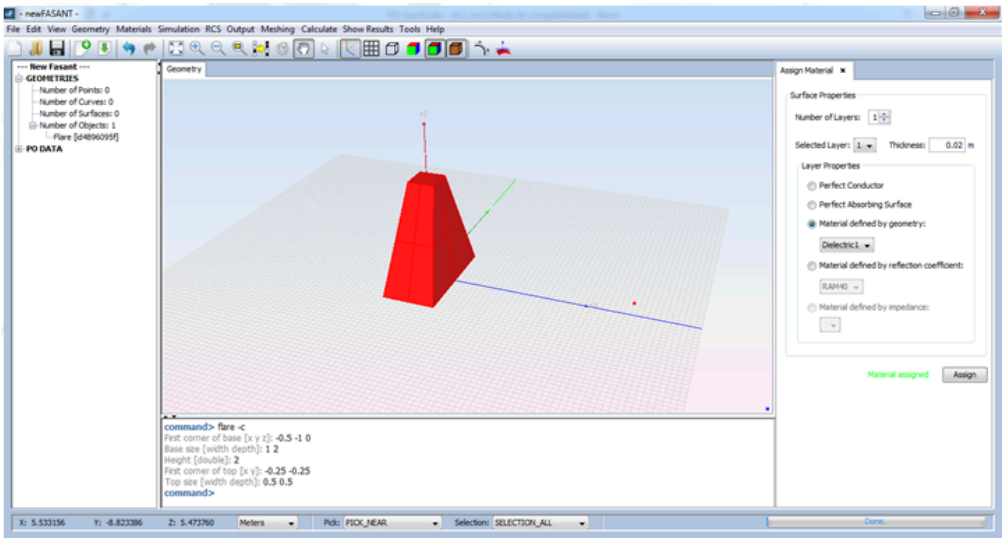


Figure 814: Material Assign object selection

Step13

Re-mesh the geometry and run the simulation again. Then open the new RCS results shown in the next Figure.

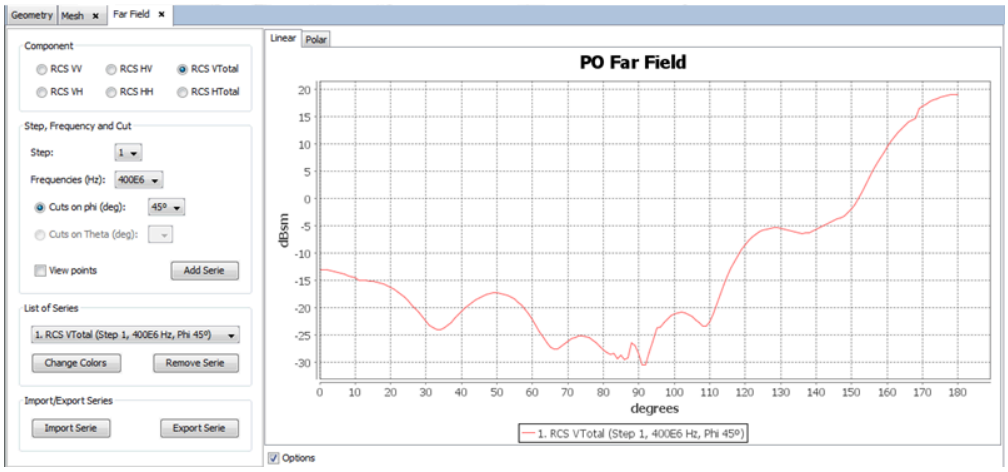


Figure 815: RCS Graphic

8.14.3 Example 3

In this example, the Monostatic RCS of two cubes is calculated, where one cube is dynamic.

Step 1

Create a new project following steps 1 to 5 in Example1 above.

Step 2

Click on Geometry → Solid → Box.

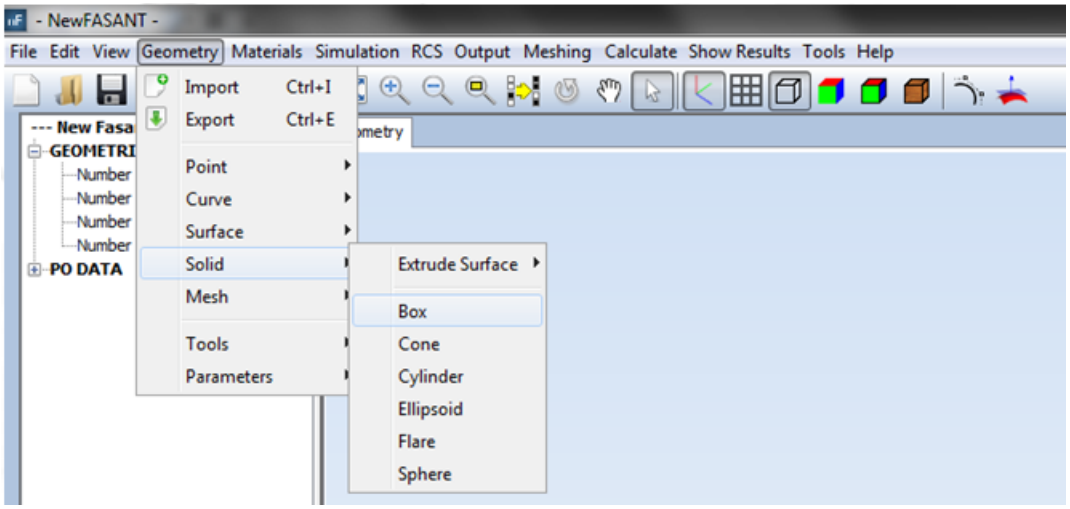


Figure 816: Create box command

Create the first box by entering the following parameters from the command line as shown in the next Figure.

· First corner of base [x y z] 0.5 -0.5 0

· Base size [width depth] 1 1

· Height 1

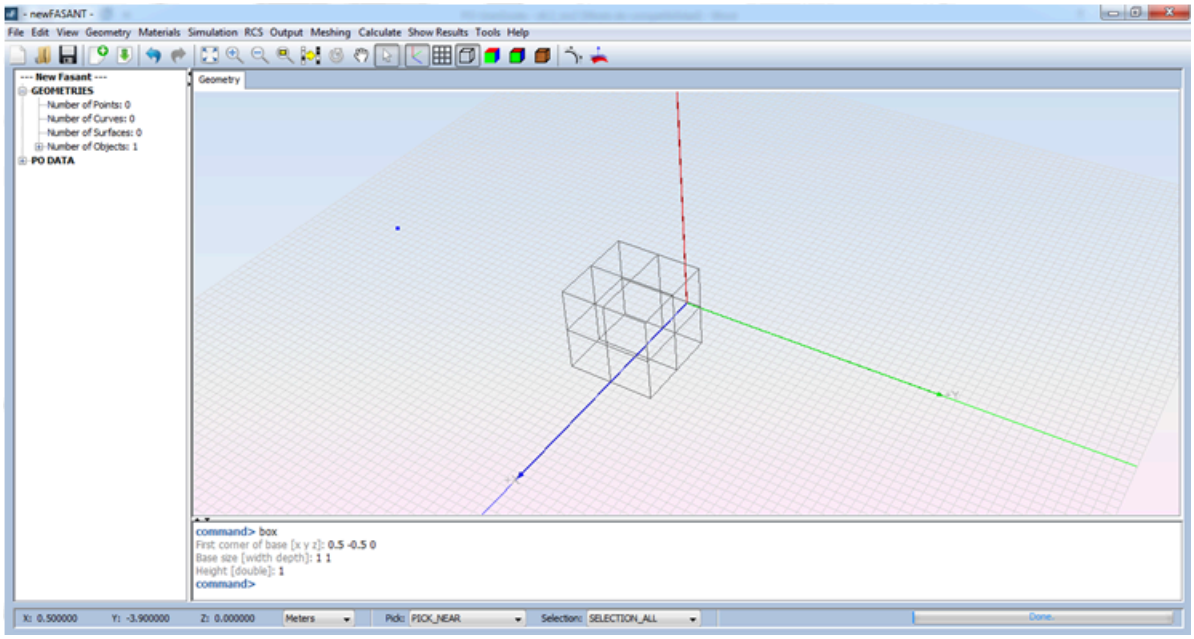


Figure 817: Box Parameters box

Step 3

Now, let's add the dynamic cube to the model. We want this cube to move in the direction of the positive Z-axis at 0.5 meters each step in the simulation. We will use three steps for this simulation.

First, we must define a parameter for the geometry that represents the Z coordinate of the base of the cube at each step of the simulation. To do this, select Geometry → Parameters → Define Parameters and the following window appears:

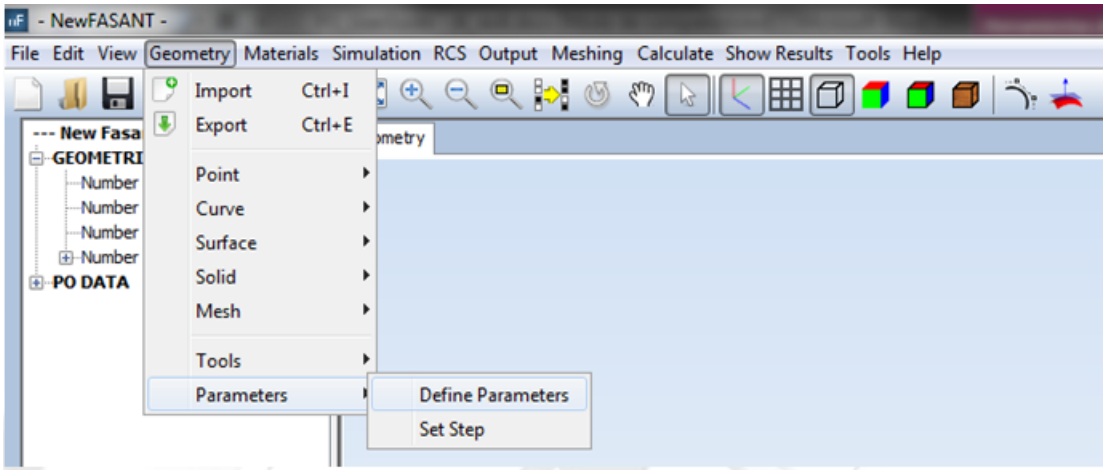


Figure 818: Define parameters

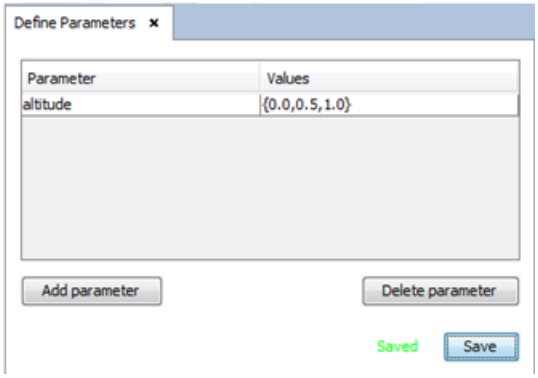


Figure 819: Define parameters panel

Press the Add parameter button, after which a row appears in the table. In the Parameter column, change the name of the created parameter to "height", for example. Then change the values to "{0.0, 0.5, 1.0}" and left-click the Save button to save the changes.

Now, create a new box as the other. It is possible to change the parameters as desired, but the z coordinate of the first corner of the base must be now the parameter "altitude".

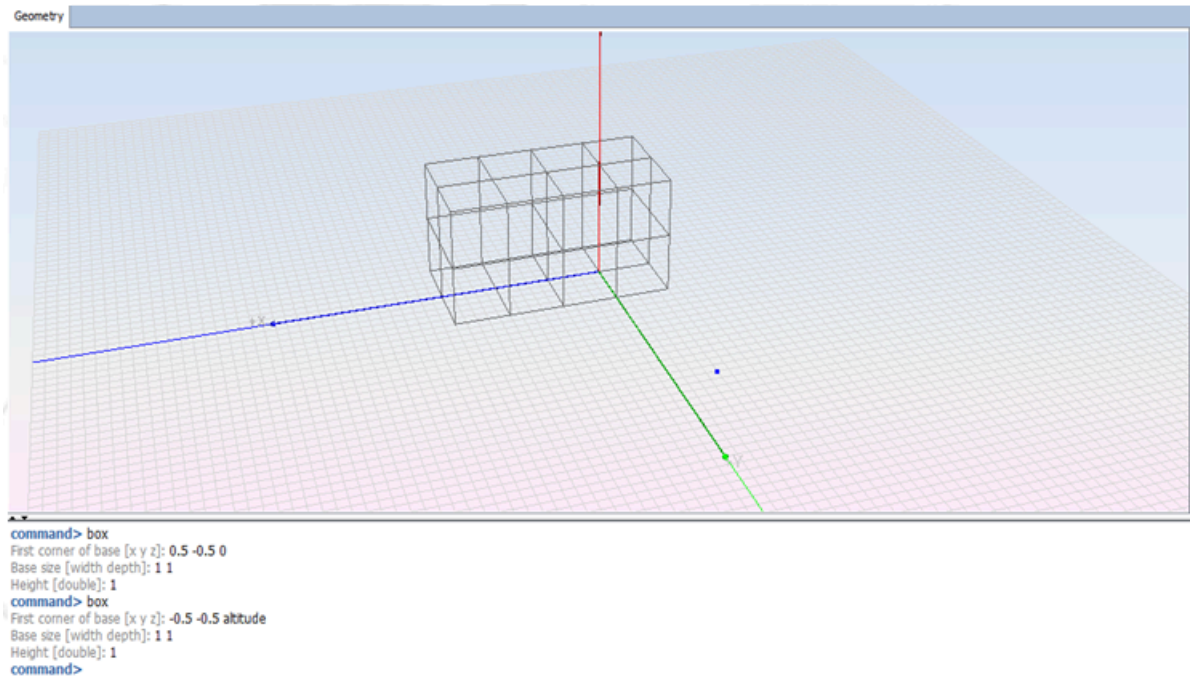


Figure 820: Creating the dynamic cube

Optional: To see a visualization of each step for the second cube, select the Geometry → Parameters → Set Step. Enable "Linear (By step)", set the desired time delay between steps, and left-click Run to see the animation on the Geometry View.

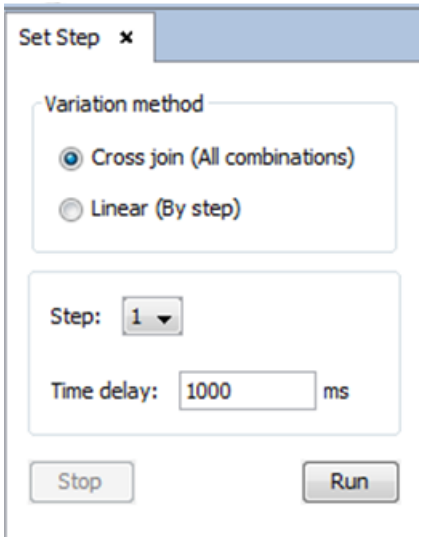


Figure 821: Execute Parameters box

It is possible to see the two remaining steps for the simulation in the next Figures.

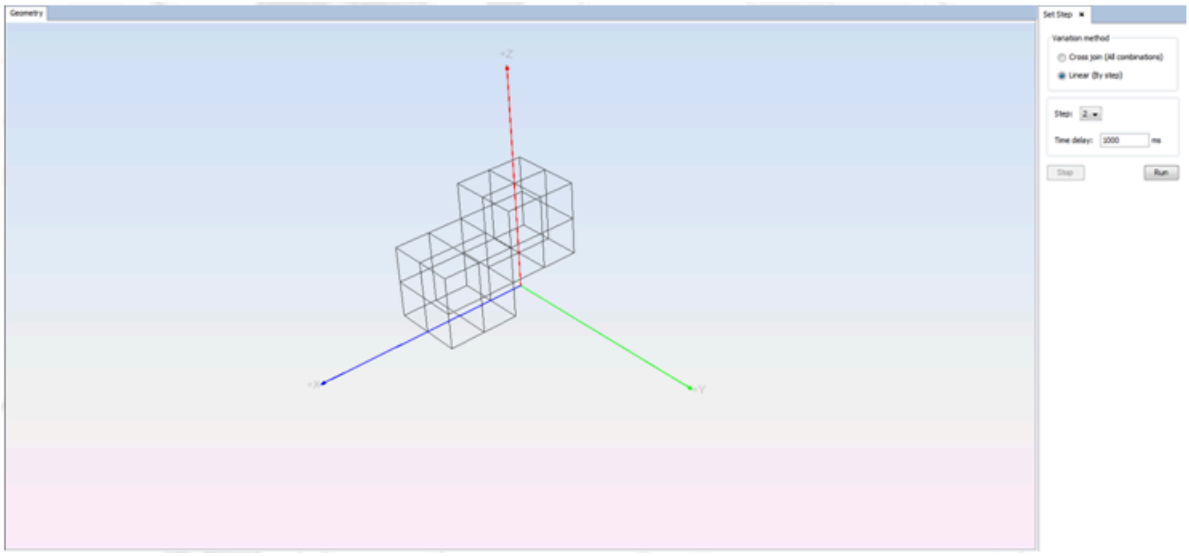


Figure 822: View of the dynamic cube (Step 2)

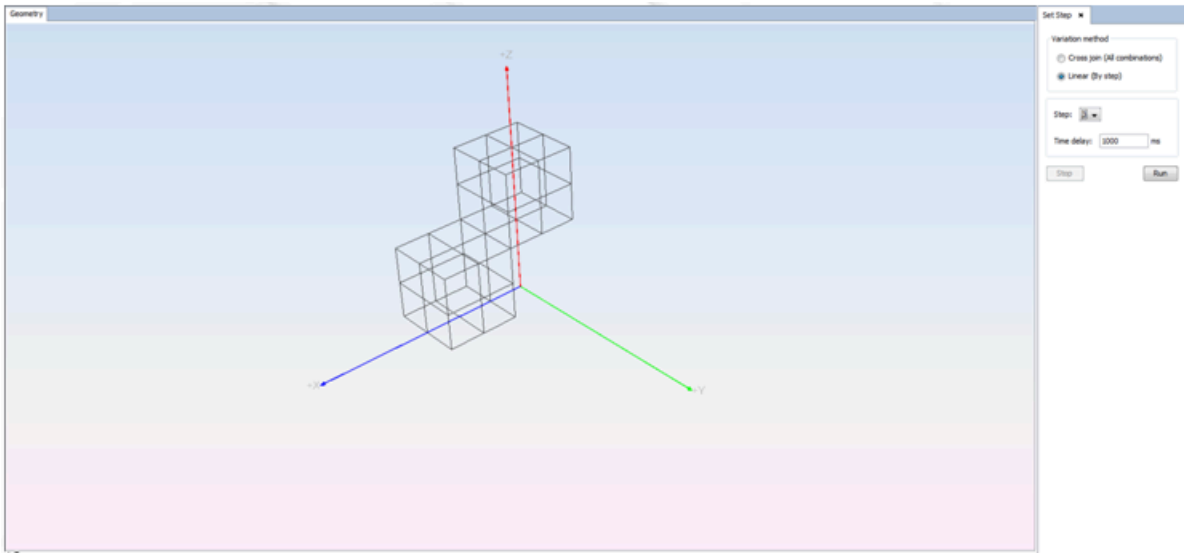


Figure 823: View of the dynamic cube (Step 3)

Step 4: Click on Simulation → Parameters. Enter the simulation parameters as shown in the next Figure.

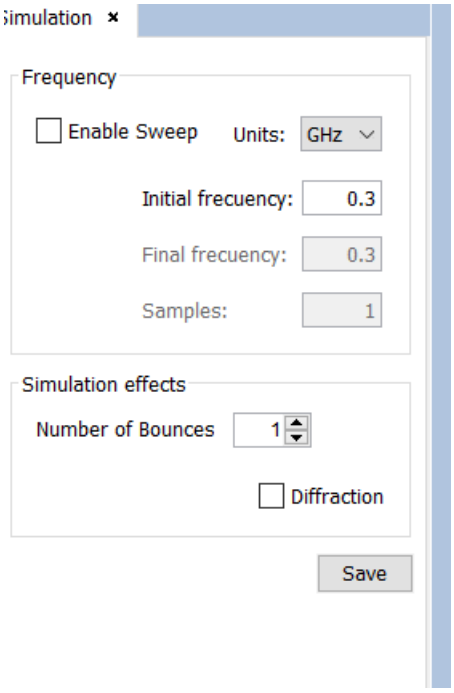


Figure 824: Simulation parameters

Step 5

Select RCS → Parameters and enter the parameters shown in the next Figure.

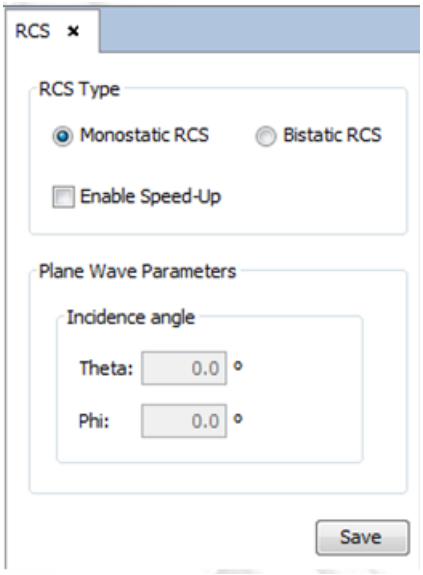


Figure 825: RCS parameters

Step 6

Select Output → Observation Directions and enter the parameters shown in the next Figure.

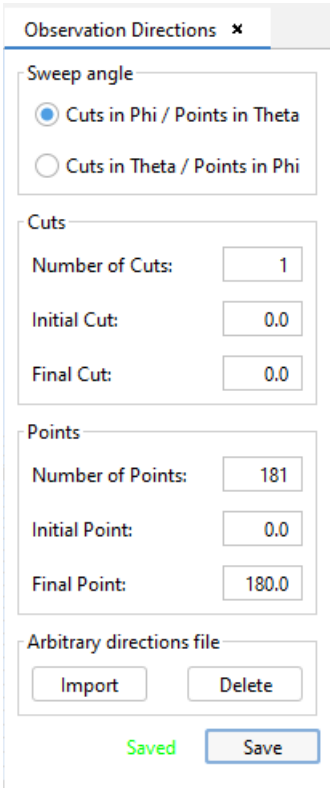


Figure 826: Observation Directions panel

Step 7

Select Meshing → Create Mesh. Select the appropriate number of processors available to run the meshing process as shown in the next Figure and left-click Mesh.

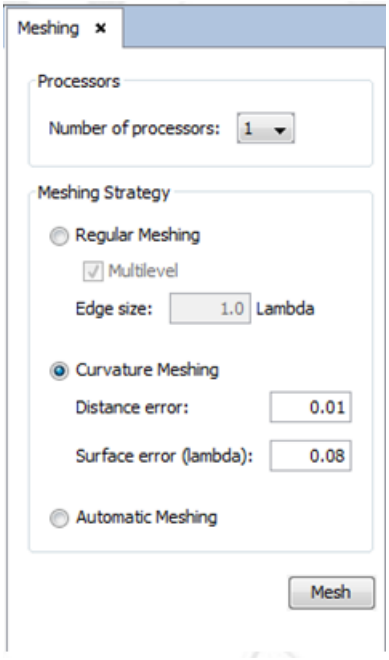


Figure 827: Meshing configuration

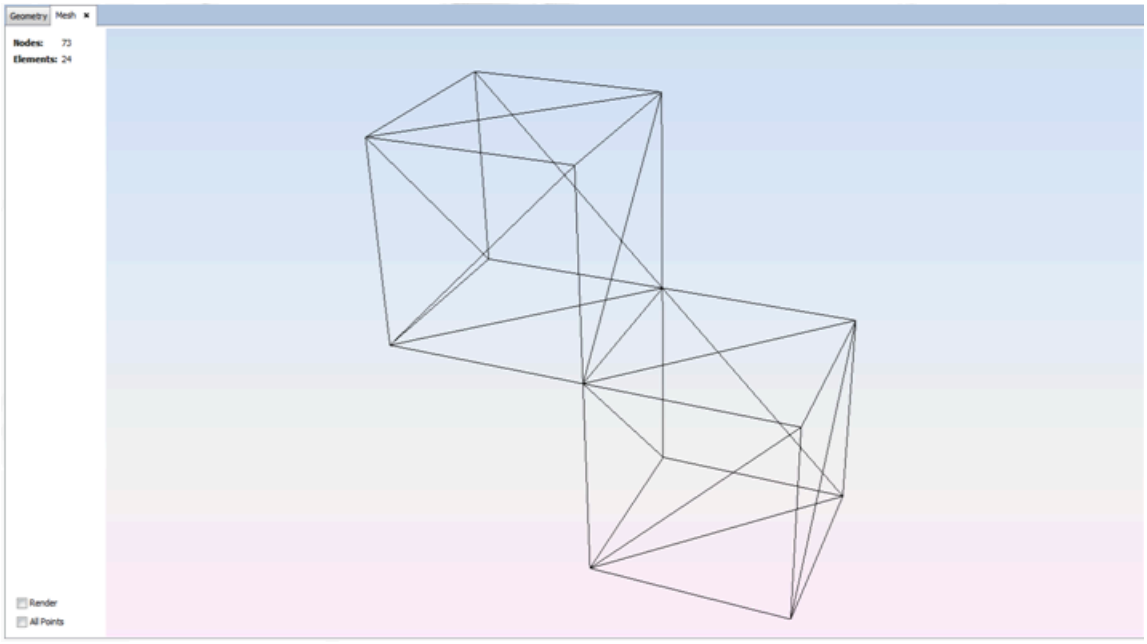


Figure 828: Resulting Mesh

Step 8

Select Calculate → Execute and enter the number of available processors for this simulation.

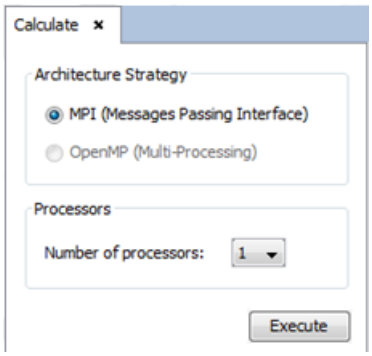


Figure 829: Calculate options

Step 9

Click on Show Results → Far Field → View Cuts, to show the RCS graph.

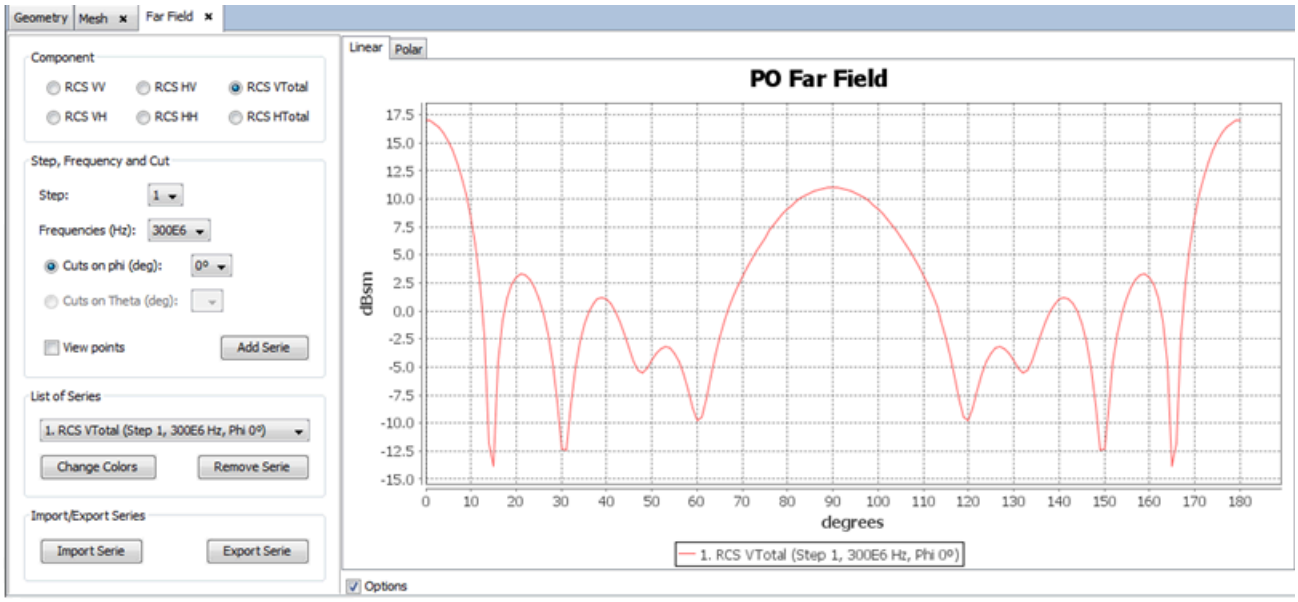


Figure 830: RCS graphic of the first step

The plot corresponds to the RCS of the first step (when the two boxes share the same Z coordinate). To plot the graphics for the remaining two steps, select “2” in the Step combo-box and left-click the Add Series button. Repeat this procedure to add Step 3 to the RCS graphic.

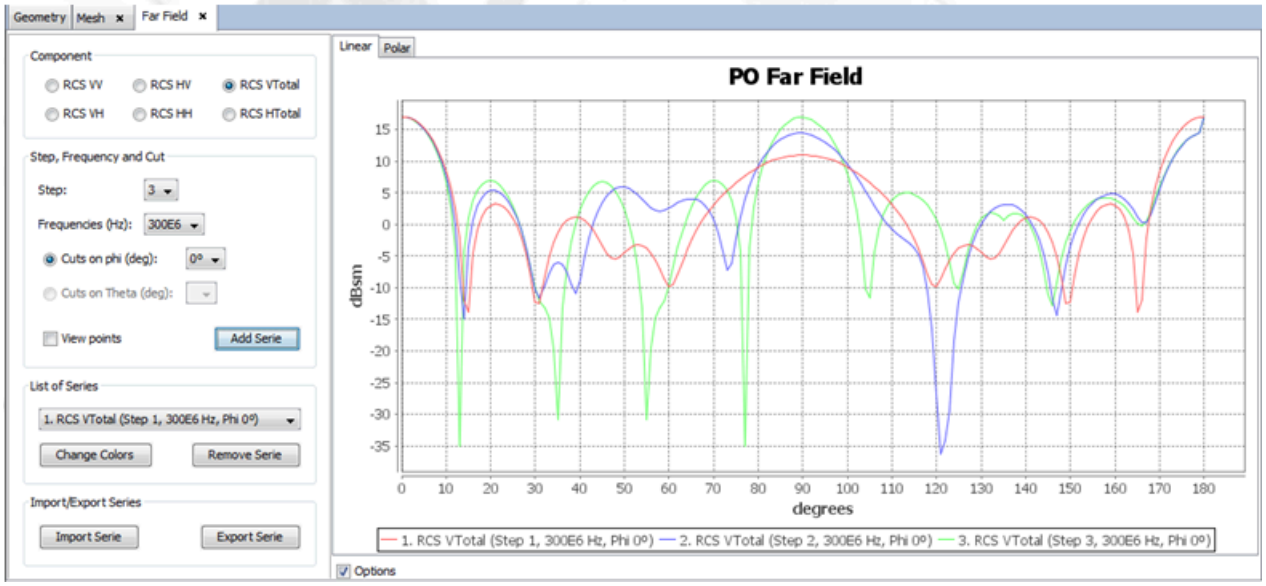


Figure 831: RCS graphic showing each step

Step 10

Click on Show Results → Far Field → View Text Files, to show the RCS data file. Select the step to obtain as a data file. Press OK to continue.

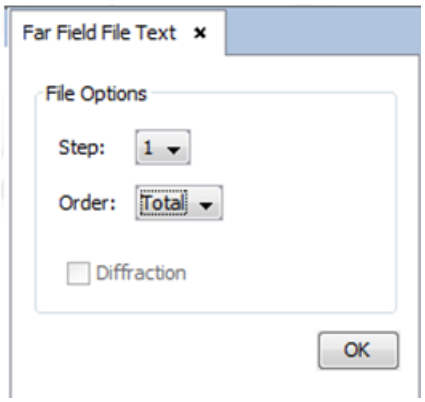
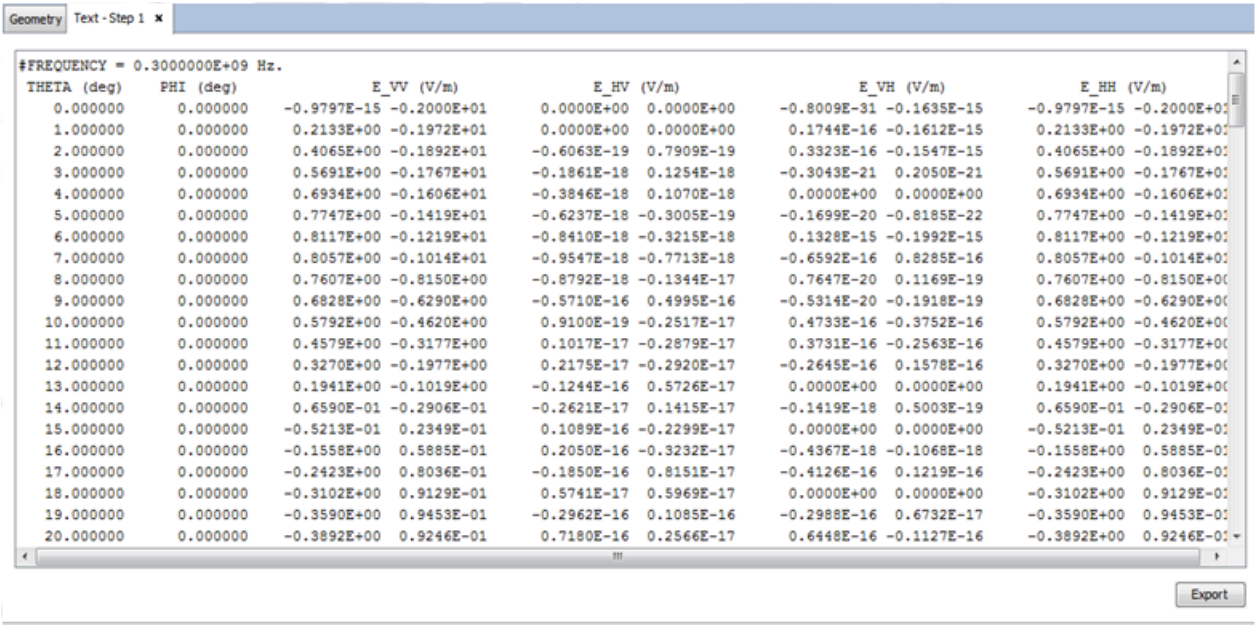


Figure 832: View Text Files options


Geometry | Text - Step 1 ×

#FREQUENCY = 0.3000000E+09 Hz.

THETA (deg)	PHI (deg)	E_VV (V/m)		E_HV (V/m)		E_VH (V/m)		E_HH (V/m)	
0.000000	0.000000	-0.9797E-15	-0.2000E+01	0.0000E+00	0.0000E+00	-0.8009E-31	-0.1635E-15	-0.9797E-15	-0.2000E+01
1.000000	0.000000	0.2133E+00	-0.1972E+01	0.0000E+00	0.0000E+00	0.1744E-16	-0.1612E-15	0.2133E+00	-0.1972E+01
2.000000	0.000000	0.4065E+00	-0.1892E+01	-0.6063E-19	0.7909E-19	0.3323E-16	-0.1547E-15	0.4065E+00	-0.1892E+01
3.000000	0.000000	0.5691E+00	-0.1767E+01	-0.1861E-18	0.1254E-18	-0.3043E-21	0.2050E-21	0.5691E+00	-0.1767E+01
4.000000	0.000000	0.6934E+00	-0.1606E+01	-0.3846E-18	0.1070E-18	0.0000E+00	0.0000E+00	0.6934E+00	-0.1606E+01
5.000000	0.000000	0.7747E+00	-0.1419E+01	-0.6237E-18	-0.3005E-19	-0.1699E-20	-0.8185E-22	0.7747E+00	-0.1419E+01
6.000000	0.000000	0.8117E+00	-0.1219E+01	-0.8410E-18	-0.3215E-18	0.1328E-15	-0.1992E-15	0.8117E+00	-0.1219E+01
7.000000	0.000000	0.8057E+00	-0.1014E+01	-0.9547E-18	-0.7713E-18	-0.6592E-16	0.8285E-16	0.8057E+00	-0.1014E+01
8.000000	0.000000	0.7607E+00	-0.8150E+00	-0.8792E-18	-0.1344E-17	0.7647E-20	0.1169E-19	0.7607E+00	-0.8150E+00
9.000000	0.000000	0.6828E+00	-0.6290E+00	-0.5710E-16	0.4995E-16	-0.5314E-20	-0.1918E-19	0.6828E+00	-0.6290E+00
10.000000	0.000000	0.5792E+00	-0.4620E+00	0.9100E-19	-0.2517E-17	0.4733E-16	-0.3752E-16	0.5792E+00	-0.4620E+00
11.000000	0.000000	0.4579E+00	-0.3177E+00	0.1017E-17	-0.2879E-17	0.3731E-16	-0.2563E-16	0.4579E+00	-0.3177E+00
12.000000	0.000000	0.3270E+00	-0.1977E+00	0.2175E-17	-0.2920E-17	-0.2645E-16	0.1578E-16	0.3270E+00	-0.1977E+00
13.000000	0.000000	0.1941E+00	-0.1019E+00	-0.1244E-16	0.5726E-17	0.0000E+00	0.0000E+00	0.1941E+00	-0.1019E+00
14.000000	0.000000	0.6590E-01	-0.2906E-01	-0.2621E-17	0.1415E-17	-0.1419E-18	0.5003E-19	0.6590E-01	-0.2906E-01
15.000000	0.000000	-0.5213E-01	0.2349E-01	0.1089E-16	-0.2299E-17	0.0000E+00	0.0000E+00	-0.5213E-01	0.2349E-01
16.000000	0.000000	-0.1558E+00	0.5885E-01	0.2050E-16	-0.3232E-17	-0.4367E-18	-0.1068E-18	-0.1558E+00	0.5885E-01
17.000000	0.000000	-0.2423E+00	0.8036E-01	-0.1850E-16	0.8151E-17	-0.4126E-16	0.1219E-16	-0.2423E+00	0.8036E-01
18.000000	0.000000	-0.3102E+00	0.9129E-01	0.5741E-17	0.5969E-17	0.0000E+00	0.0000E+00	-0.3102E+00	0.9129E-01
19.000000	0.000000	-0.3590E+00	0.9453E-01	-0.2962E-16	0.1085E-16	-0.2988E-16	0.6732E-17	-0.3590E+00	0.9453E-01
20.000000	0.000000	-0.3892E+00	0.9246E-01	0.7180E-16	0.2566E-17	0.6448E-16	-0.1127E-16	-0.3892E+00	0.9246E-01

Export

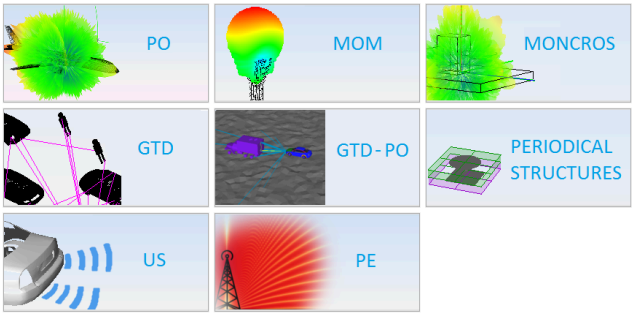
Figure 833: Far-Field text file

8.14.4 Example 4: Monostatic RCS of a Ship

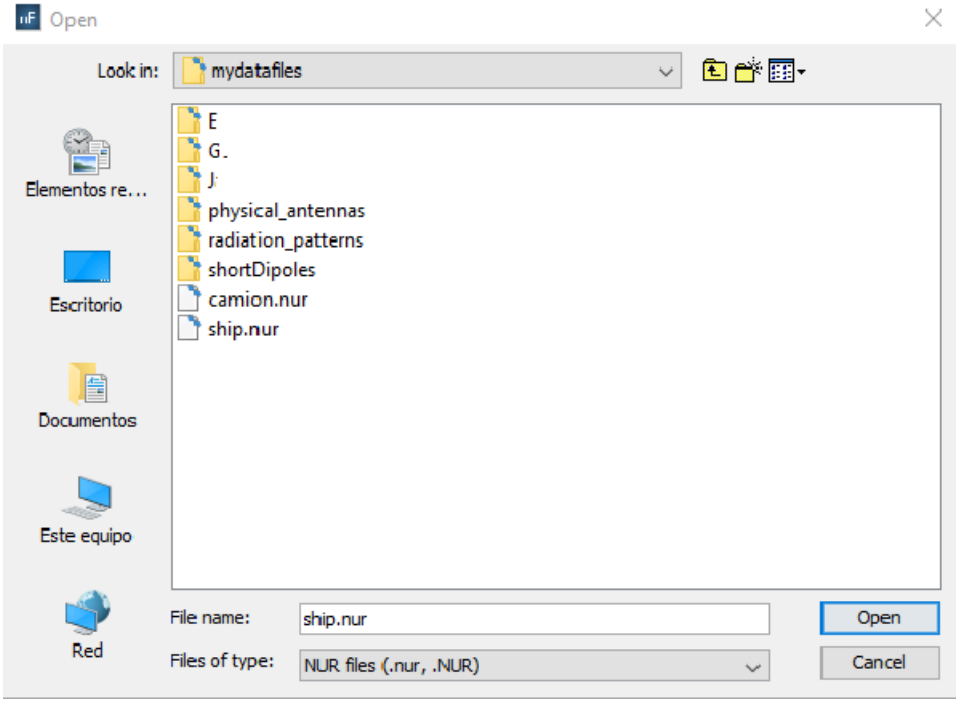
This case explains how to calculate the monostatic RCS of an imported geometry.

Step 1: Start newFASANT and click on **File** → **New**

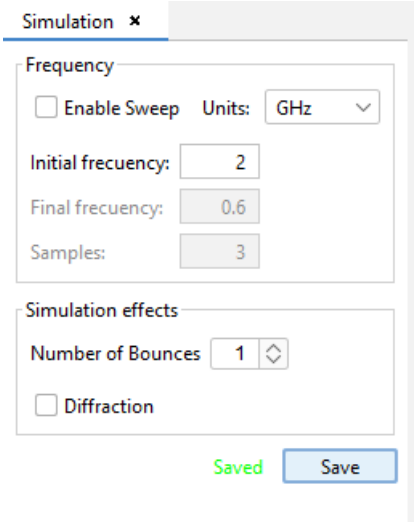
Step 2: Select PO.



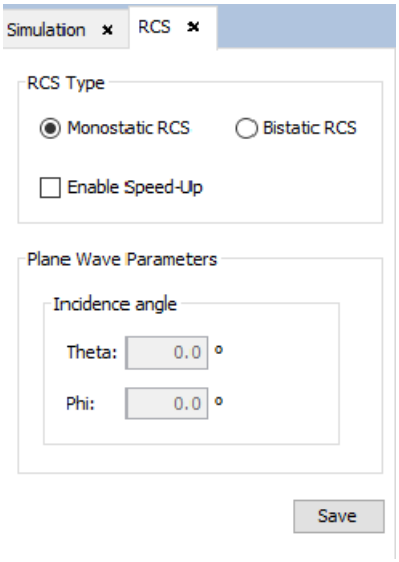
Step 3: Select the geometry to import.



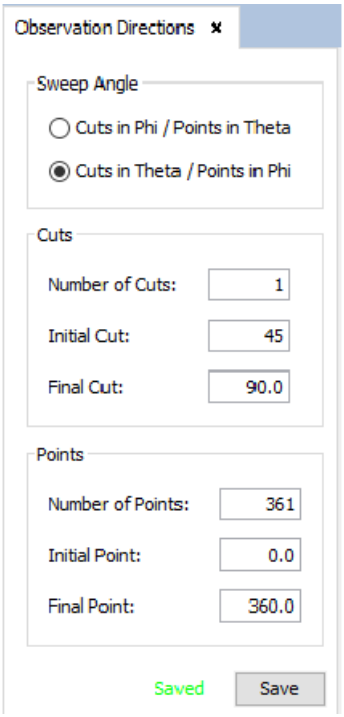
Step 4: Click on Simulation → Parameters to change the project parameters.



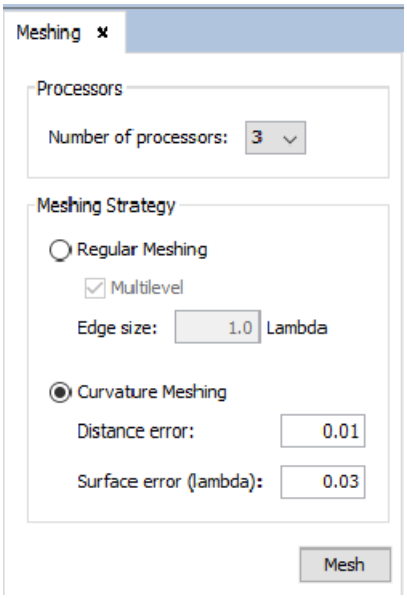
Step 5: Click on RCS → Parameters to set de RCS parameters of the project.



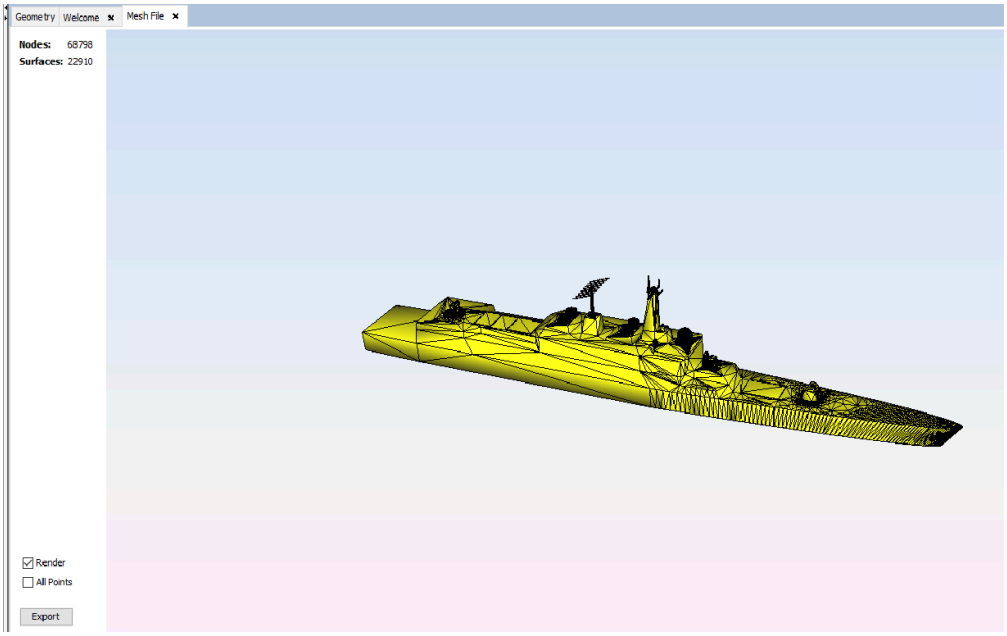
Step 6: Click on Output → Parameters to modify the source parameters for this project.



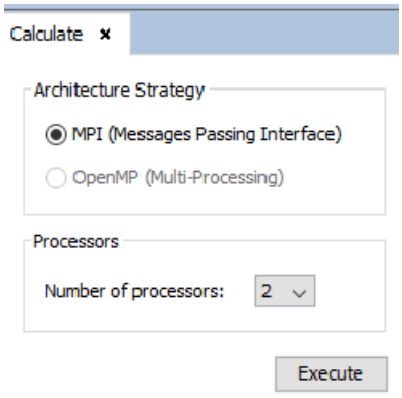
Step 7: Before running this example, create the project mesh by doing click on Meshing → Create Mesh.



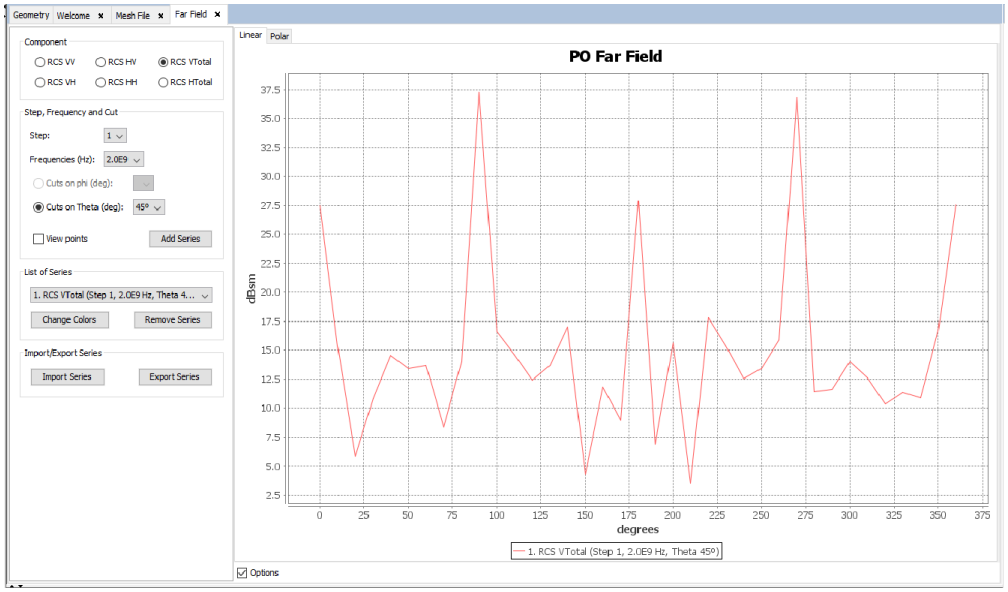
Step 8 Visualize the created mesh. To do this click on Meshing → Visualize mesh. Now you will be able to choose the generated mesh file.



Step 9: Now we are going to simulate the project. Click on Calculate → Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.



Step 10: Now you can see the results of the simulation by doing click on Show Results and selecting the desired results.



GTD-PO

9

GTD-PO combines the GTD and PO high-frequency techniques for the RCS analysis of complex bodies as well as antenna and radio propagation in large urban environments.

This chapter covers the following:

- 9.1 File Menu (p. 840)
- 9.2 Edit Menu (p. 841)
- 9.3 View Menu (p. 842)
- 9.4 Geometry Menu (p. 843)
- 9.5 Materials Menu (p. 844)
- 9.6 Simulation Menu (p. 845)
- 9.7 RCS Menu (p. 851)
- 9.8 Source Menu (p. 852)
- 9.9 Output Menu (Alt +O) (p. 857)
- 9.10 Meshing Menu (p. 861)
- 9.11 Calculate Menu (Alt + C) (p. 867)
- 9.12 Show Results Menu (p. 869)
- 9.13 Tools Menu (p. 893)
- 9.14 Help Menu (p. 894)
- 9.15 Training Examples (p. 895)

9.1 File Menu

For information about the File menu, refer to the GUI Use Guide.

9.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

9.3 View Menu

For information about the View menu, refer to View Menu.

9.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

9.5 Materials Menu

For information about the Materials menu, refer to Materials.

9.6 Simulation Menu

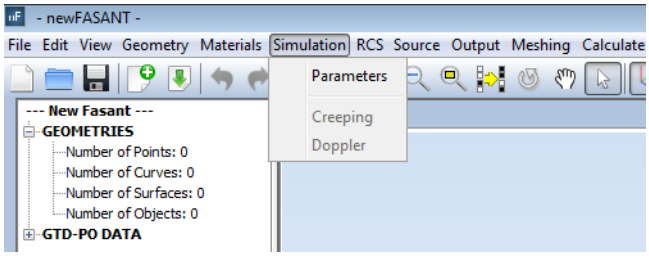


Figure 834: Simulation Menu

9.6.1 Parameters

The parameters are defined by the user on the right side of the window. It is possible to select a single frequency simulation, or a frequency sweep by left-clicking the Enable Sweep box. The user will then define the Initial Frequency, Final Frequency, and the number of frequency points (Samples) for the simulation.

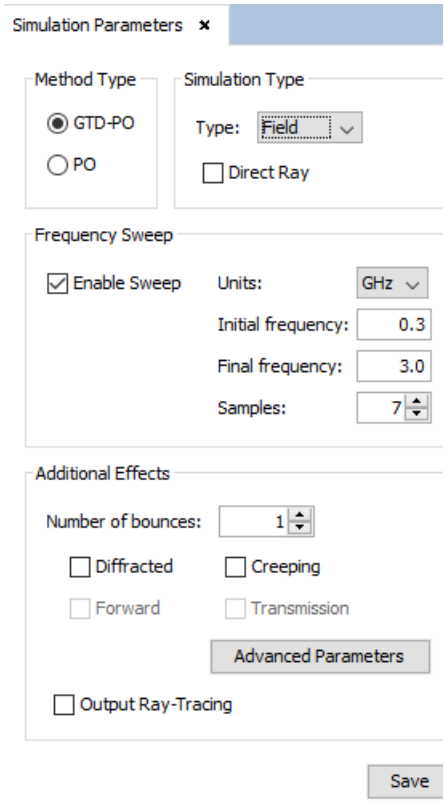


Figure 835: Parameters panel

In this panel, the user can set the main parameters of the simulation:

***Method Type**.*
The user can select the method to be applied (GTD-PO or PO).

***Simulation Type**. The user can select four different simulation types:*

- ***RCS***: to compute the radar cross section.
- ***Field***: to compute only the electric field. When this option is selected it is also possible to select direct ray, which is selected by default.
- ***Coupling***: to compute the coupling. When this option is selected it is also possible to select direct ray in order to take it into account during the analysis.
- ***Doppler***: to compute the Doppler shift due the displacement of some objects of the scene.

***Frequency Sweep**.*
The user can select a single frequency or a frequency sweep to calculate the field.

- ***Enable Sweep***: select this option to simulate a range of frequencies.
- ***Units***: to set the magnitude of the frequency. Hz, kHz, MHz and GHz are available.
- ***Initial frequency***: the first step in the frequency range, in the selected *Units*. If the frequency *Sweep* is disabled, this value coincides with the simulation frequency.
- ***Final frequency***: the last step in the frequency range, in the selected *Units*. This parameter may be only modified when the frequency *Sweep* is enabled.
- ***Samples***: number of frequency samples to be analyzed on the frequency *Sweep*. This parameter may be only modified when the frequency *Sweep* is enabled.

***Additional Effects**.*
It is also possible to specify additional effects.

- ***Number of bounces***: It is possible to define the number of interactions (bounces) to be taken into account in the simulation for each observation point.
- ***Diffracted***: if this option is selected, the simulation will be performed taken into account diffraction.
- ***Creeping***: it this option is selected, the simulation will be performed taken into account creeping waves.
- ***Transmission***: if this option is selected, the simulation will be performed taken into account transmission.
- ***Generate Ray-Tracing***: enable this option for obtaining the ray-tracing results. If this option is disabled, the *Show Results - Ray Tracing* option will not be available after the simulation.
- Selecting ***Advanced Parameters*** button, the following dialog will appear:

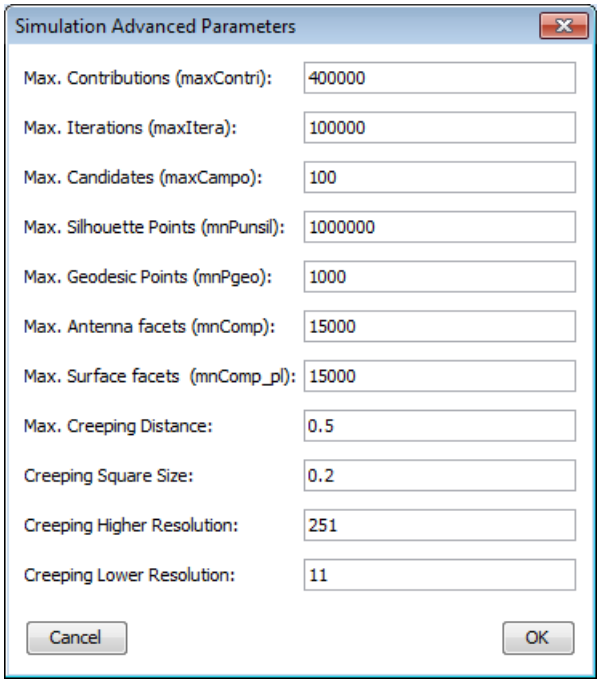


Figure 836: Advanced parameters dialog

In this dialog the user can modify several advanced parameters of the simulation, most of them relative to the analysis of the creeping waves:

- **Max. contributions** maximum number of contributions to taken into account. A contribution is for instance that due to a given double ray-path.
- **Max. iterations** maximum number of contributions that are taken into account.
- **Max. candidates** maximum number of surfaces that can shadow other surfaces.
- **Max. Silhouette Points** maximum number of points for the silhouette when creeping wave paths are computed.
- **Max. Geodesic Points** maximum number of geodesic points when computed a creeping wave path.
- **Max Antenna facets** maximum number of facets to be stored in the anxel of Z-Buffer matrix of the antenna.
- maximum number of facets to be stored in the anxel of Z-Buffer matrix of every surface.
- **Max Creeping Distance** maximum distance between a surface and the antenna to increase the creeping sample resolution.
- **Creeping Square Size** size edge of the square where the creeping sample resolution is increased near the antenna.
- **Creeping Higher Resolution** higher sampling resolution per surface used to compute the creeping silhouette.
- **Creeping Lower Resolution** lower sampling resolution per surface used to compute the creeping silhouette.

After setting up the simulation parameters it is necessary to press the **Save** button to save the configuration changes.

9.6.2 Creeping

If the creeping effect has been selected in the Simulation Parameters Panel, the Creeping parameters option in the Simulation Parameters menu will be available. When this option is selected, the following panel will appear:

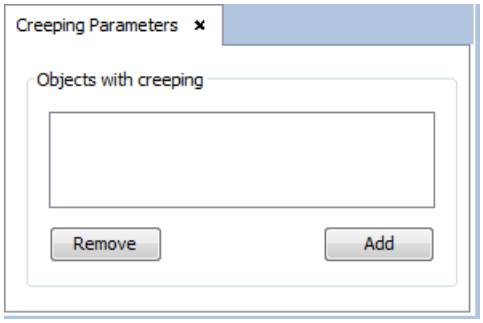


Figure 837: Creeping parameters panel

In this panel, we can add a creeping effect to objects in the geometry. To do this, select an object from the geometry view and press the Add button. The selected object will be added to the list of objects with creeping. It is possible to remove the creeping effect from an object by selecting it from the list and pressing the Remove button.

9.6.3 Doppler

If the Doppler simulation type has been selected in the Simulation Parameters panel, the Doppler option will be available. When this option is selected, the following panel will appear:

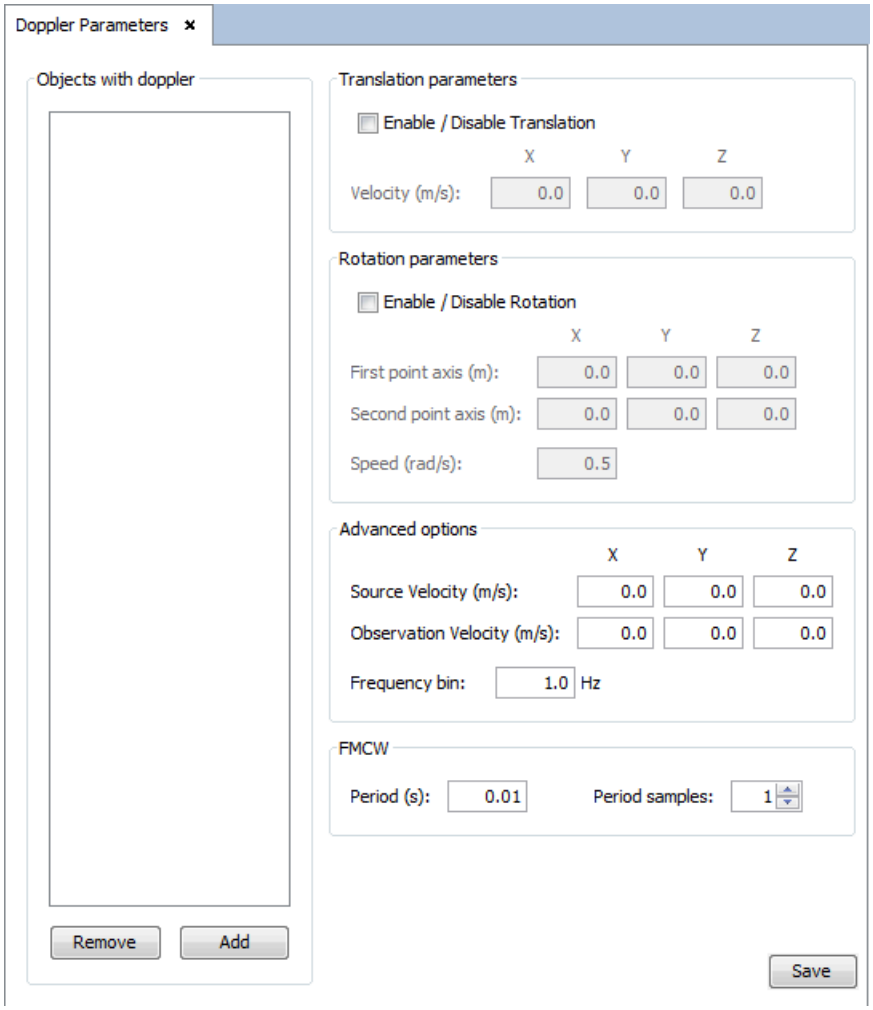


Figure 838: Doppler Parameters panel

In this panel, we can add Doppler Effect to objects in the geometry. To do this, select an object from the geometry view and press the Add button. The selected object will be added to the list of objects. It is possible to remove the Doppler Effect from an object by selecting it from the list and pressing the Remove button.

- **Translation Parameters.** Any object in the list could have a translation movement. Using this panel the user can define the linear velocity of the object for each coordinate. The Enabled/ Disabled Translation check box must be selected.
- **Rotation Parameters.** Any object in the list could have a rotation movement. The Enabled/ Disabled Rotation check box must be selected. Using this panel the user can define:
  - First point axis coordinates of the first point axis.
  - Second point axis coordinates of the second point axis.
  - Speed rotation speed.
- **Advanced Options.** Using this panel the user can define:
  - Source velocity Using this panel the user can define the linear velocity of the source for each coordinate.

  - Observation Velocity Using this panel the user can define the linear velocity of the observation for each coordinate.
  - Frequency bin defines the frequency resolution for de doppler shift analysis.

- **FMCW.** This option allows the user to compute Doppler Effect centered with the analysis of continuous wave FM radar. This option performs one meshing and one simulation for each of the periods defined considering the geometry located in the corresponding positions following the defined movements per unit of time and the periods for Doppler Effect. Using this panel the user can define:
  - Period Duration, in seconds, of the period of the FMCW wave.
  - Number of periods number of periods.

Once the parameters of the movement of one object have been defined, it is necessary to click on **Save** button in order to save this information and to assign it to the object selected.

9.7 RCS Menu

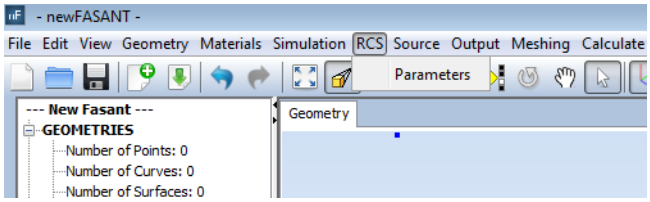


Figure 839: RCS menu

9.7.1 Parameters

Selecting this option the user indicates the type of observation to be performed. If the Bistatic option is selected, the user must define the angle of incidence using the Source menu. In both cases, the output parameters must be specified using the Output menu.

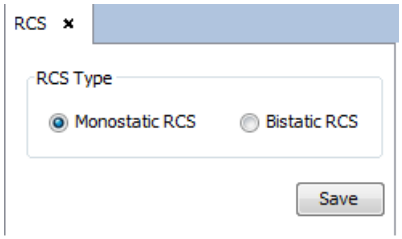


Figure 840: RCS panel

9.8 Source Menu

The options in this menu allow the user to specify the source. The menu contains the following options:

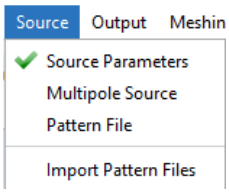


Figure 841: Source Menu

9.8.1 Source Parameters

When this option is selected, the following panel will be shown:

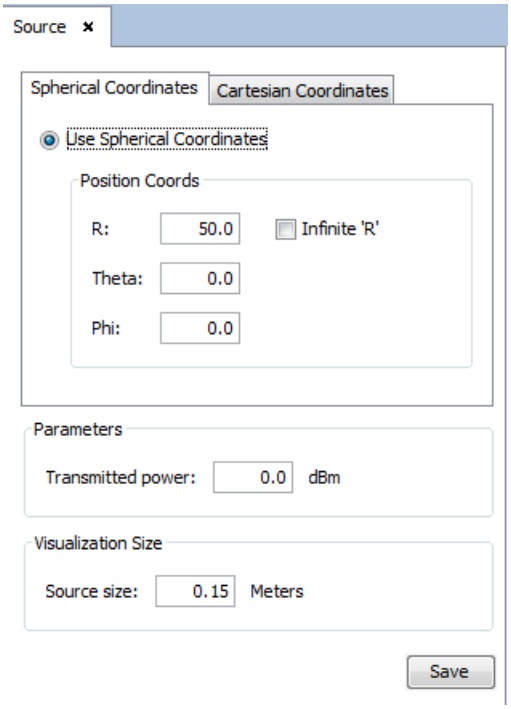


Figure 842: Source Parameters panel (spherical coordinates).

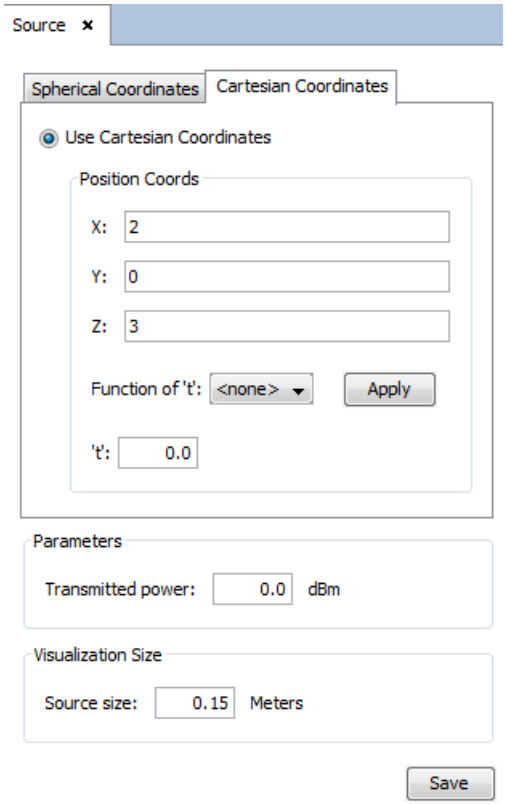


Figure 843: Source Parameters panel (Cartesian coordinates).

In this panel the user can set:

Spherical Coordinates
: when **Use Spherical Coordinates** radio button has been selected, the spherical coordinates (R, theta and phi) must be specified to define the source position.

Cartesian Coordinates
: when **Use Cartesian Coordinates** radio button has been selected, the Cartesian coordinates (x, y, z) must be specified to define the source position. In this case, it is possible to define the coordinates values with a function and the parameter t.

Transmitted power:
: This option allows the user to specify the transmitted power of the source.

Source size
: This option allows the user to scale the antennas represented in the geometry view. The user needs to indicate the factor of scale in meters.

9.8.2 Multipole Source

When this option is selected, the following panel is shown:

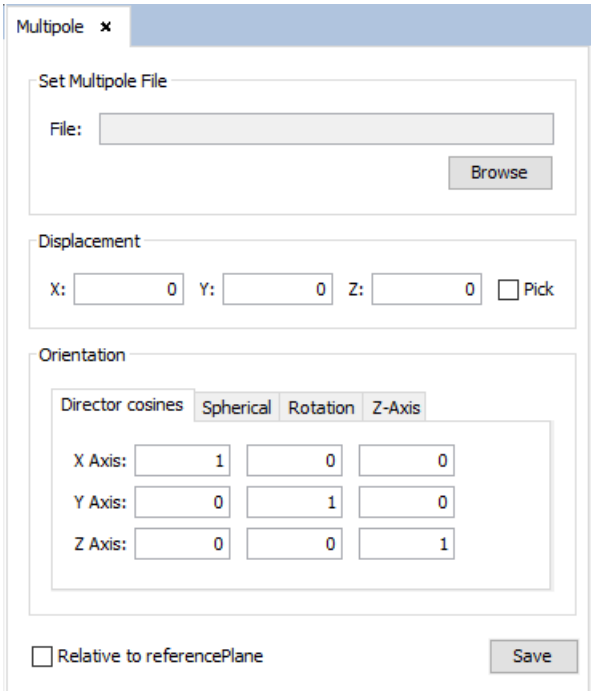


Figure 844: Multipole antenna panel

In this panel, the user can select a multipole antenna file (`.suj`) from the local file system. This file contains the specification for a multipole antenna arrangement. The user can also specify a transformation to apply to the loaded multipole. The transformation consists on a displacement (given by a three coordinate vector) and a rotation (given by the X, Y and Z axes of the local coordinate system of the rotation). If the **Relative to reference plane** check box is selected, the entered displacement and rotation will be relative to the reference plane coordinate system. Pressing the **Save** button will add the multipole antenna.

9.8.3 Pattern File

This option allows the user to introduce several files (at different frequencies) that define the radiation pattern. Firstly, the user has to import a radiation pattern file using the Import DIA file option of the Antenna menu. This options is not available if RCS&Field has been selected as the simulation type.

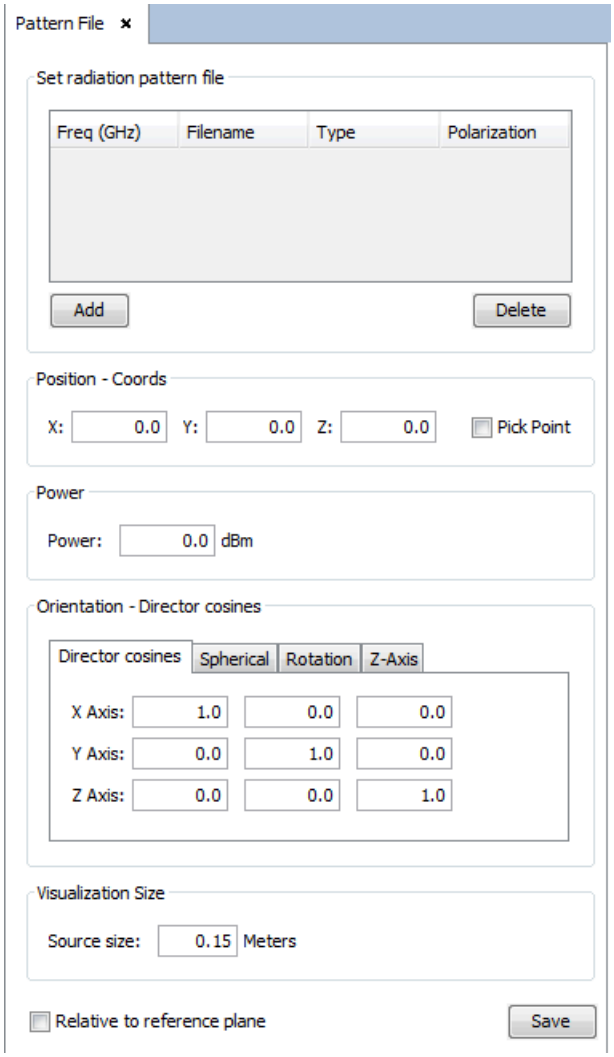


Figure 845: Pattern File Antenna panel.

The following parameters may be edited:

Set Radiation pattern file
: Click on the Add button to add the imported file to the table where it is possible to specify the frequency, the type and the polarization of the file.

Position - Coords
: location of the antenna (X, Y and Z).

Power
: radiation power of the file in dBm.

Orientation - Director cosines
: orientation of the antenna coordinate system related to the absolute coordinate system.

Visualization size
: This option allows the user to scale the antennas represented in the geometry view. The user needs to indicate the factor of scale in meters.

The required format of the radiation pattern file is described in DIA Files.

9.8.4 Import Pattern Files

Before defining an antenna as a radiation pattern file, it is necessary to import the file. To do this, click on the left icon and select the file to import. Finally, this file will appear in the panel.

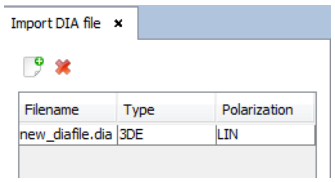


Figure 846: Import DIA file panel

9.9 Output Menu (Alt +O)

This menu allows the user to set different parameters that determine what results are calculated in the simulation. Depending on the simulation type selected in the **Simulation Parameters** menu, the options shown in this menu will be different:

- If " **RCS & Field**" or "**Field**" simulation type are selected, the **Observation Directions** option will be shown in the Output menu.
- If " **Coupling**" simulation type is selected, the **Coupling** option will be shown in this menu.
- If " **Doppler**" simulation type is selected, the **Output** menu is not enabled.

Next, the different Output options are explained in detail:

9.9.1 Observation Directions

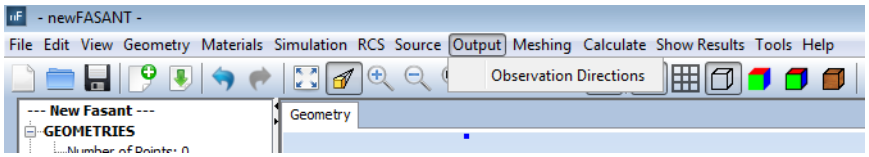


Figure 847: Output Menu

This option allows the user to set the position of the observation.

- **Spherical observation**. If the position of the source has been defined in spherical coordinates, this option will be available. It is possible to define a parametric sweep for one variable (radius, theta or phi) selecting the check box Source Position Sweep. The initial value for R, theta and phi have to be specified.
  - **Final value** final value for the chosen variable (R, theta or phi).
  - **Samples** number of samples.

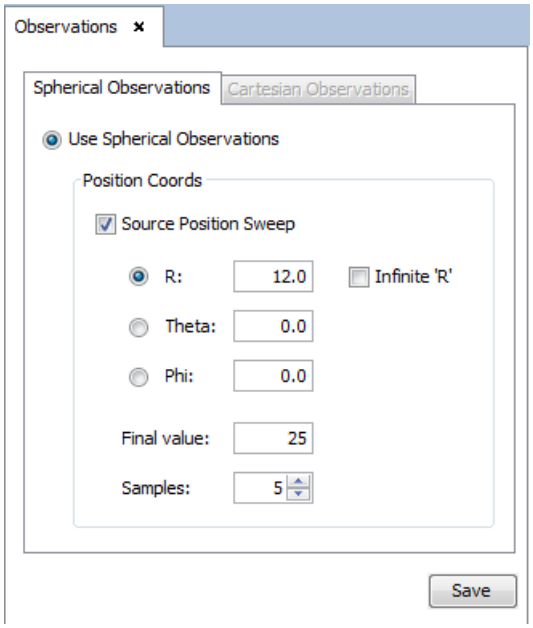


Figure 848: Spherical observation panel.

- **Cartesian observation**. If the position of the source has been defined in Cartesian coordinates, this option will be available. It is possible to define a parametric sweep on t selecting the check box Sweep on 't'. The coordinates x, y and z can be defined with a function which values could depend on the parameter 't'. The initial value for R, theta and phi have to be specified.
  - **Initial 't'** the initial value for t.
  - **Final 't'** the final value for t.
  - **Samples on 't'** number of samples.

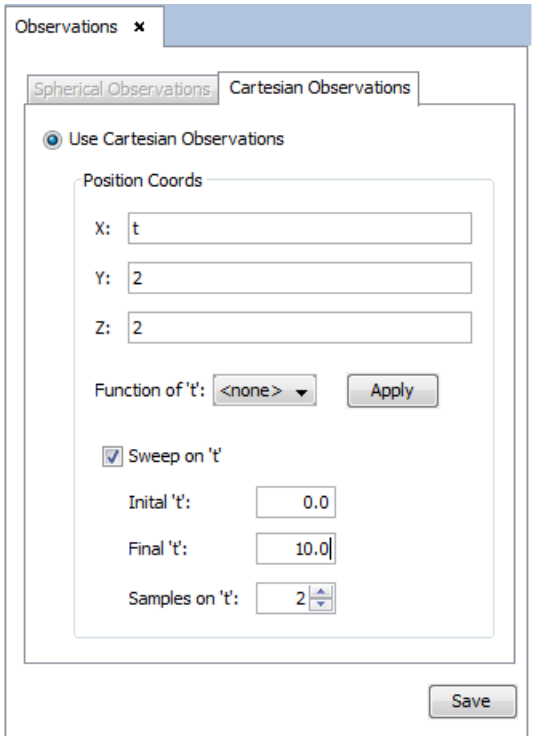


Figure 849: Cartesian observation panel

9.9.2 Coupling

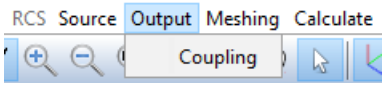


Figure 850: Output Menu

If the user selects the**Coupling**option (which is only available if**Coupling**simulation type is enabled), the following panel is shown:

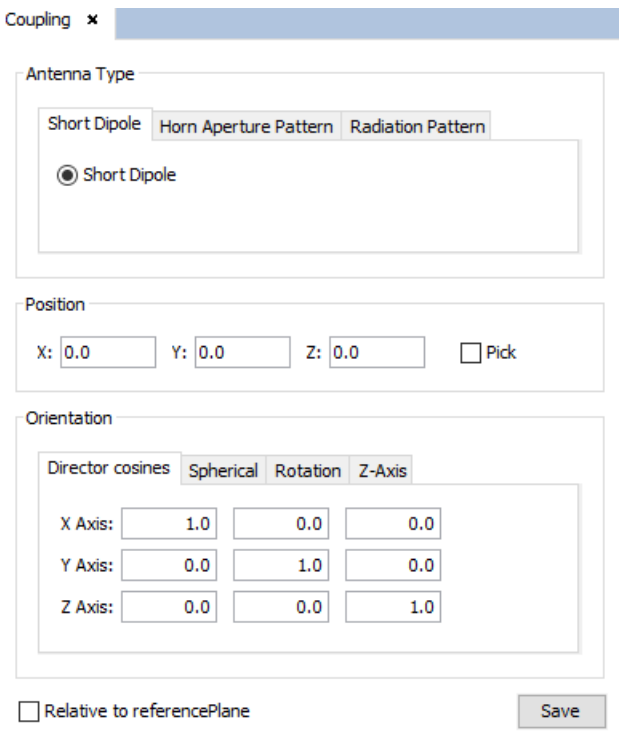


Figure 851: Passive antenna configuration panel

The following parameters may be edited:

- **Antenna Type** the passive antenna can be:
  - **Short dipole**: the passive antenna is defined by a short dipole.
  - **Horn aperture pattern**: The user has to select the pattern file of a horn antenna and its polarization vector (given by its real and imaginary parts for each coordinate). The required format of the radiation pattern file is described in DIA Files. This option will be selected when the pattern file to be used as the definition of the passive antenna, correspond to a horn aperture.
  - **Radiation pattern file**: The user has to select the pattern file of an antenna. The format of the radiation pattern file can be **REV**, **RV2** or **3DE**, and the polarization can be linear (**LIN**) or circular (**CIR**). The required format of the radiation pattern file is described in DIA Files. This option will be selected when the passive antenna is defined by a radiation pattern file.
- **Position-Coords**: location of the passive antenna (from the keyboard or using the mouse with ' **Pick Point Edition**' option).
- **Orientation Director Cosines**: orientation of the passive antenna coordinate system related to the absolute coordinate system.

If all the information is correct, the antenna will be shown on the screen. A red parabola represents the active antenna and a magenta parabola the passive antenna. The axis of the antenna is also visualized. If the antenna is not visualized on the screen after pressing **Save** button, the most common reason is that the radiation pattern has not been found in the **NewFasant** installation folder.

9.10 Meshing Menu

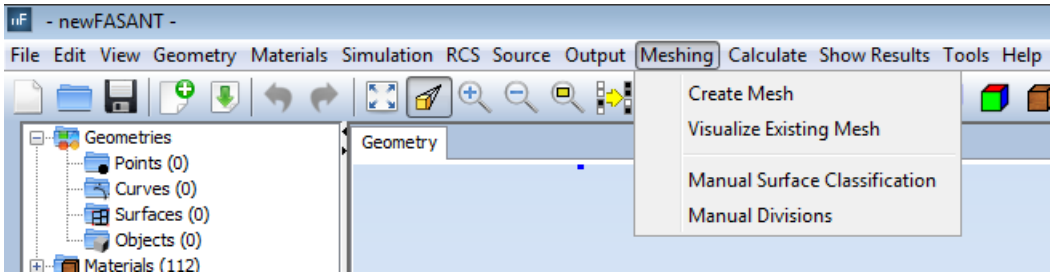


Figure 852: Meshing Menu

9.10.1 Create Mesh

In the GTD-PO module, the mesh generation is a required step that is performed immediately before simulating. In the Meshing Parameters window, represented in the next Figure, the following options are available:

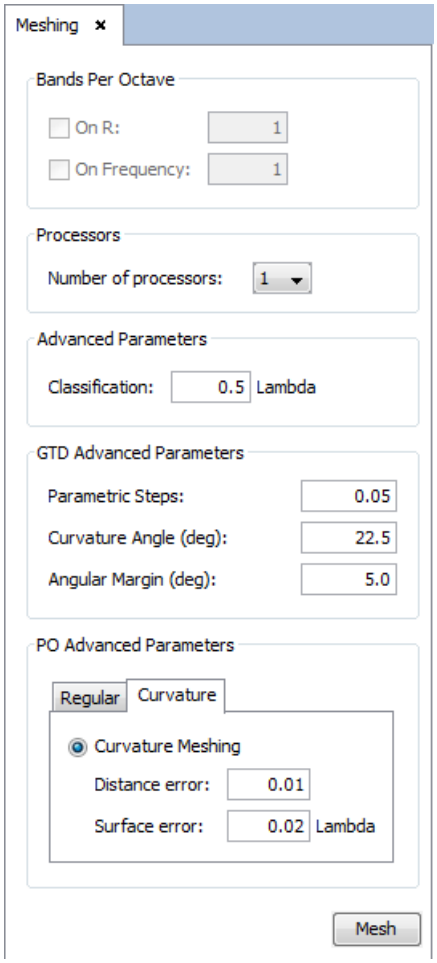


Figure 853: Parameters panel

- **Bands Per Octave**. Set the different meshes to be generated, according to the bands per octave divisions. If no options are chosen, just a mesh at the lowest simulation frequency is generated. This option is only highlighted when previously a frequency shift or a distance to source sweep are selected.
  - **On R**: the octaves are considered according to the distance to the source. This option only can be set when the source is defined by spherical coordinates.
  - **On Frequency**: the octaves are considered according to the frequency range.
- **Number of processors**. Sets the number of processors for the meshing process.
- **GTD Advanced Parameters**: the user can change the curvature angle and the angular margin in order to control how the curved surfaces of the geometry will be partitioned. The contributions are previously evaluated on a parametric internal mesh of the geometry where the curved surfaces are simplified. The resolution of this mesh may be edited according to the below parameters, which are set by default with the recommended values.
  - **Parametric Steps**: accuracy in the curvature evaluation, normalized. Decrease this parameter to get a higher precision in the parametric discretization.

- **Curvature angle (deg)**: maximum curvature considered on targets. It indicates the maximum curvature allowed in the input surfaces for the meshing step. Decrease this parameter to get a higher resolution mesh.
- **Angular Margin (deg)**: maximum deviation allowed around the curvature reference angle. The curvature discretization is achieved by an approximated iterative method, so an angular error around the previous parameter is allowed. Decrease this parameter to increase the precision in the curvature discretization.

- **PO Advanced Parameters**
  - **Regular Meshing**. The size of the generated elements is selected by the user. Regular meshing is the common algorithm used in other modules such as MOM. It provides a mesh of triangles of similar sizes with a distribution as uniform as possible. It enables the user to set the exact size edge of the triangles to be generated, given in lambda units. However, a huge number of mesh elements may results when the geometry contains large surfaces.
    - Multilevel. Use Multilevel to automatically generate the mesh in several sequenced steps. This option is usually more efficient (in terms of runtime) than the common mesh generation option, so it is selected by default. However, there may be minor differences between the meshes obtained with and without the Multilevel mode.
    - Edge size. This value is the desired size for the edge of the elements, in wavelengths.

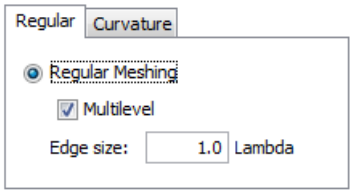


Figure 854: Regular meshing

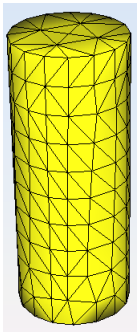
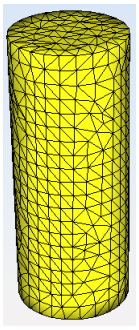
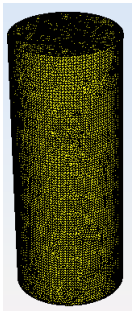
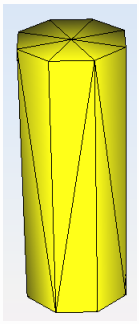
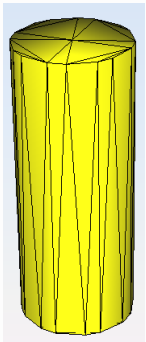
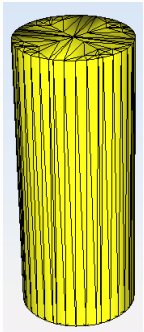
  - **Curvature Meshing**:

    Using curvature meshing we get the meshed geometry defined in terms of triangles. This geometry should be very similar to the original geometry. We can force that the meshed geometry be close to the original one using the surface error (in wavelengths). In the mesher we have default values for this magnitude but the user can change it. The mesher tries to minimize the number of triangles of a meshed geometry such that the distance from any point of this geometry to the original one is smaller than the "selected surface error".

    If we have a cube, say the original geometry is defined by six flat quadrangles the meshed geometry is composed by twelve triangles. In this case the distance of any point of the meshed geometry to the original one is null, say both geometries are identical for every frequency.

    If an sphere of 1 m of radius at 300 MHz the "selected value" using the defaults values is 0.08*wavelength = 8.0 cm. The mesh is composed by 104 triangles. If you increase the frequency we obtain the same mesh until about 0.7 GHz. Due to topological restriction the number of elements of a mesh of triangle can not increases continuously, say we can not go from 104 triangles to 105 triangles for the sphere (next for the sphere can be 174 triangles).

The **Distance error** error is other parameter of the meshing by curvature. It only applies to flat facets. The longer edge of the planar surface is split by this parameter. For instance if you set 10 for the cube case you get $12*(10^2) = 1200$ triangles. By default we have 0.01, that is, if this parameter is lower than 1.0 not division of flat facets is consider.

	1 lambda	0.5 lambda	0.01 lambda
Regular			
	0.2 surface error	0.1 surface error	0.01 surface error
Curvature			

The user may select the number of processors used to run the simulation and press the **Mesh** button to start the meshing process. This process may take a while (depending on the number of processors selected and the computing power of the machine running the simulation).

While running, the status of the meshing process is reported in the process log panel. From this panel, it is possible to abort the running meshing process or save the log to a text file.

9.10.2 Visualize Existing Mesh

When this option is selected, the following panel is shown, in which the user can select the step, the frequency and the radius. Only when a Doppler type of simulation is going to be performed, it is also possible to select the Doppler step.

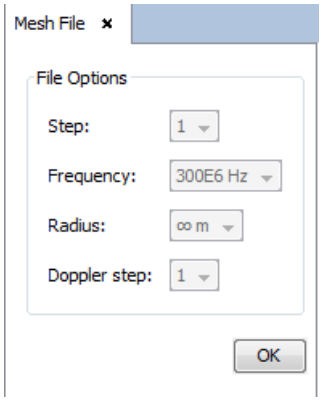


Figure 855: Meshing file panel

It is possible to visualize the mesh as shown in the following Figure. This option is useful to obtain information about which surfaces are going to be analyzed by GTD (coloured in red) or by PO (coloured in blue).

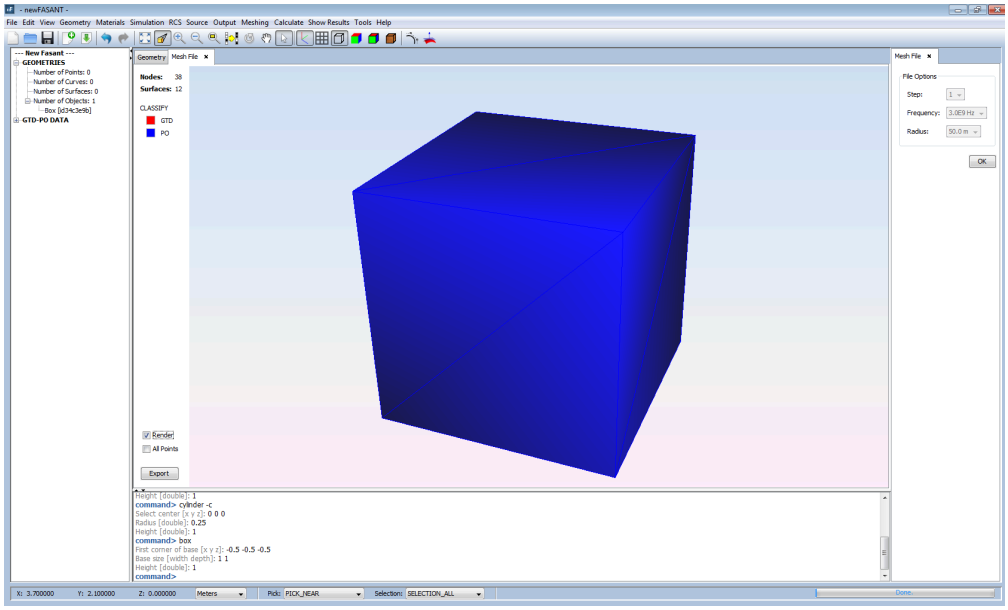


Figure 856: Visualize Existing Mesh

9.10.3 Manual Surface Classification

With this option, the user can perform a manual classification of the geometry.

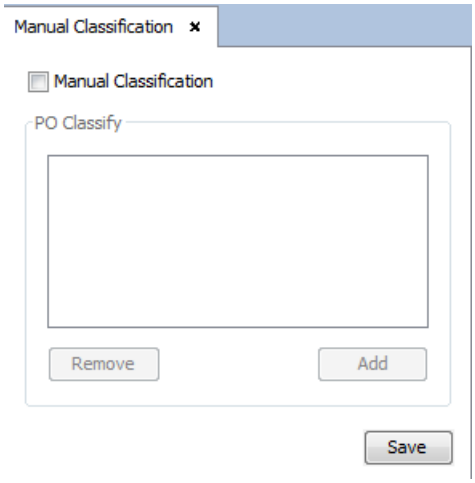
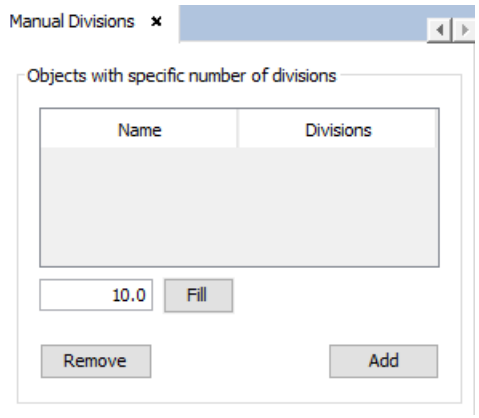


Figure 857: Manual classification panel

Once the check box is selected, the panel box is available. The user can select the surfaces from the scene to be added to the list. This surfaces will be classified automatically as PO surfaces. To remove any surface from the list, select the surface and click on Remove button.

9.10.4 Manual Divisions

With this option the user can set how many divisions are used for meshing the selected the geometry. It may be useful for having higher or lower level of detail in some targets.



The user can select the surfaces from the scene to be added to the list by clicking on Add button. By default, the new objects added to the list are meshed by using 10 Divisions per wavelength, but it may be edited in two different ways clicking twice in the Divisions cell of the list element to be edited and pressing Enter key after modifying the number of divisions; or selecting the row to be edit and clicking on Fill button with the new number of Divisions. Click on Remove button to delete the selected rows from the list.

9.11 Calculate Menu (Alt + C)

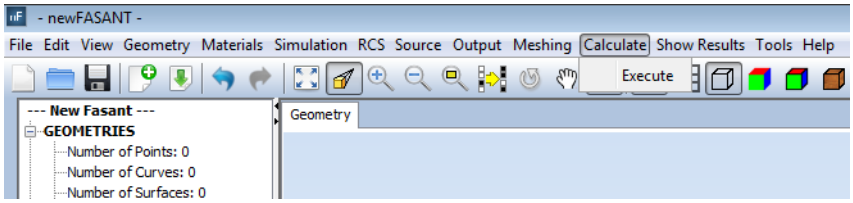


Figure 858: Calculate Menu

Before executing the simulation the user needs to make sure that every simulation parameter has been correctly configured. If something is wrong with a user defined parameter a warning message is displayed to let the user know that it is necessary to correct one or more parameters.

9.11.1 Execute

This option runs the simulation. The user can define the number of processors.

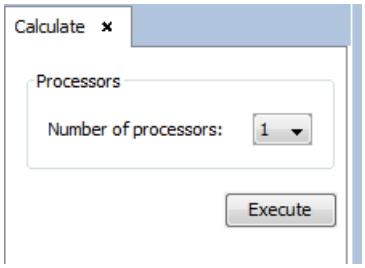


Figure 859: Calculate Menu panel

When the simulation starts, a new screen is displayed:

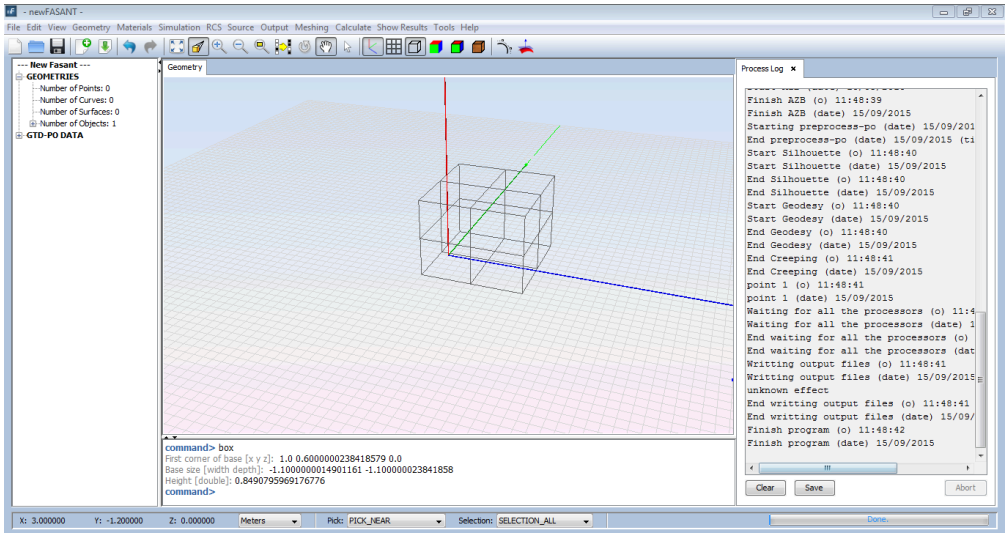


Figure 860: Processing screen

During a solution process, the screen reports the status of each phase of the calculation process date and CPU time for every phase of the solution process, etc.

This data can be saved by the user by clicking the Save button, as shown in the next Figure.

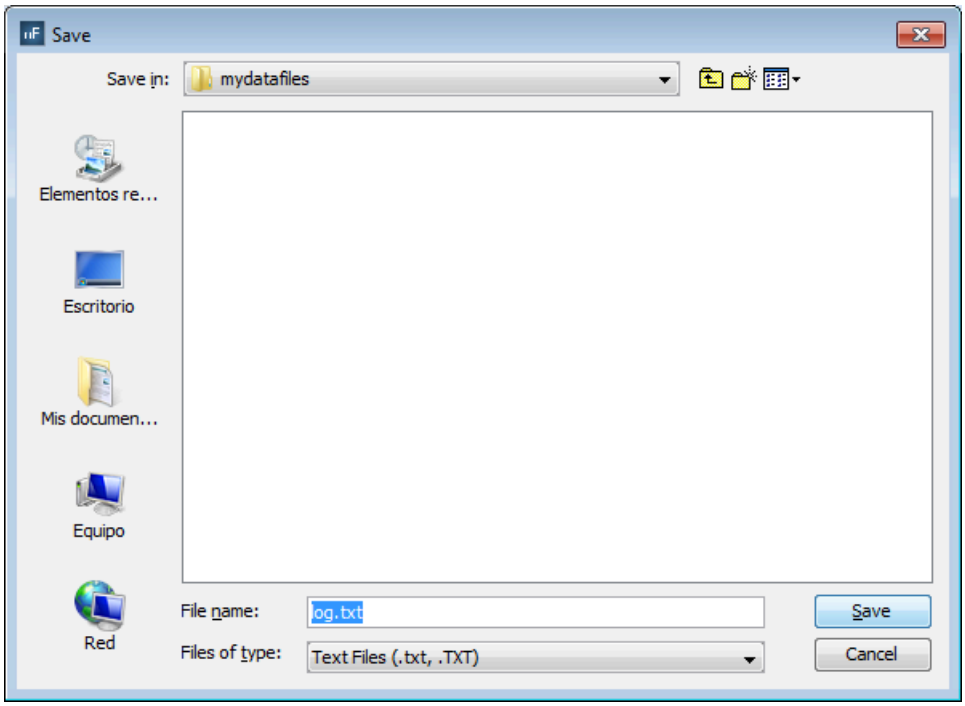


Figure 861: Save processing window

9.12 Show Results Menu

This menu allows the user to visualize the results of the simulation. The results are stored in external files that the user can access to obtain the information corresponding to that simulation.

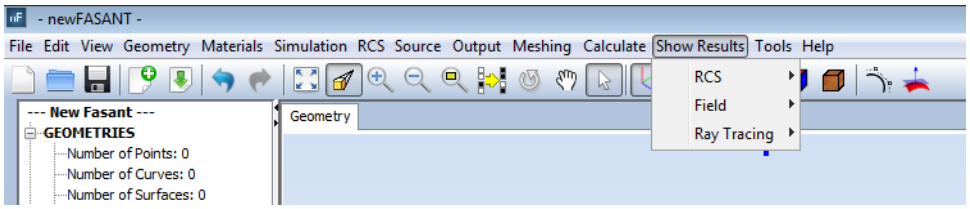


Figure 862: Show Results menu

Depending on the type of the simulation, the list of possibilities results to open is different:

- RCS&Field simulation. This type of simulation has RCS, Field and Ray Tracing results.
- Field simulation. This type of simulation has Field and Ray Tracing results.
- Coupling simulation. This type of simulation has Coupling and Ray Tracing results.
- Doppler simulation. This type of simulation has Field, Ray Tracing and Doppler results.

Most of plots show 2-D curves of a result field versus an input parameter. This type of graphics share additional features of zoom, raxis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

9.12.1 RCS

Using this menu it is possible to analyze the amplitude of the RCS through the observation directions previously defined in the Output section.

The following options are available:

- View Cuts: display a plot that relates the amplitude against a Theta cut or a Phi cut.
- View Cuts by Frequency: displays a plot that relates the amplitude against a frequency sweep.
- View Cuts by Step: displays a plot that relates the amplitude against a parametric step.
- View Text Files: displays the numeric results obtained in the simulation process.

View Cuts

This command plots the amplitude of the RCS fields computed for each angular cut of the simulation geometry. Several frequencies can be plotted in the same graphic. To plot a graph, the user can select the RCS component and the frequency. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import**

Series' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a data file written as a text file.

This panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against every possible frequency that has been simulated. More series can be added, by selecting component, step and spherical coordinates in the control panel that appears leftside to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Frequency:** select a particular frequency.

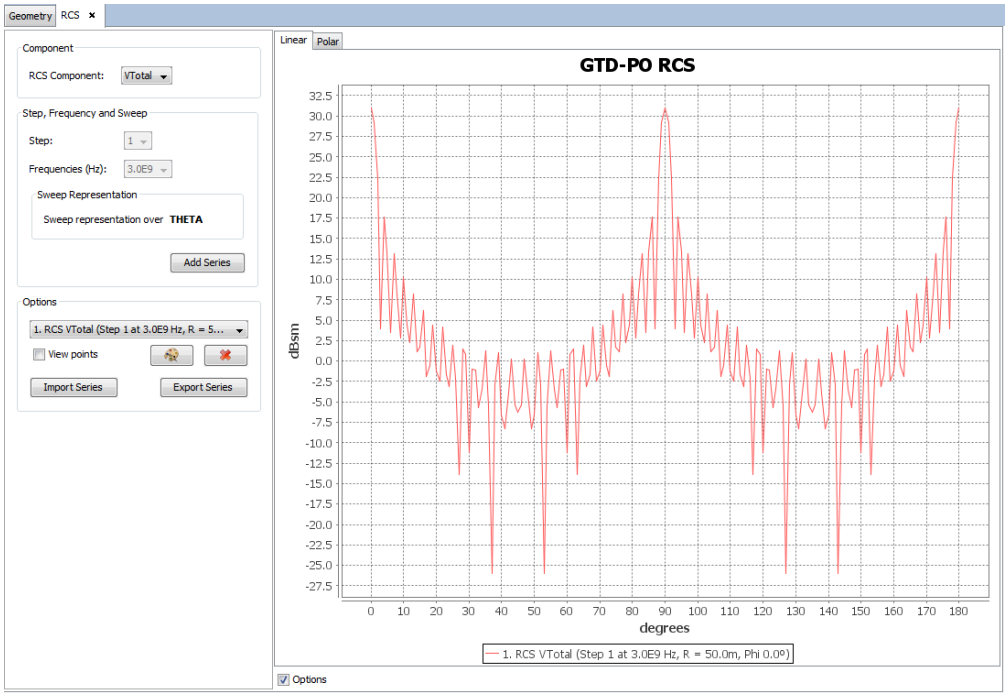


Figure 863: Cuts Linear window

The user can also draw a polar plot with the same options used in the previous window.

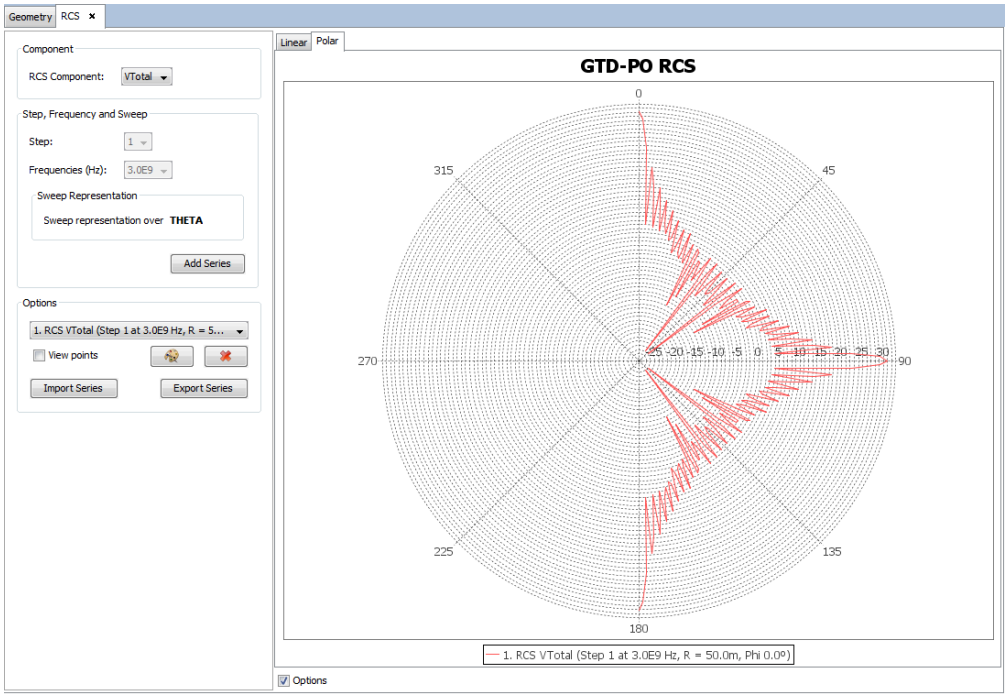


Figure 864: View Cuts Polar window

View Cuts By Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, cut and point in the sweep, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation Settings, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the Show Results > RCS > View Cuts By Frequency menu, the panel presented in the figure below will appear.

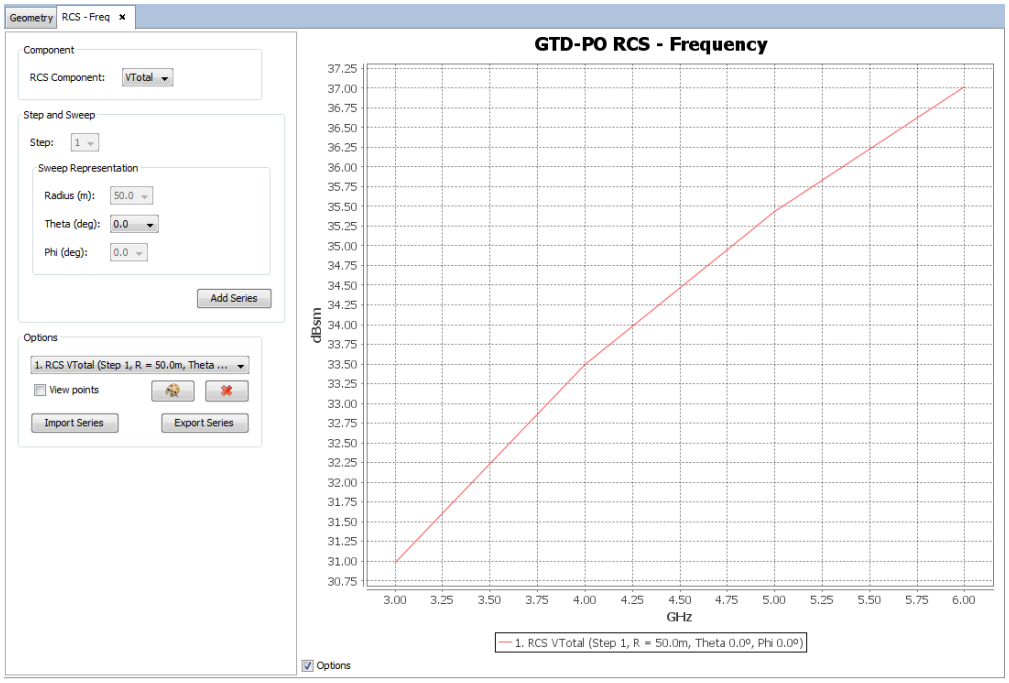


Figure 865: View Cuts By Frequency window

Similar to the behaviour of the standard View Cuts, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and radius, theta or phi, against every possible frequency that has been simulated. More series can be added, by selecting component, step and sweep representation information in the control panel that appears left-side to the plot.

- **Component**: the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Sweep representation**: select one value for one of the variables radius, theta or phi.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Cuts By Step

This kind of plot lets the user analyze RCS measurements made by the simulation provess for a particular component, frequency and observation point against the parametric steps defined in the

project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the Show Results > RCS > View Cuts by Steps menu, the panel presented below will appear:

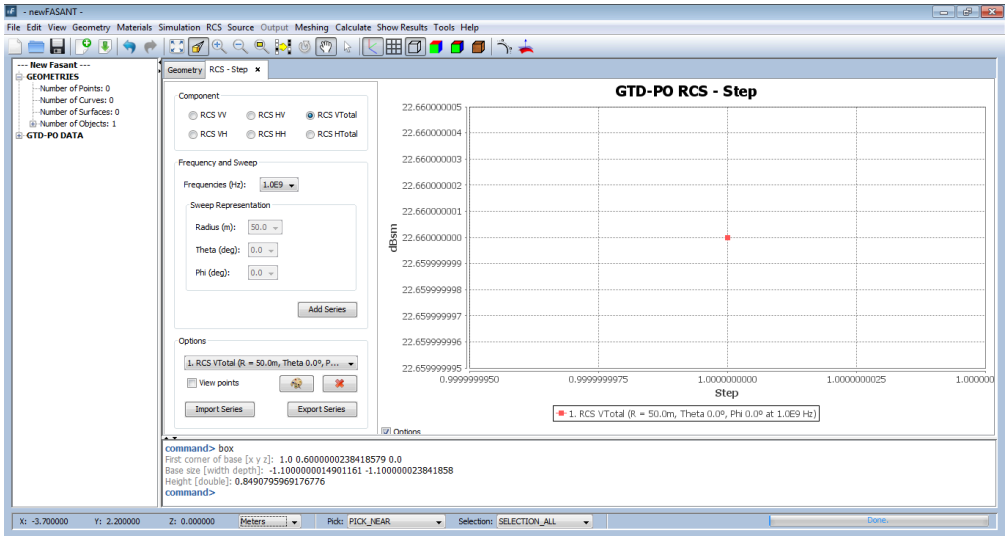


Figure 866: View Cuts By Step window

Similar to the behaviour of the standard RCS chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, frequency and observation point, against every possible step that has been simulated. More series can be added, by selecting component and frequency in the control panel that appears left-side to the plot.

- **Component:** the field component to inspect. Select one of the possible values Etheta, Ephi, Etotal
- **Frequency** frequencies that have been simulated.
- **Sweep representation:** select one value for one of the variables radius, theta or phi.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Text Files

This option shows the simulation results as a text file. First, a panel should appear after selecting this option. Selecting this option will display a panel where the user should select a step, an order, a radius and a frequency. To use this section, select a parametric step from the list and one of the orders shown

in the list to get the results from and click on **OK** to load the data. A table will be presented with the values for the electrical field. To save the result click on the **Export** button below the table.

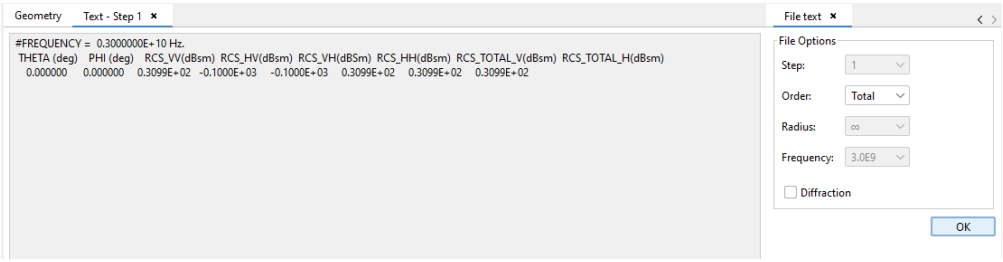


Figure 867: View Text Files window

9.12.2 Field

Using this menu it is possible to analyze the amplitude of the field through the observation directions previously defined in the Output section.

The following options are available:

- View Cuts: display a plot that relates the amplitude against a Theta cut or a Phi cut.
- View Cuts by Frequency: displays a plot that relates the amplitude against a frequency sweep.
- View Cuts by Step: displays a plot that relates the amplitude against a parametric step.
- View Text Files: displays the numeric results obtained in the simulation process.

View Cuts

This command plots the amplitude of the fields computed for each angular cut of the simulation geometry. Several frequencies can be plotted in the same graphic. To plot a graph, the user can select the field component and the frequency. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a data file written as a text file.

This panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against every possible frequency that has been simulated. More series can be added, by selecting the component, step and spherical coordinates in the control panel that appears left side to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal

- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Frequency:** select a particular frequency.

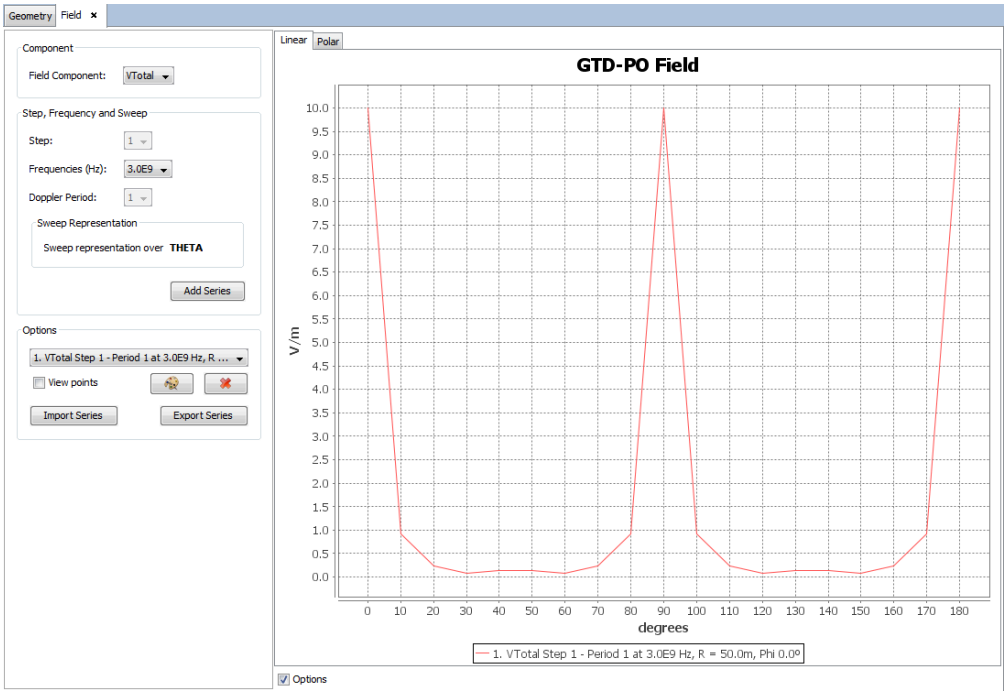


Figure 868: View cuts linear window

The user can also draw a polar plot with the same options used in the previous window.

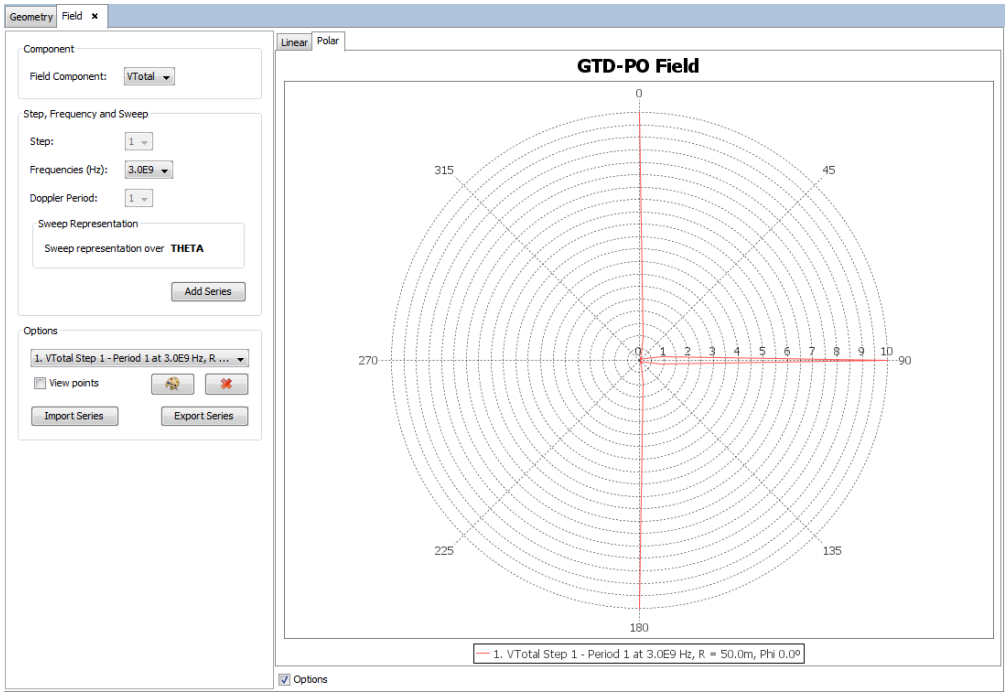


Figure 869: View cuts polar window

View Cuts by Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, cut and point in the sweep, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation Settings, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the Show Results > Field > View Cuts By Frequency menu, the panel presented in the figure below will appear.

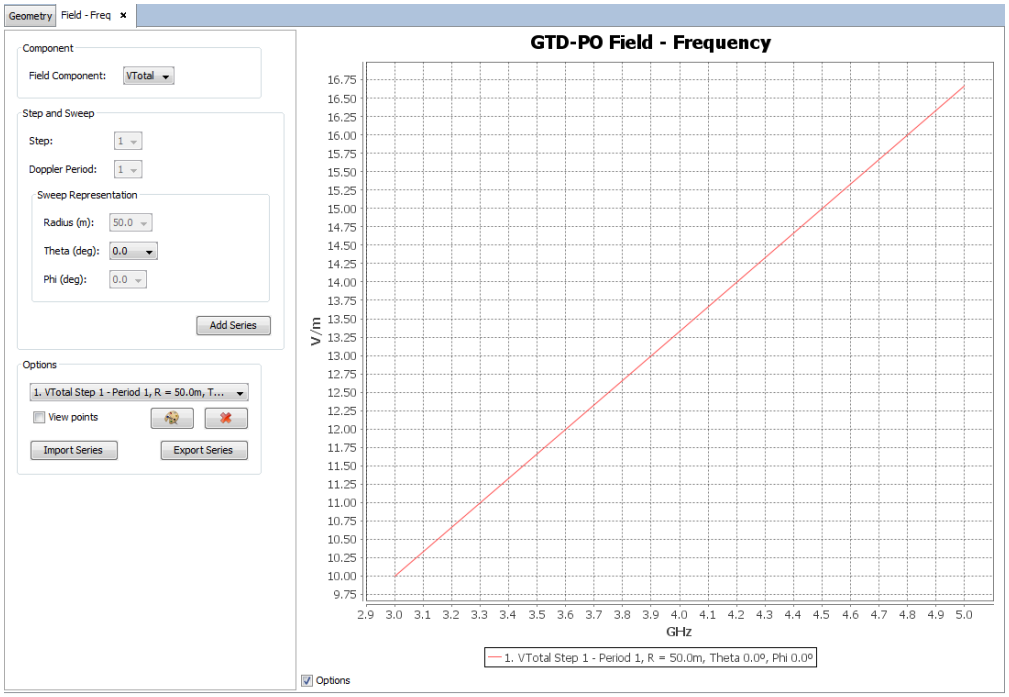


Figure 870: View cuts by frequency window

Similar to the behavior of the standard View Cuts, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation direction, against every possible frequency that has been simulated. More series can be added, by selecting the component, step and sweep representation information in the control panel that appears left side to the plot.

- **Component:** the plane wave component to inspect. Select one of the possible values VV, HV, VTotal, HH, VH, HTotal
- **Step:** the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Sweep representation:** select one value for one of the variables radius, theta or phi.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Cuts by Step

This kind of plot lets the user analyze field measurements made by the simulation proves for a particular component, frequency and observation point against the parametric steps defined in the

project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the Show Results > Field > View Cuts by Steps menu, the panel presented below will appear:

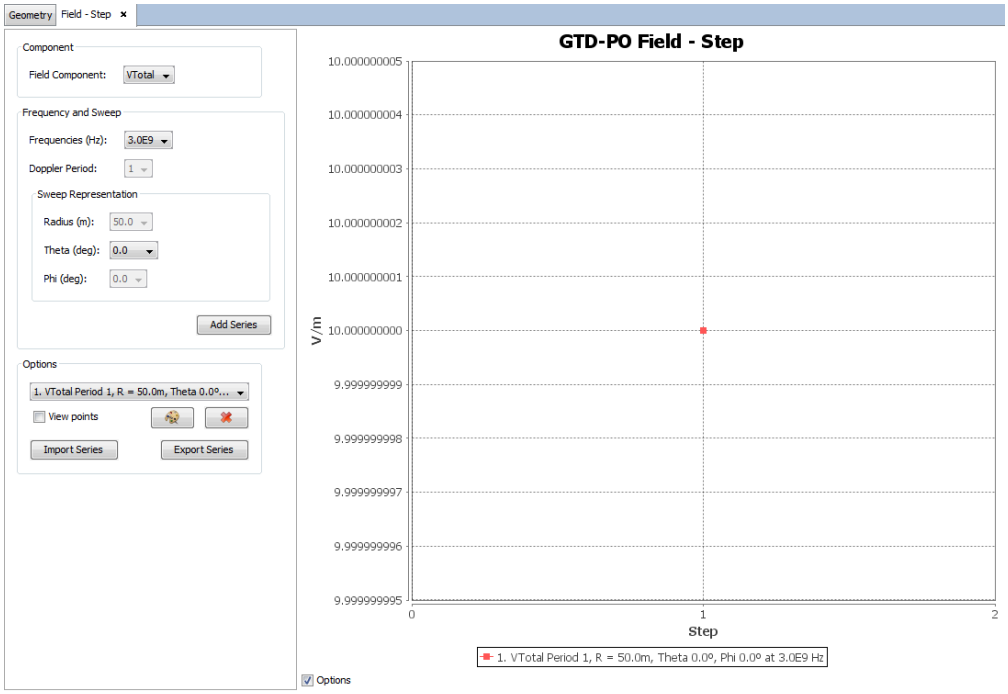


Figure 871: View cuts by step window

Similar to the behavior of the standard field chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, frequency and observation point, against every possible step that has been simulated. More series can be added, by selecting component and frequency in the control panel that appears left side to the plot.

- **Component:** the field component to inspect. Select one of the possible values Etheta, Ephi, Etotal.
- **Frequency** frequencies that have been simulated.
- **Sweep representation:** select one value for one of the variables radius, theta or phi.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**. To remove a series from the plot representation, click on the **Remove series** button.

Series can be imported and exported. To save a series click on the **Export series** button. Later it is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

View Text Files

This option shows the simulation results as a text file. First, a panel should appear after selecting this option. Selecting this option will display a panel where the user should select a step, an order, a radius and a frequency. To use this section, select a parametric step from the list and one of the orders shown in the list to get the results from and click on **OK** to load the data. A table will be presented with the values for the electrical field. To save the result click on the **Export** button below the table.

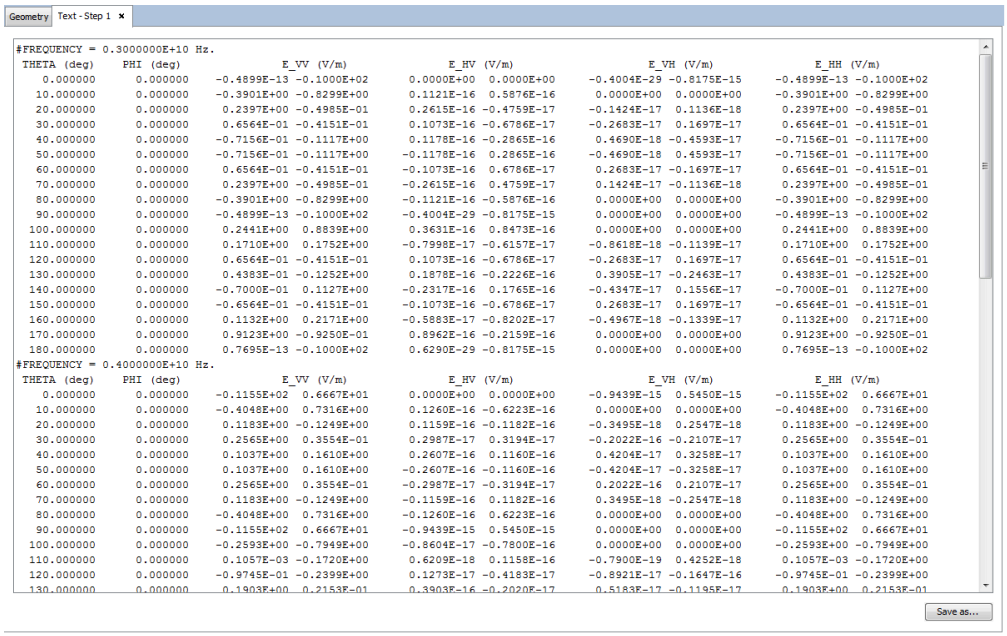


Figure 872: View text field window

9.12.3 Ray Tracing

View Ray

This option allows the user to view the rays emitted by the source in the simulation.

This command plots the rays emitted by the source in the simulation.

When initially open, the panel will display the rays for a particular step, order and observation direction. All the effects type are selected by default. The user can change:

Ray effect

it is possible to select one of this options (direct, reflected, transmitted, creeping and diffracted) or all of them.

Ray selection

- **Order**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Radius**: select a particular radius value.

- **Theta**: select a particular theta value.
- **Phi**: select a particular phi value.

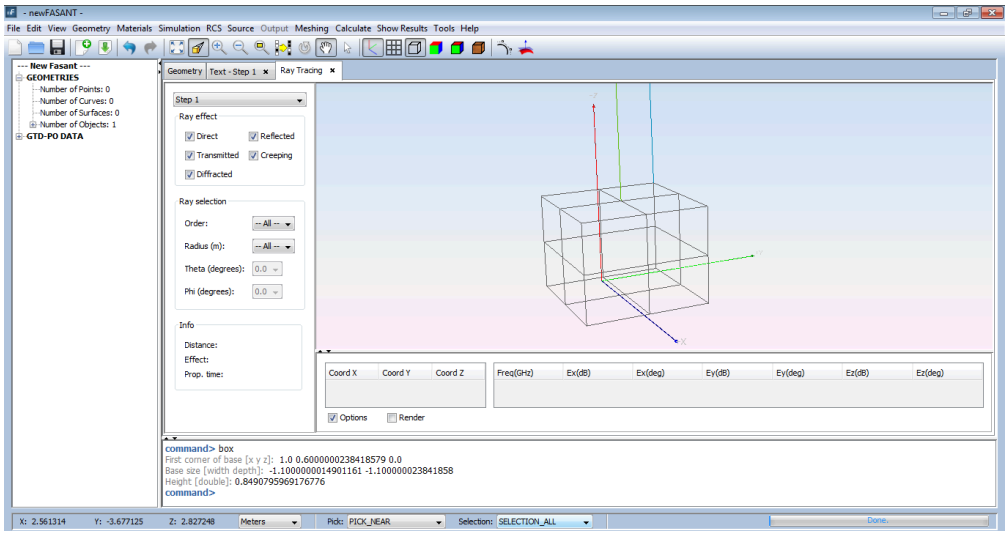


Figure 873: Ray visualization panel

The user can filter the rays that are displayed using the check boxes on the left panel. The user can also visualize the rays having a specific order.

After selecting a ray it will be highlighted in yellow color and its information will be shown in the panel at the bottom. The table on the left shows the coordinates of the points (source point, first reflection/diffraction point, second reflection/diffraction point... and the observation point). The table on the right shows the amplitude and phase for each of the field components. The values of the first line correspond to the vertical source polarization and the values of the second line correspond to the horizontal source polarization. However, if the source has been defined with a radiation pattern, this table will only have one line.

Coord X	Coord Y	Coord Z
17.101	0.0	46.9846
2.6667	-0.1667	0.5
17.101	0.0	46.9846

Freq(GHz)	Ex(dB)	Ex(deg)	Ey(dB)	Ey(deg)	Ez(dB)	Ez(deg)
3.0	-14.0887	89.256	-200.0	0.0	-22.8674	269.256
3.0	-200.0	0.0	-13.5484	89.256	-200.0	0.0

Options Render Show Axis **First line -> Vertical Polarization. Second line -> Horizontal Polarization**

Figure 874: Ray visualization information

View Text Files

When the user selects the **View Text Files** option, a panel where the user can configure the results to be shown in the generated text file appears. The user can filter the results by step and radius.

When the user presses the OK button, the text file is generated and visualized.

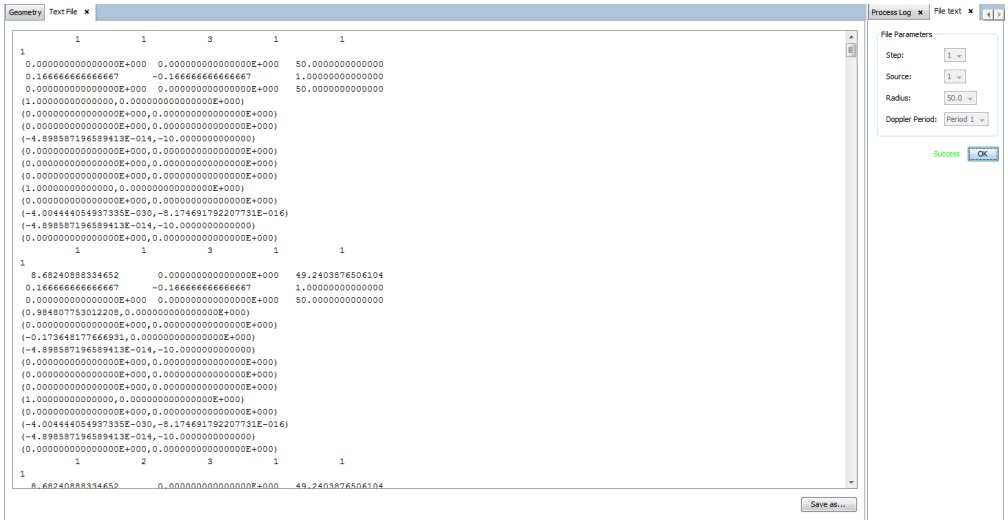


Figure 875: Showing the result text file

9.12.4 Coupling

When the user has executed a simulation of type "Coupling", the following options will be available under the Show Results menu:

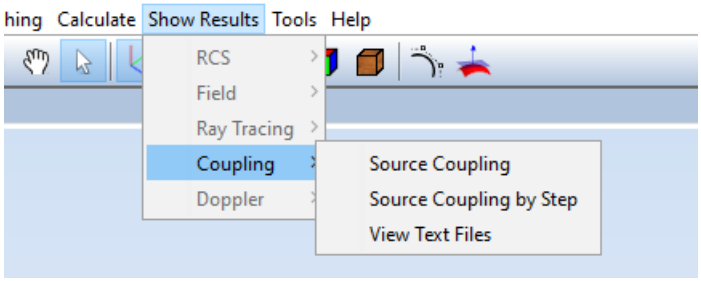


Figure 876: Coupling submenu of the Show Results menu

Source Coupling

This option allows the user to inspect the coupling values at each step, at a given frequency and a given observation point. The coupling value will be shown next to the "Antenna Set" label in dB.

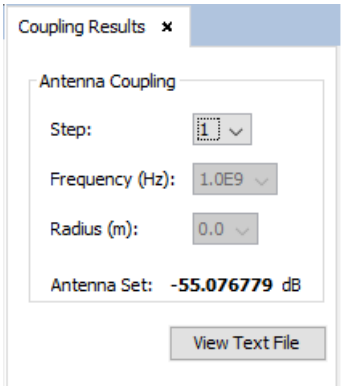


Figure 877: Viewing the source coupling values

Note The step, frequency or observation drop-down lists may appear grayed-out if there is only one available option.

The user may also want to visualize the results in text format. To do so, they need to press the "View Text File" button.

Source Coupling by Step

This option allows the user to visualize the coupling values at each step in a graph. The user needs to select the frequency, observation and polarization they want to plot the values for. When the user presses the "Add Series" button, a plot will be added.

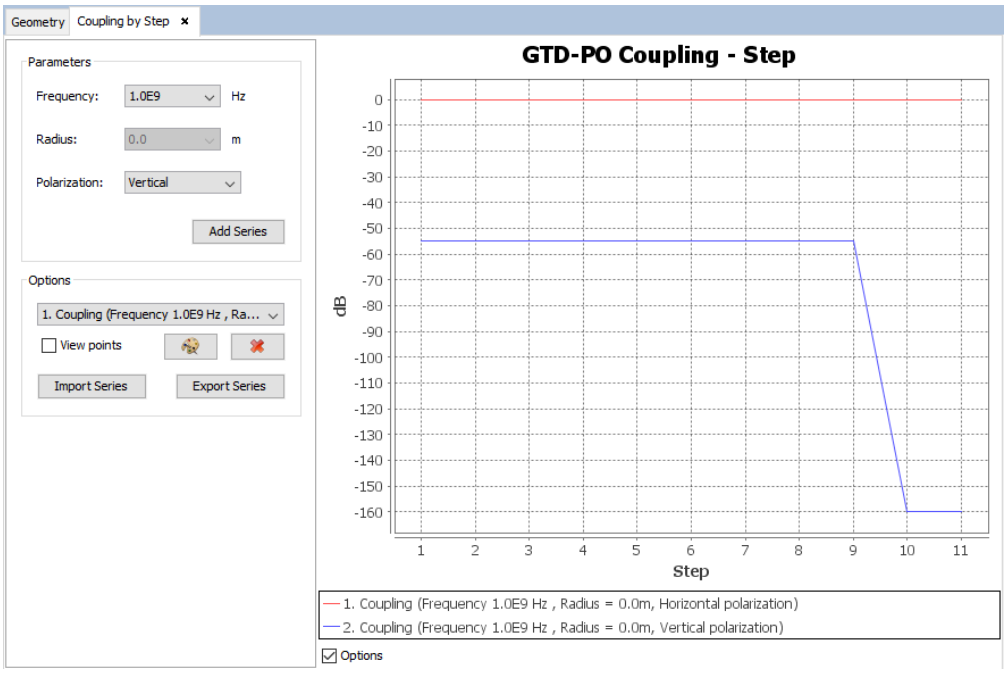


Figure 878: Visualizing coupling values by step

The user can also import and export each of the plots (series) from/to a text file. To import, the user needs to press the "Import Series" button and select the text file. Similarly, to export, the user needs

to select the desired series to export, press the "Export Series" button and select the location the user wants to save the file to. The user can also remove previously added series, as well as change their display colors.

View Text Files

This option allows the user to visualize the coupling values of the simulation for a given step and observation in text format. To do so, the user needs to select the step and observation and press the "OK" button.

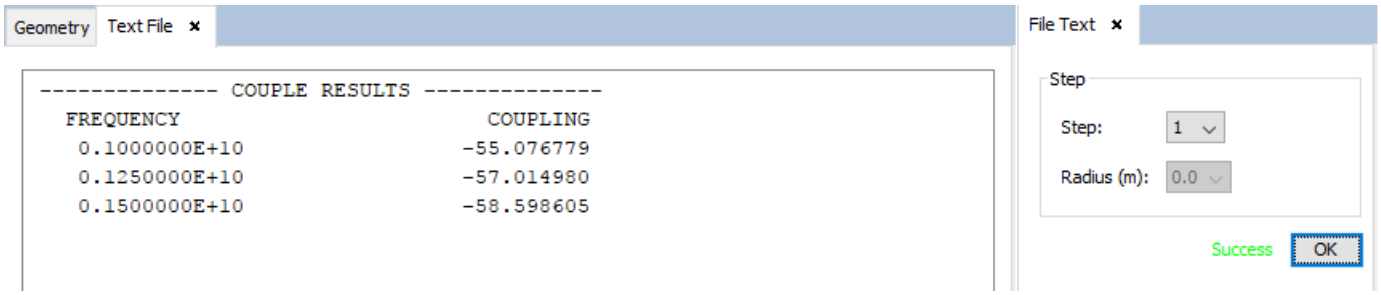


Figure 879: Viewing coupling values in text format

If the user wants to save the results to a file, they need to press the "Save as..." button located below the text area showing the results and select a location to save the file to.

9.12.5 Doppler

This option will be available when the simulation type selected in the **Simulation Parameters** window is Doppler.

Doppler Spectrum

This option allows the user to see a 2D graph of the field values for each frequency of the Doppler frequency Spectrum Shift. The frequency scale is relative to the carrier frequency, that means that a 0.0 frequency shift is for a signal that does not suffer any frequency shift (it frequency is exactly the carrier frequency).

To plot a graph, the user can select the field component, the polarization, the step, the carrier frequency and the Doppler Period (). A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry (see GUI for details), it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **'Remove Series'**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a data file written as a text file.

This panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, polarization and step. More series can be added, by selecting component, polarization and step that appears leftside to the plot.

- **Component:** select the Cartesian component of the field to visualize the results. The enabled options are Ex, Ey, Ez and ETotal.
- **Polarization:** select the polarization component of the wave to visualize the results. Select one of the possible values Vertical, Horizontal. If the source has been defined as a radiation pattern, this combo box will not be available.
- **Step:** select the simulation scene to visualize the results. These should have been defined prior to the calculation when designing the simulation.
- **Frequency** select the simulation frequency (as the carrier frequency) to visualize the results.
- **Period** select the simulation Doppler period (Only will be enabled if Doppler FMCW option is selected on Doppler parameters) to visualize the results.

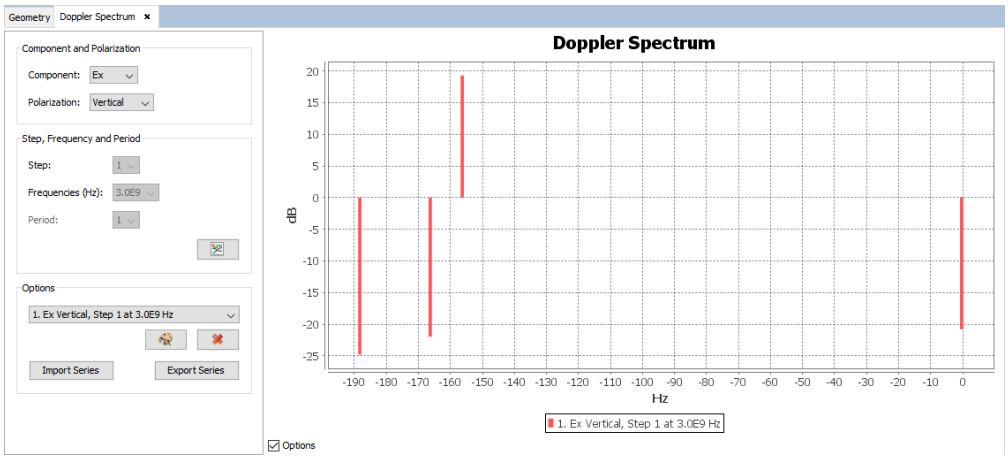


Figure 880: Doppler Spectrum panel

Text Files

This option shows the same file as in section View Ray on Ray Tracing results. First, a panel should appear after selecting this menu option to wait for step and period parameters to open the desired file.

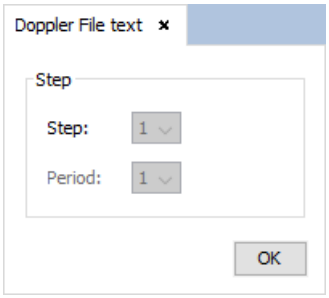


Figure 881: File Selection panel

To use this section, select a parametric step from the list and a Doppler period (only will be enabled if Doppler FMCW option is selected on Doppler parameters) to get the results from and click on OK to load the data of the present rays.

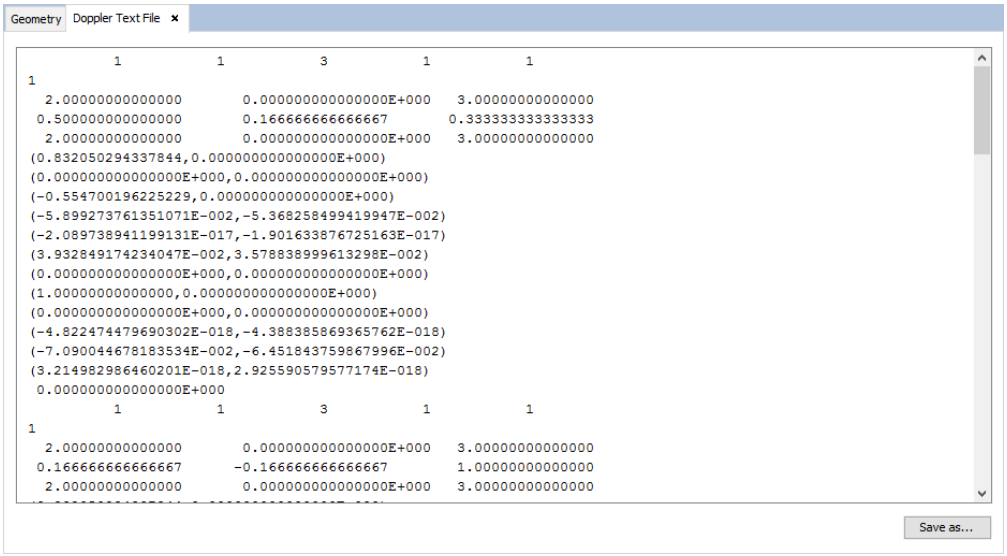


Figure 882: Ray File Panel

To generate a copy of the file click on ‘Save as…’ button and select the file path to save the copy.

FMCW

When the user selects a simulation of type Doppler and defines the Doppler parameters including FMCW, this option will be visible on the ‘Show Results --> Doppler’ option of the menu bar.

This option allows the user to visualize the spectrum of the field for one time or more times in the signal periods. Then, selecting a determinate instant of time, the user can see the Doppler Spectrum. The user can select one or more instants of time over all the periods simulated.

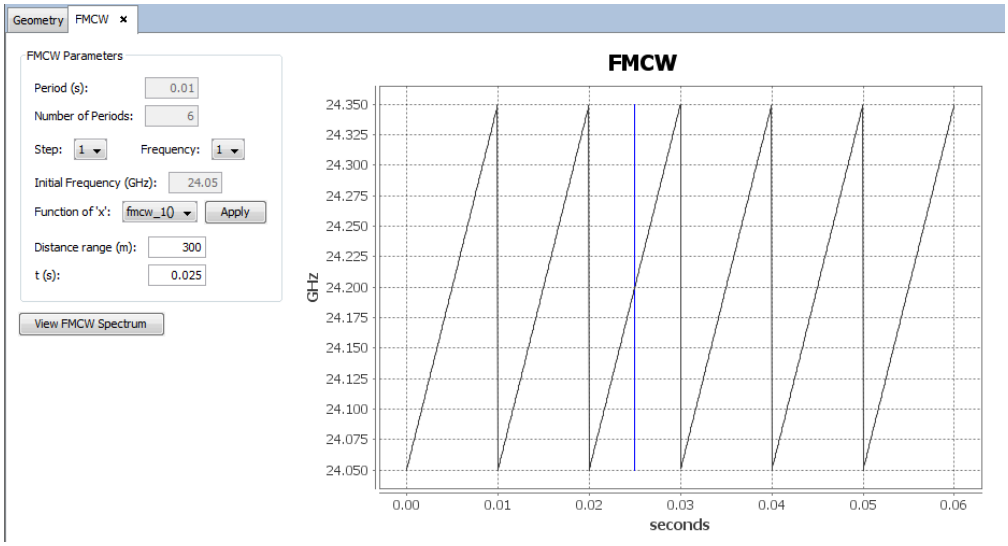


Figure 883: FMCW Parameters

- Period indicates, on seconds, the period duration on time. This parameters is configured in section Doppler.
- Number of periods indicates the number of repeats for the frequency variation. This parameters is configured in section Doppler.
- Step select the parametric scene to realize the analysis.
- Frequency select the carrier frequency for the analysis. This frequency will be one of the simulation frequencies.
- Initial Frequency indicates, on GHz, the frequency selected to be the carrier frequency.
- Function of 'x' assign the user function frequency signal.
- Distance Range indicates, on meters, the maximum distance considered in the simulation that rays can travel from transmitter to receptor including rebounds with obstacles.
- t(s) allows to add instant of time to analyze. Selecting an instant of time, in seconds, and clicking on the 'Add' button (green icon with the plus sign) the instant will be added to the analyzing list. Selecting a row from the list and clicking on the 'Delete' button (red 'X' icon) the selected instant will be removed from the analyzing list.

The frequency signal can be defined by a user function, where the x coordinate is the time in seconds. These functions are defined without parameters and return a double data type. These functions can use the $x variable, which is assigned the value of time in seconds and $period variable that equals the period value. The $x variable is always less or equal than $period.

The application includes by default five user functions for usage in FMCW. They are in the **functions** directory.

Sawtooth wave (fmcw_1.java):

```
doublefmcw_1(){
// This function returns the frequency increment from time '$x'
// Global parameters
// $period period time in seconds.
// $x time within the period.
```

```
//
// Note '$x' it is always less than '$period'
//
doubleMAX_BROADBAND=0.3;// GHz
return($x*MAX_BROADBAND)/$period;}
```

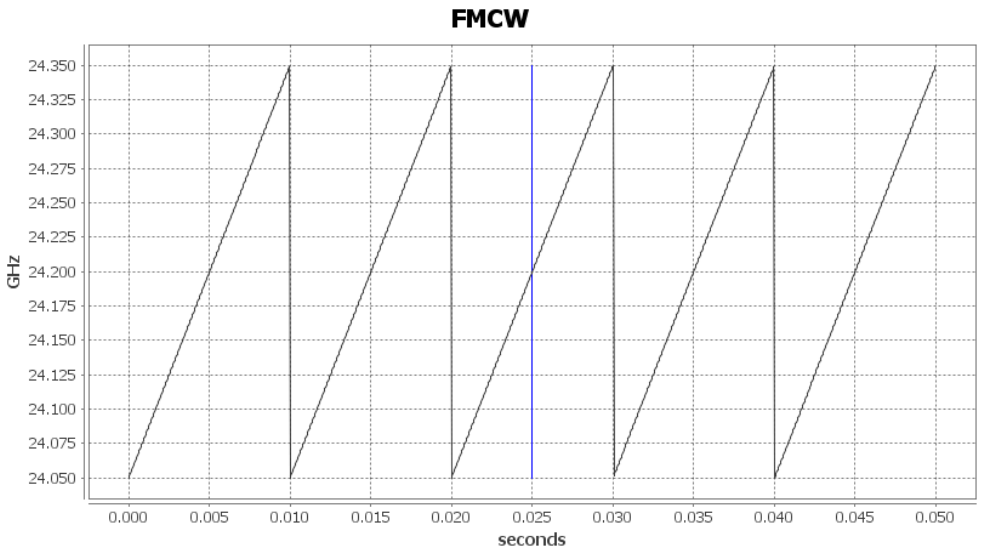


Figure 884: FMCW User function sawtooth wave

Square wave (fmcw_2.java):

```
doublefmcw_2(){
// This function returns the frequency increment from time '$x'
// Global parameters
// $period period time in seconds.
// $x time within the period.
//
// Note '$x' it is always less than '$period'
//
doublex1=1./3*$period;doublex2=2./3*$period;if($x<x1){return0.0;}elseif($x<x2)
{return0.1;}else{return0.2;}}
```

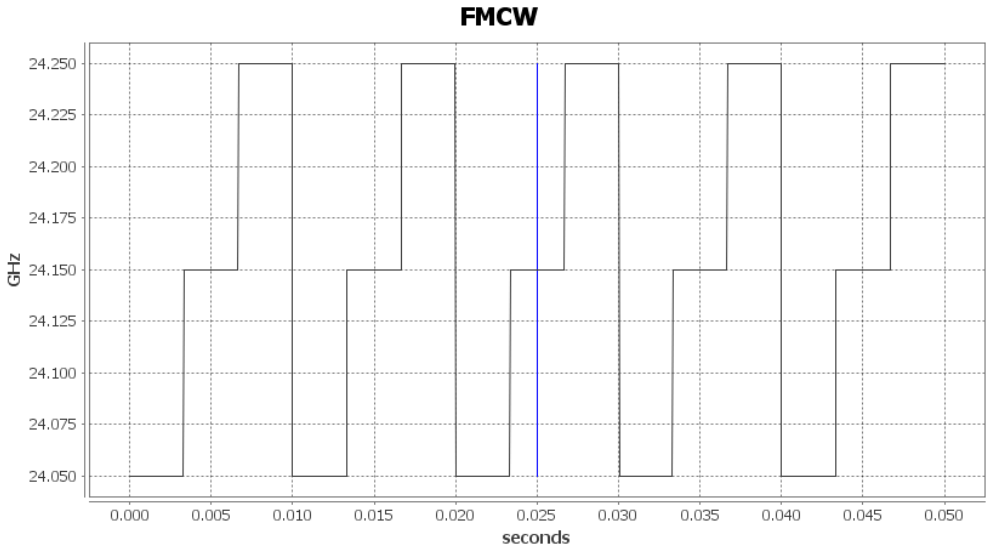


Figure 885: FMCW User function square wave

Range wave (fmcw_3.java):

```
doublefmcw_3(){
// This function returns the frequency increment from time '$x'
// Global parameters
// $period period time in seconds.
// $x time within the period.
//
// Note '$x' it is always less than '$period'
//
doublex1=1./3*$period;doublex2=2./3*$period;doublex3=
$period;doubley1=0.1;doubley2=0.2;if($x<x1){returny1/x1*$x;}elseif($x<x2)
{returny1;}else{returny1+((y2-y1)/(x3-x2))*($x-x2);}}
```

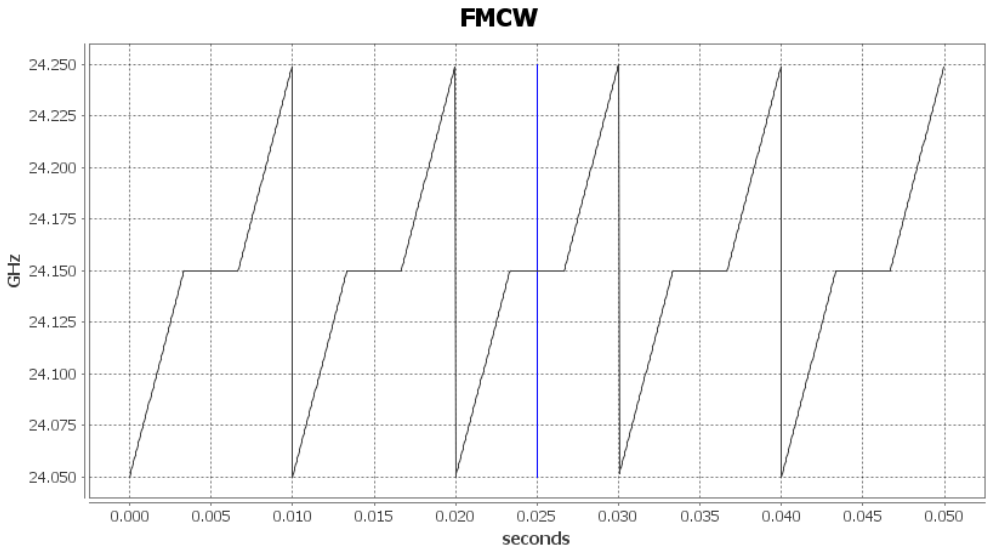


Figure 886: FMCW User function range wave

Triangular wave (fmcw_4.java):

```
doublefmcw_4(){
// This function returns the frequency increment from time '$x'
// Global parameters
// $period period time in seconds.
// $x time within the period.
//
// Note '$x' it is always less than '$period'
//
doubleBROADBAND=0.3;// first point
doublex0=0;doubley0=0;// second point
doublex1=1./2*$period;doubley1=BROADBAND;// third point
doublex2=$period;doubley2=0;if($x<x1){doublepending=(y1-y0)/
(x1-x0);doublexOffset=x0;doubleyOffset=y0;returnpending*($x-
xOffset)+yOffset;}else{doublepending=(y2-y1)/(x2-
x1);doublexOffset=x1;doubleyOffset=y1;returnpending*($x-xOffset)+yOffset;}}
```

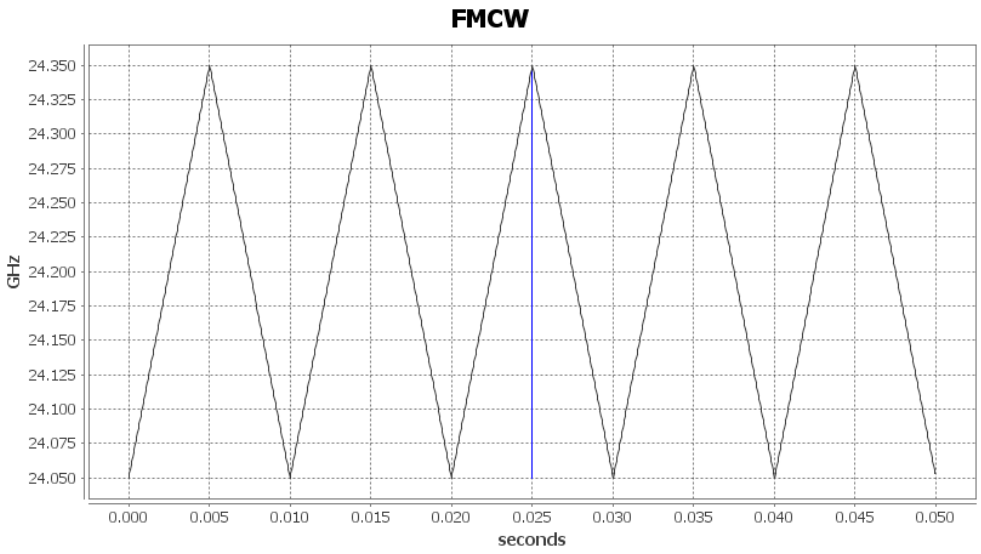


Figure 887: FMCW User function triangular wave

Sinusoidal wave (fmcw_5.java):

```
doublefmcw_5(){
// This function returns the frequency increment from time '$x'
// Global parameters
// $period period time in seconds.
// $x time within the period.
//
// Note '$x' it is always less than '$period'
//
doubleBROADBAND_GHZ=3.0e-3;// change [0, $period] range to [0, 2*PI] range
doubleradians=($x*(Math.PI*2))/$period;returnMath.sin(radians)*BROADBAND_GHZ;}
```

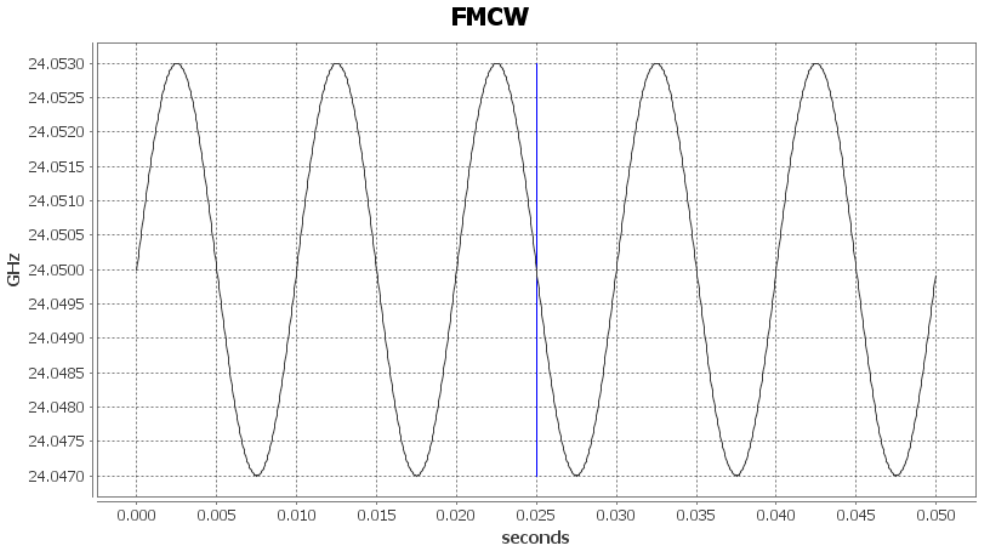


Figure 888: FMCW User function sinusoidal wave

See further information in User Functions.

When all the previous parameters will be defined, clicking on "View FMCW Spectrum" button, the spectrum chart appears.

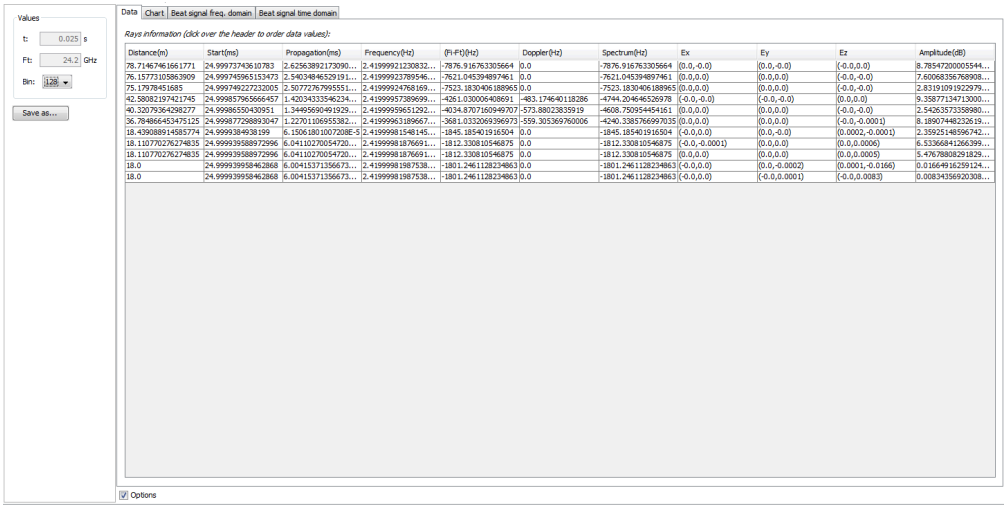


Figure 889: FMCW Spectrum, Data Values

To save the data table values click on the ‘Save as...’ button and select the path for the file.

This chart indicates, for each frequency on the broadband, the contribution of the rays emitted in the frequency and came to the receptor at time ‘t’. The frequency values on X-axis will be indicated referenced to the current frequency (‘Ft’) on the analyzing instant, then a contribution of -2000 Hz means a contribution of a rays emitted to 2000 Hz below of the current frequency.

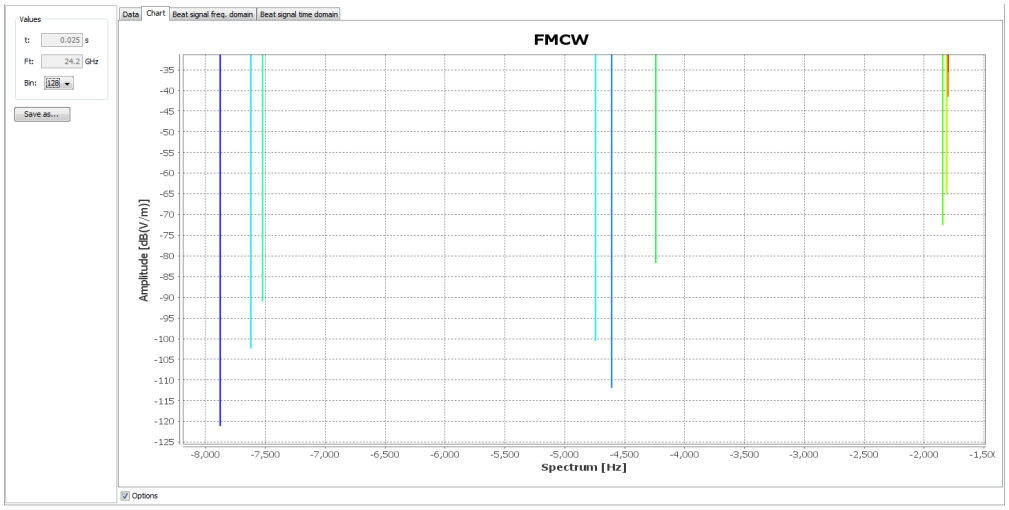


Figure 890: FMCW Spectrum, Chart

- t: Indicates the instant of time that 'Time' parameter referenced.
- Ft: indicates the current frequency. This frequency is the frequency on the instant of time 't'.
- Bin: divides the spectrum and modifies the **beat signal frequency domain** and the **beat signal time domain** charts.

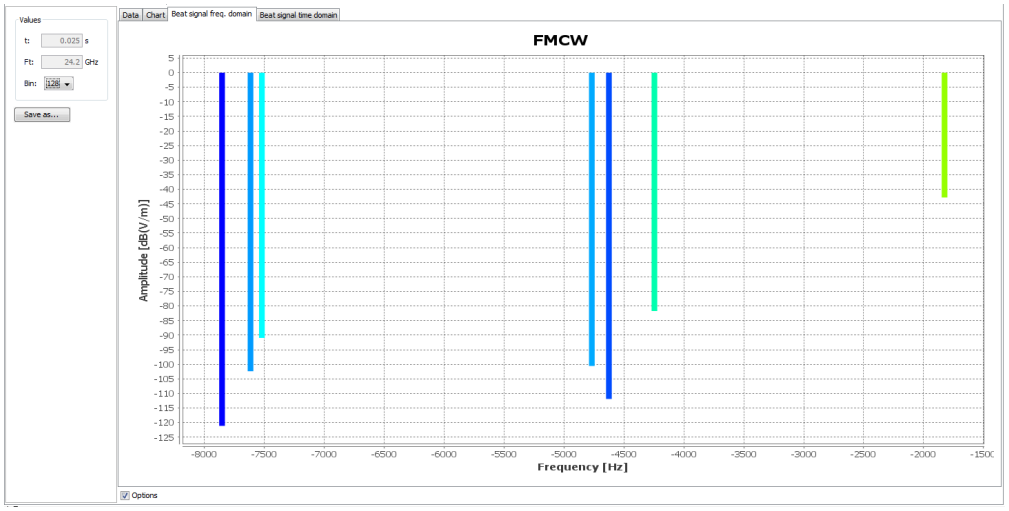


Figure 891: FMCW Spectrum, Beat signal frequency domain

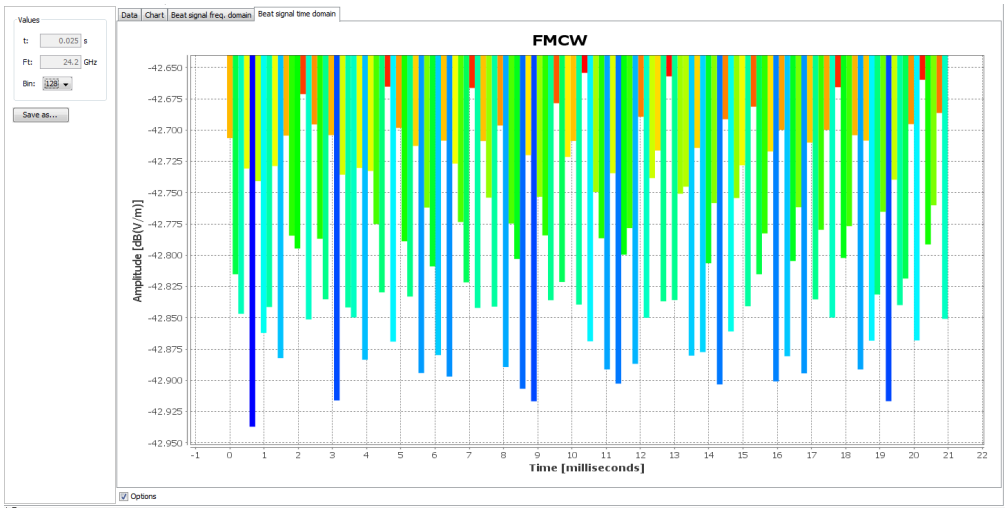


Figure 892: FMCW Spectrum, Beat signal time domain

9.13 Tools Menu

For information about the Tools menu, refer to Tools.

9.14 Help Menu

For information about the Help menu, refer to Help Menu.

9.15 Training Examples

9.15.1 Example 1: Monostatic RCS of a Sphere and a Box

In this example, the Monostatic RCS of a box and a sphere is calculated.

Step 1

Start newFASANT.

Step 2

Select File and click on New.

Step 3

Select GTD-PO.

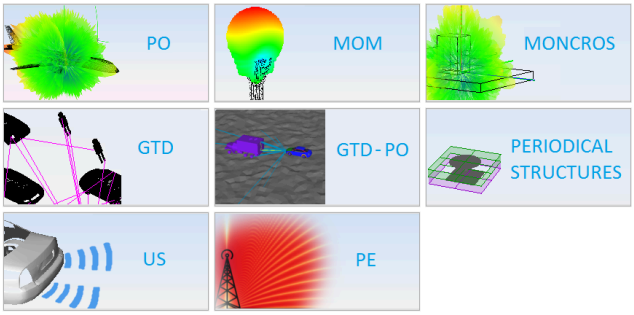


Figure 893: Method Type selection

Step 4

Click on Geometry→Solid→Sphere, which requires the center and the radius, as shown in the next Figure.

In this example the user enters the following values into the command line:

- Center: 0 0 0
- Radius: 1.0

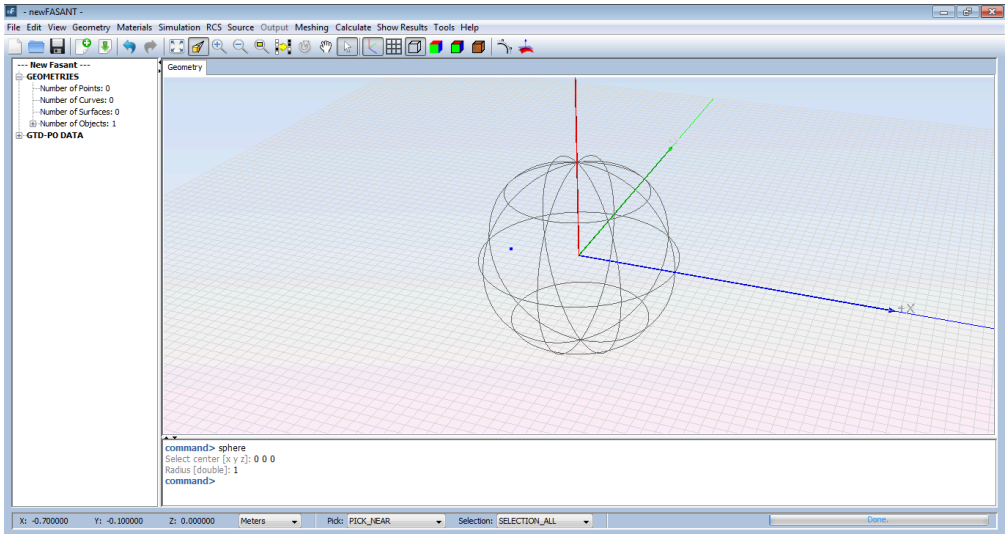


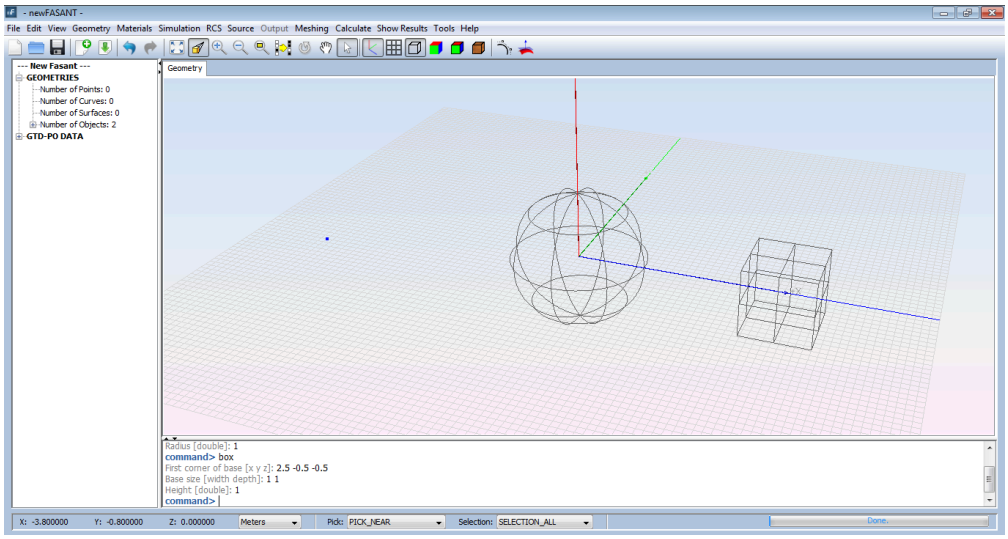
Figure 894: Create sphere command

Step 5

Click on Geometry → Solid → Box, which requires the first corner of the base, width, depth, and height as shown in the next Figure.

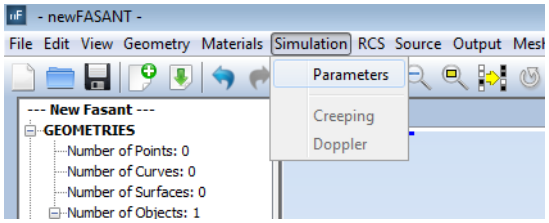
In this example the user enters the following values into the command line:

- First corner base: 2.5 -0.5 -0.5
- Width: 1.0
- Depth: 1.0
- Height: 1.0



Step 6

Click on Simulation → Parameters.



Step 7

On the top left side of the newFASANT window, the user can enter the desired parameters for this simulation, as shown in next Figure.

Select 1 bounce (simple reflection) and a frequency of 3.0 GHz and left-click the Save button.

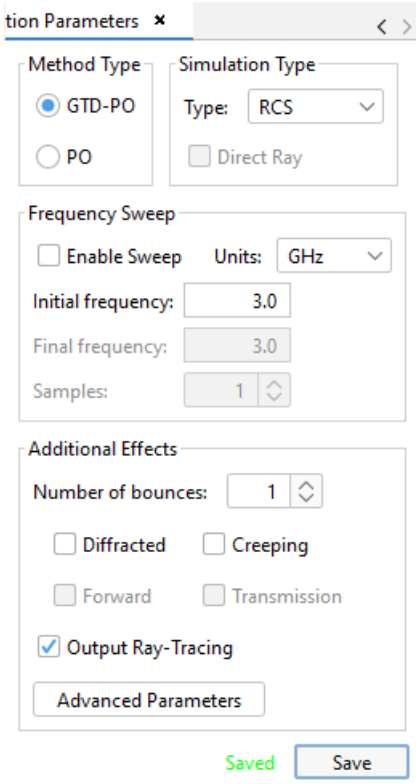


Figure 895: Simulation Parameters panel

Step 8

Select RCS → Parameters.

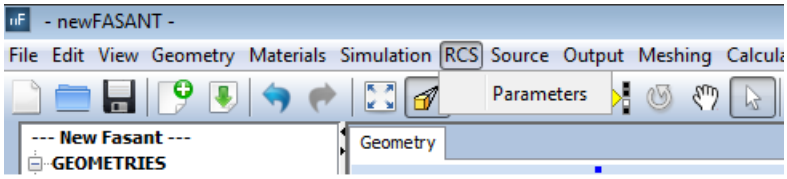


Figure 896: RCS menu

Step 9

This command appears at the top left side of the newFASANT window as shown in the next Figure.

Select Monostatic RCS only.

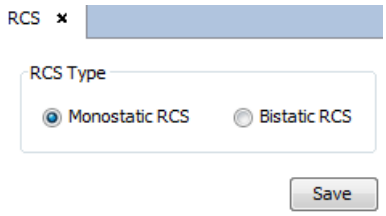


Figure 897: RCS panel

Step 10

Select Source→ Parameters.

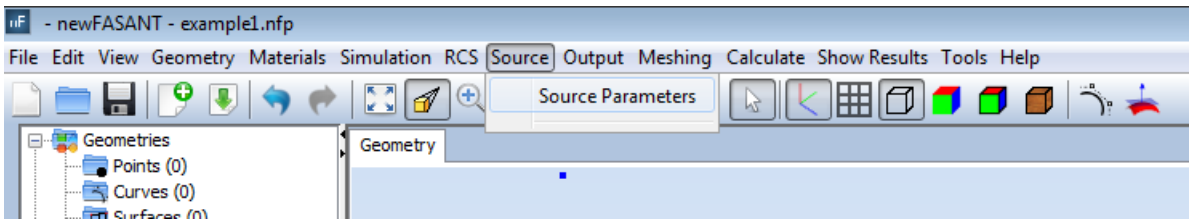


Figure 898: Source menu

Step 11

Introduce the source position (r, theta, and phi), as shown in the next Figure and left-click on the Save button.

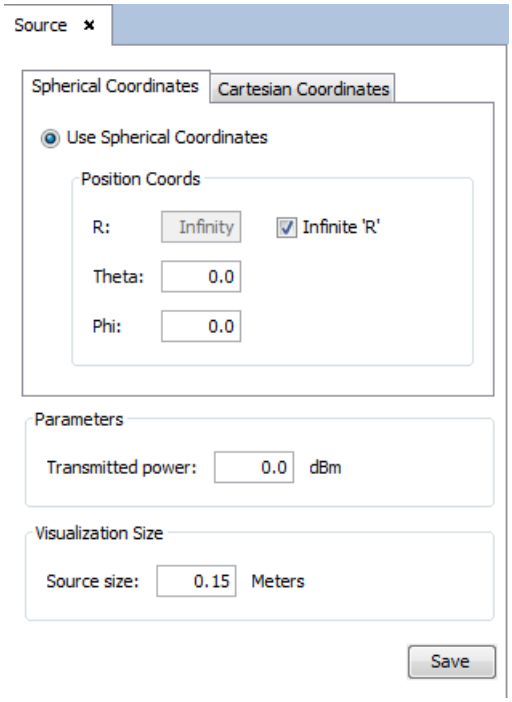


Figure 899: Source panel

Step 12

In the Output tab, introduce the observation information sweep on theta with 181 samples.

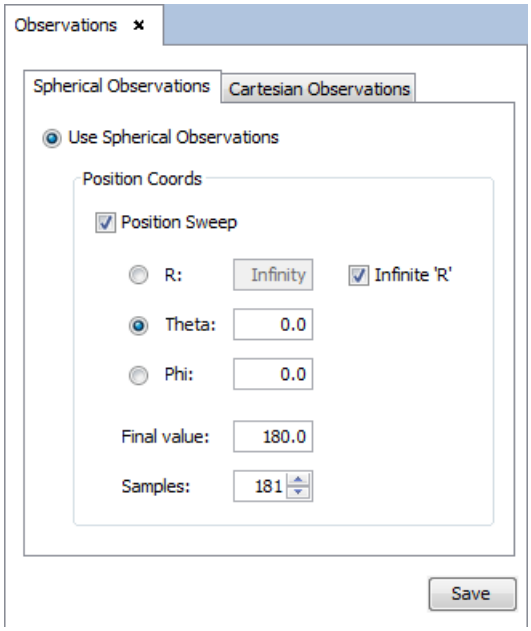


Figure 900: Observation directions panel

Step 13

Before running the case, select Meshing → Create Mesh.

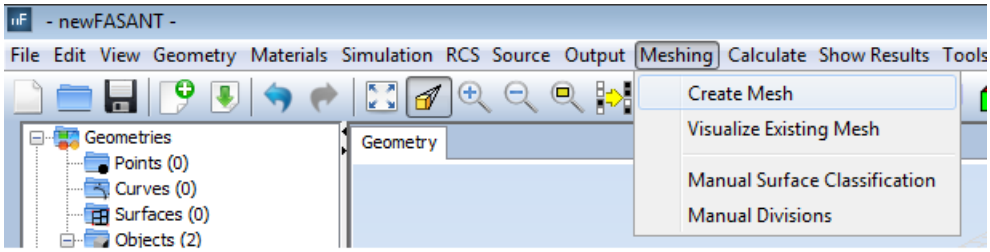


Figure 901: Meshing menu

Step 14

Select 1 processor and define the curvature mesh option with a distance error of 0.01 and surface error of 0.08 and left-click on Mesh.

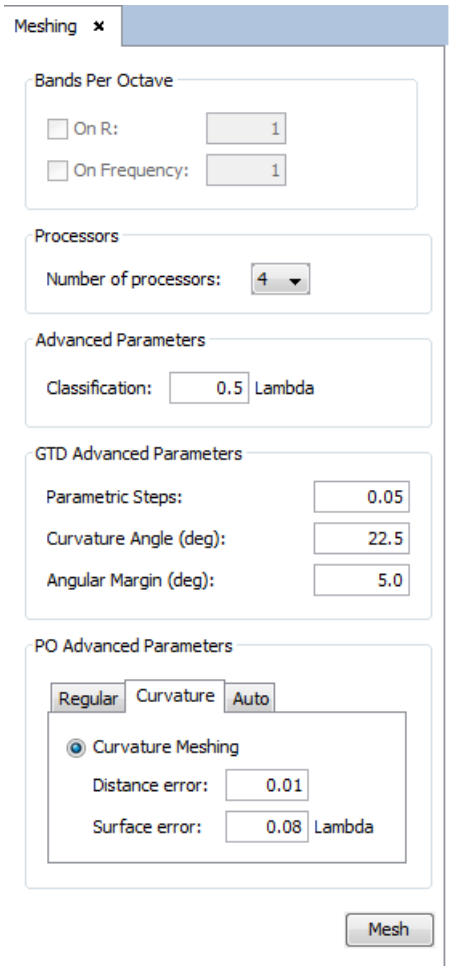
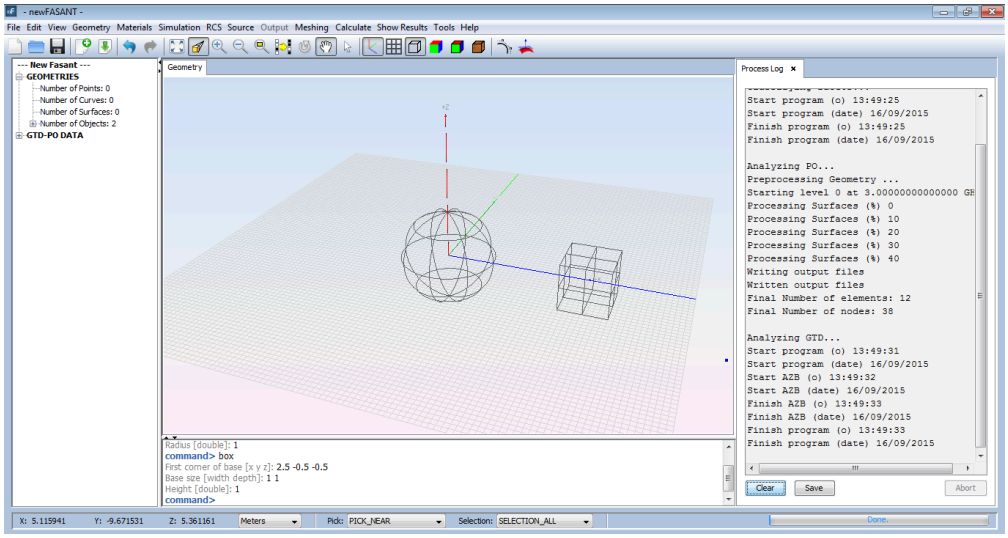


Figure 902: Meshing panel

Left-clicking on Mesh launches the meshing engine as shown in the next Figure:



Step 15

To obtain information about the mesh, select Meshing →Visualize Existing Mesh.

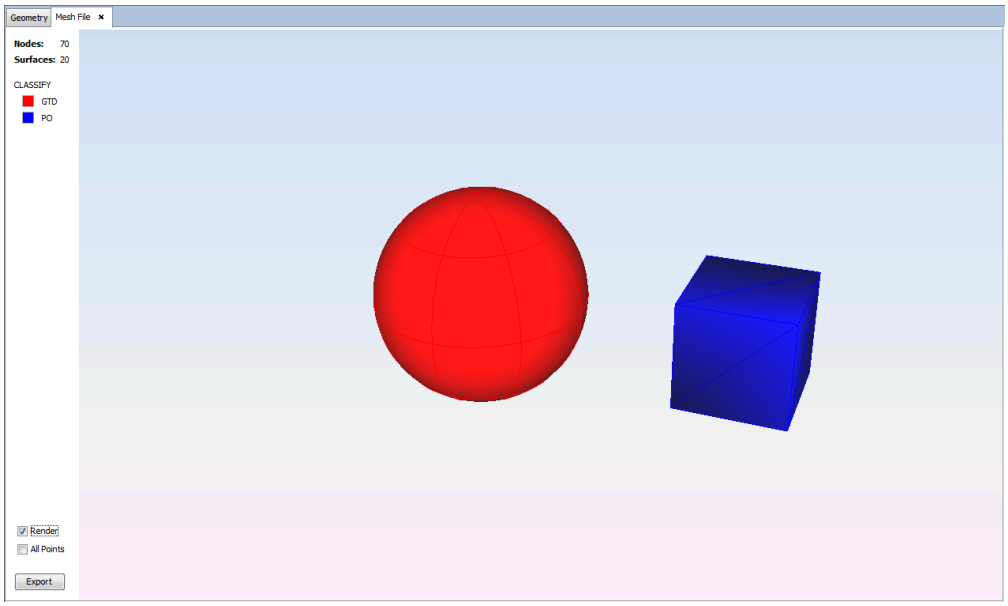


Figure 903: Meshing visualization

Step 16

Select Calculate → Execute and select the number of processors available to simulate this case.

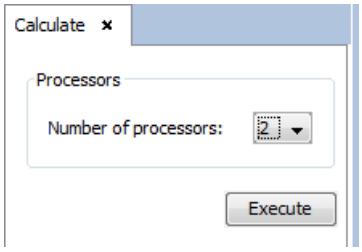


Figure 904: Calculate panel

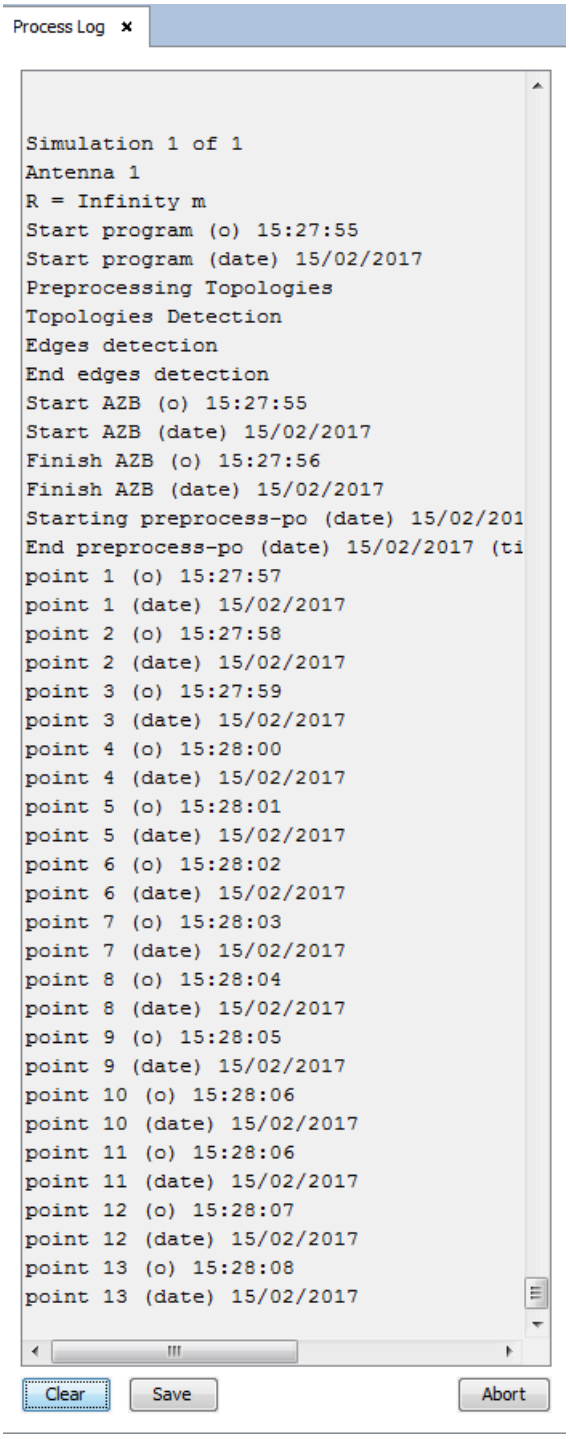


Figure 905: Process Log panel

Step 17

When the simulation finishes we can visualize the simulation results. Click on Show Results →RCS→ View Cuts, which allows the user to see the RCS graphic.

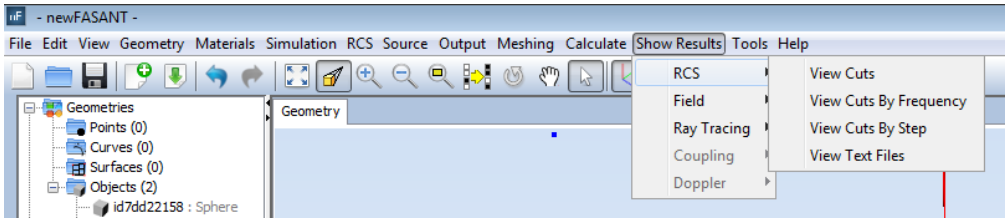


Figure 906: Show Results menu

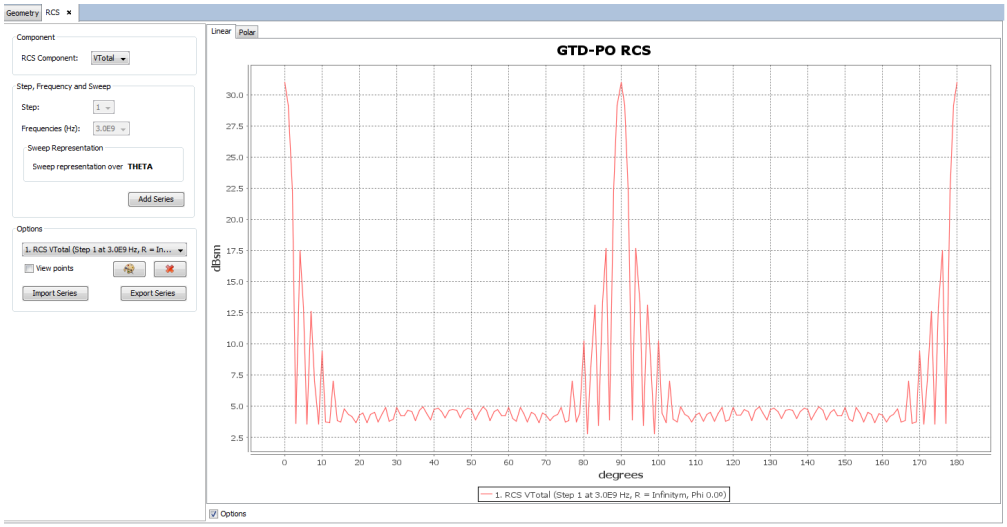


Figure 907: RCS visualization

Step 18

Click on Show Results → RCS → View Text Files. Then select the Steps and press OK to see the RCS data file.

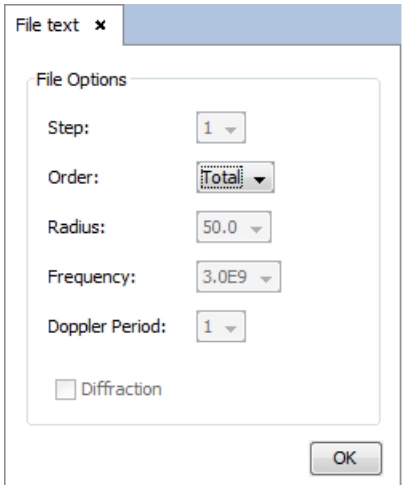
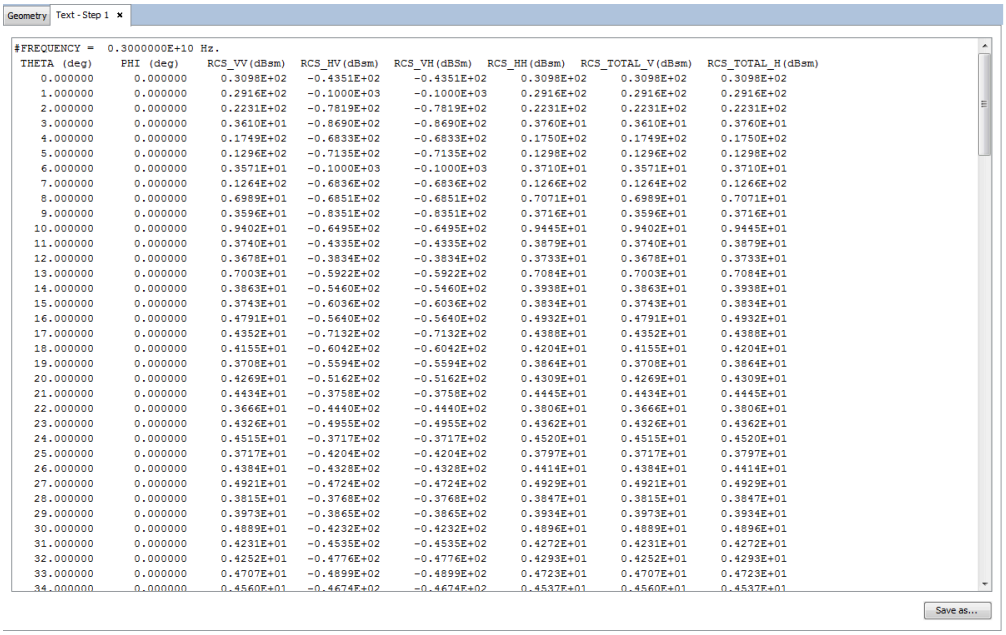


Figure 908: File text panel



Step 19

Click on Show Results → Ray Tracing → View Ray. It is possible to represent the ray tracing of the sources interacting with the geometry

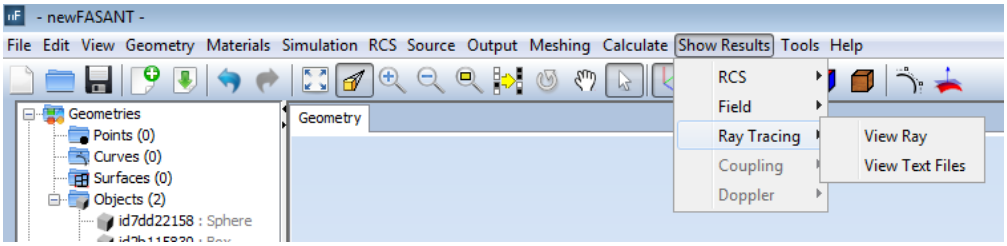


Figure 909: Show Results menu

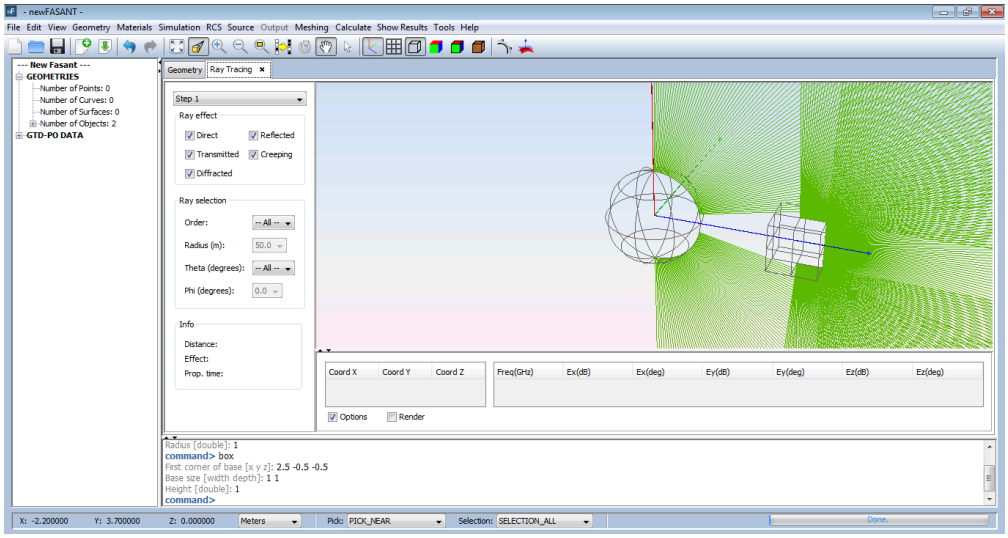


Figure 910: Ray-tracing visualization

9.15.2 Example 2: Monostatic RCS of a Ship

This case explains how to calculate the monostatic RCS of an imported geometry.

Step 1

Start newFASANT.

Step 2

Select File and click on New.

Step 3

Select GTD-PO.

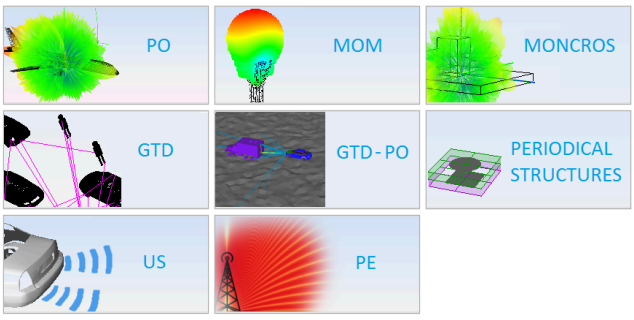


Figure 911: Method Type selection

Step 4

Select the geometry to import.

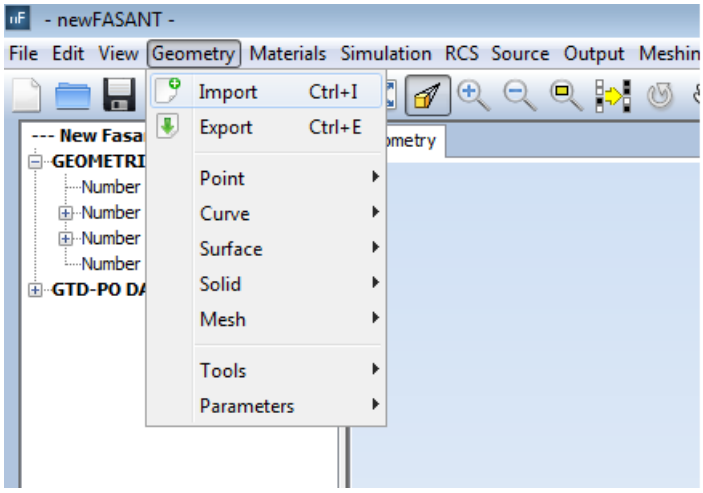


Figure 912: Import geometry menu

Step 5

Click on Simulation -> Parameters to change the project parameters.

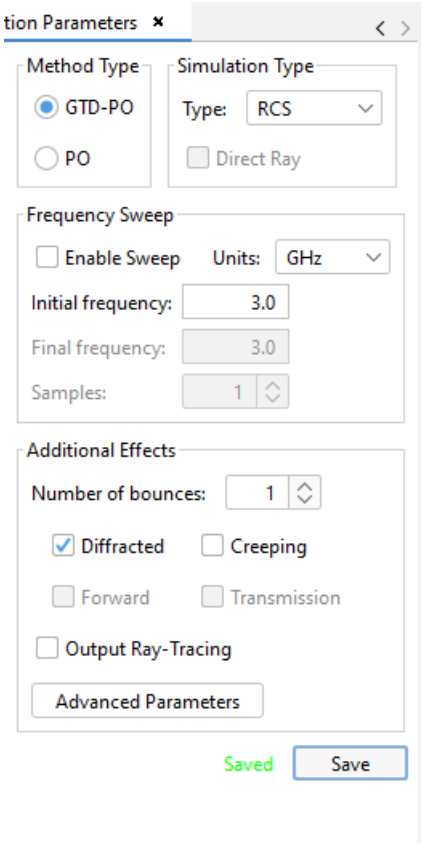


Figure 913: Simulation parameters window

Step 6

Click con RCS -> Parameters to set de RCS parameters of the project.

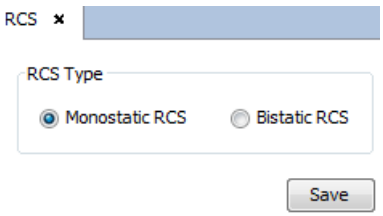


Figure 914: RCS parameters window

Step 7

Click on Source -> Parameters to modify the source parameters for this project.

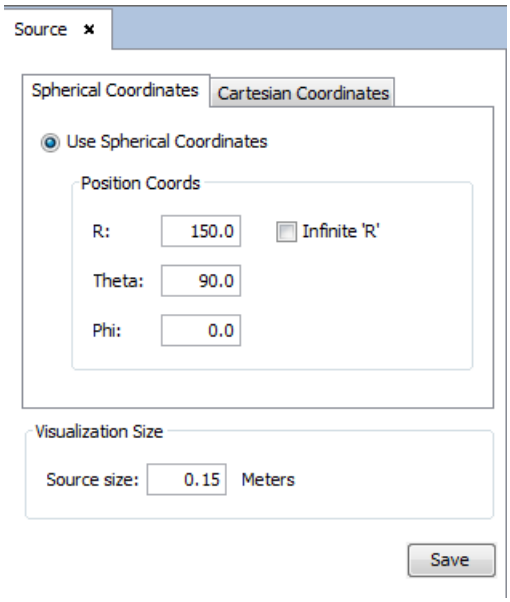
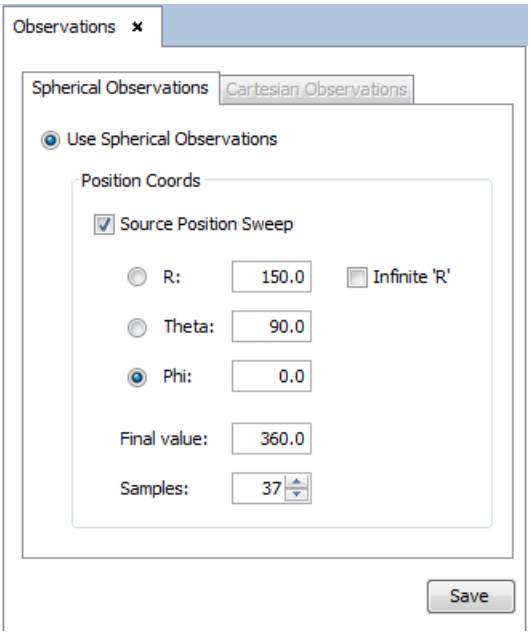


Figure 915: Source parameters panel

Step 8

Introduce the observation information as follows:



Step 9

Before running this example, create the project mesh by doing click on Meshing -> Create Mesh.

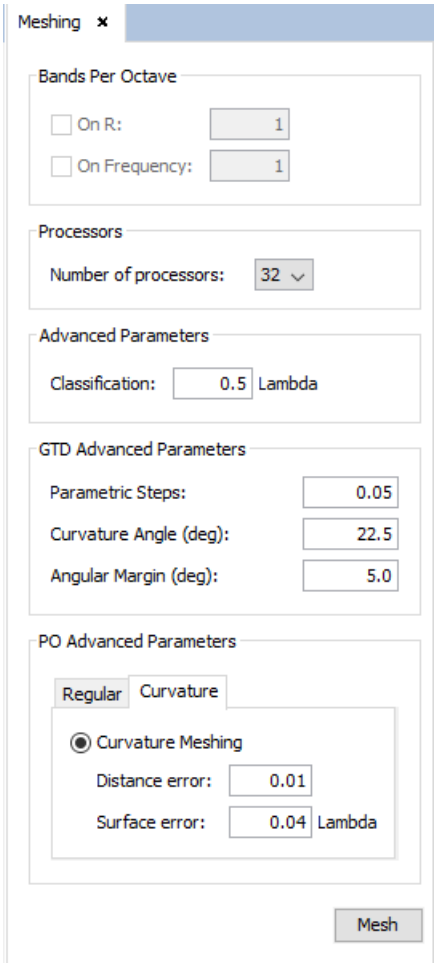


Figure 916: Meshing panel

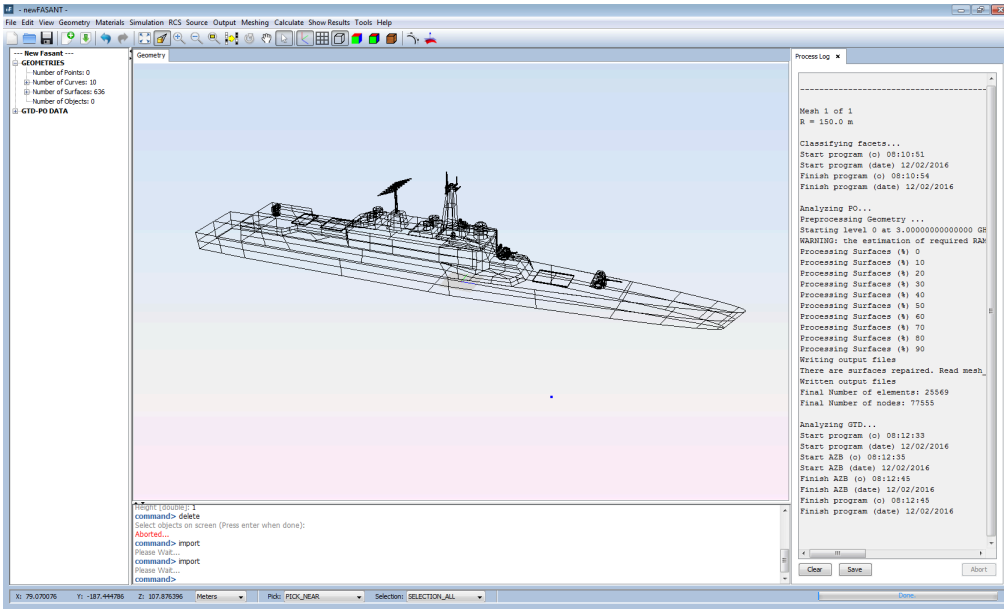


Figure 917: Meshing execution log

Step 10

Visualize the created mesh. To do this click on Meshing -> Visualize mesh.

Now you will be able to choose the generated mesh file.

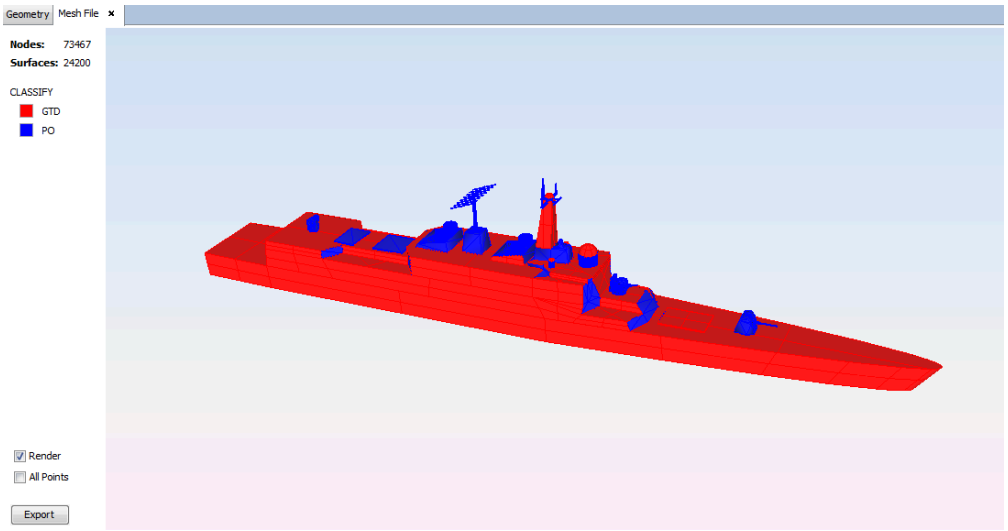


Figure 918: Visualize generated mesh

Step 11

Now we are going to proceed to simulate the project.

Click on Calculate -> Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.

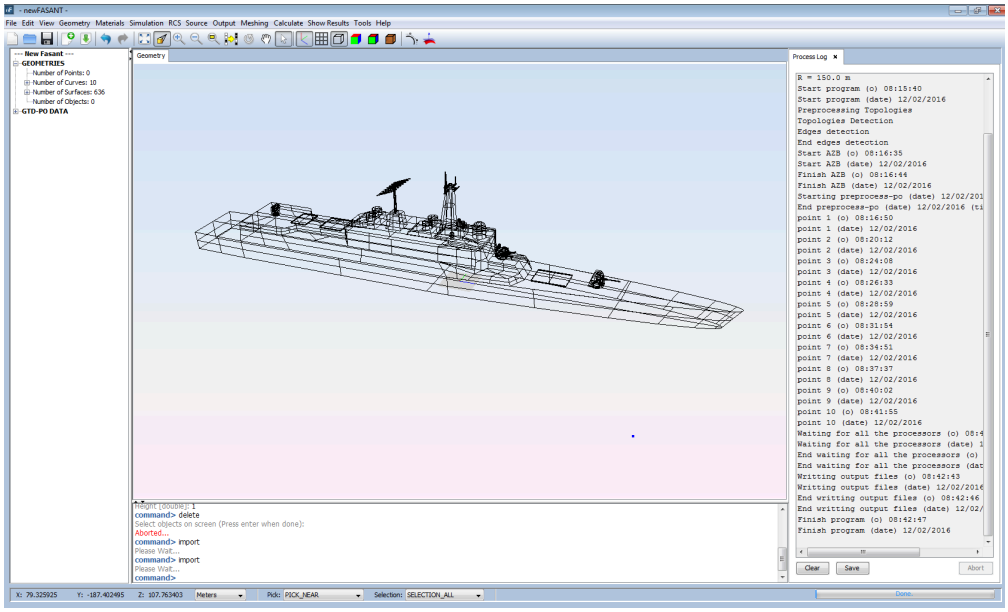


Figure 919: Simulation log

Step 12

Now you can see the results of the simulation by doing click on Show Results and selecting the desired results.

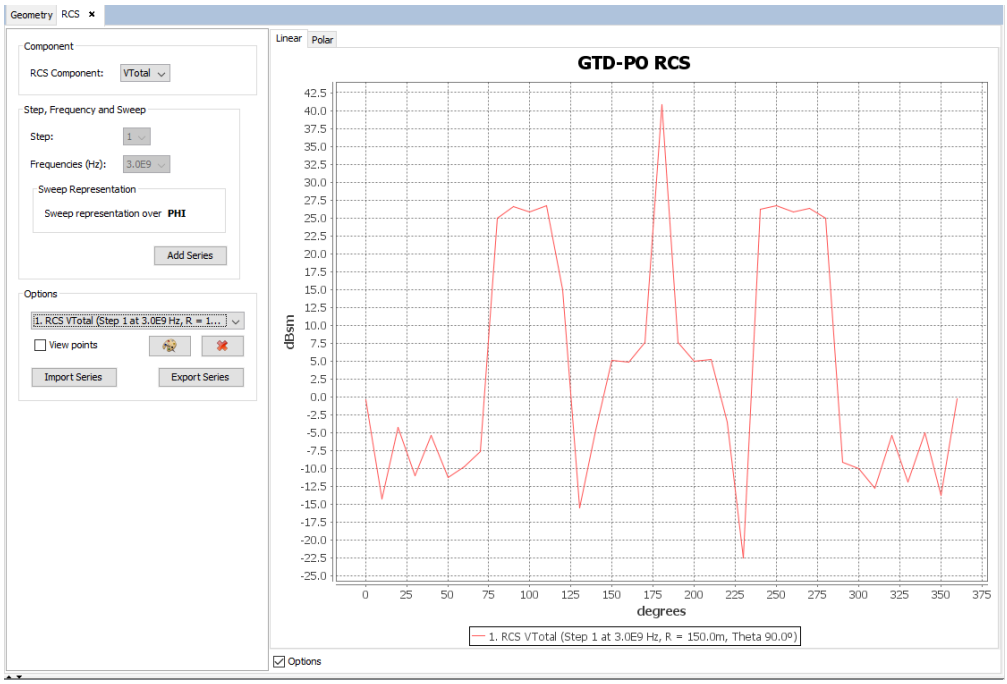


Figure 920: View RCS cut result

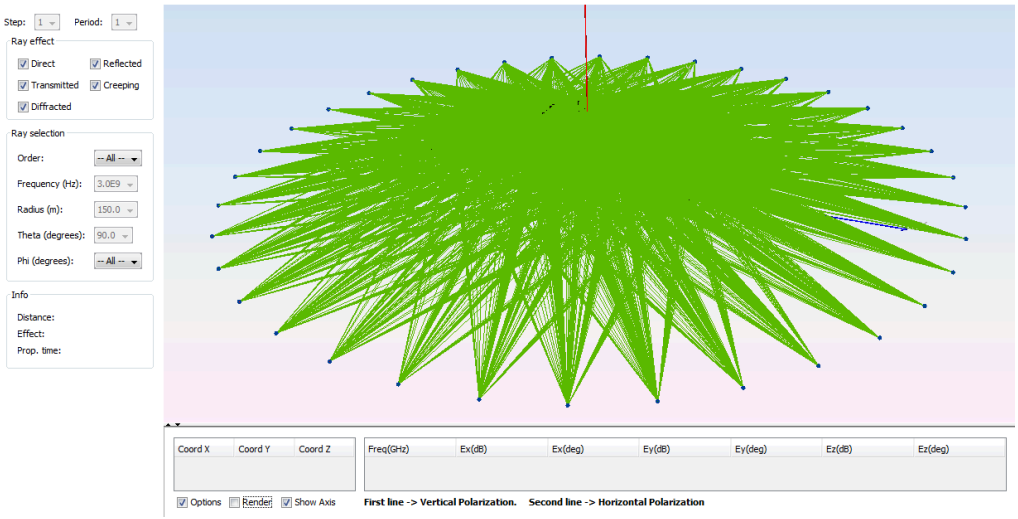


Figure 921: Ray tracing results

9.15.3 Example 3: Bistatic RCS of a Box

This case explains how to compute the bistatic RCS of a box.

Step 1

Start newFASANT.

Step 2

Select File and click on New.

Step 3

Select GTD-PO.

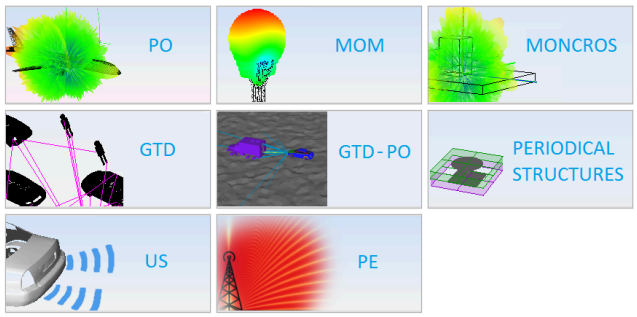


Figure 922: Method Type selection

Step 4

Define the geometry of a box.

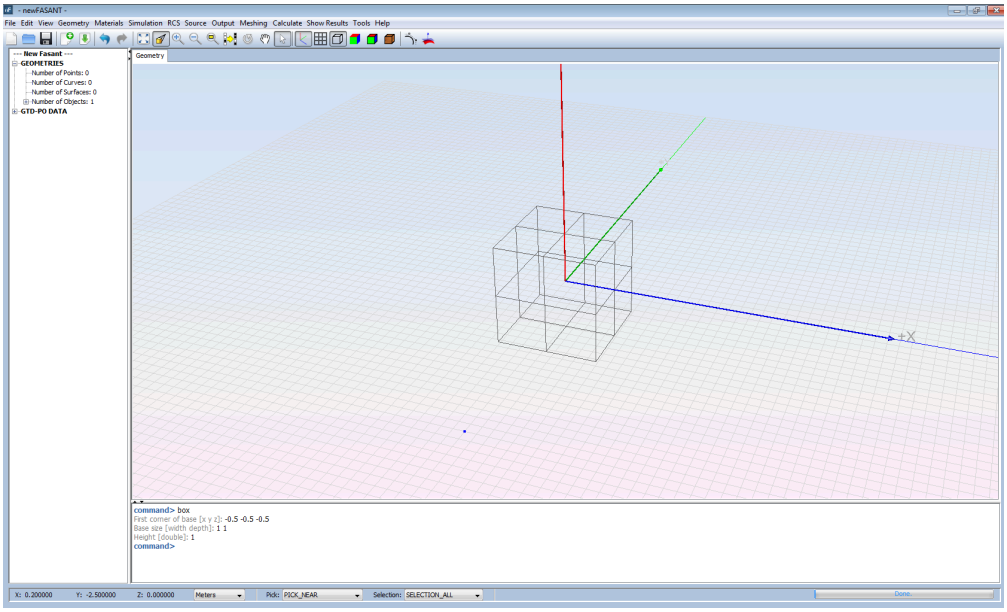


Figure 923: Geometry of a box

Step 5

Click on Simulation -> Parameters to change the project parameters.

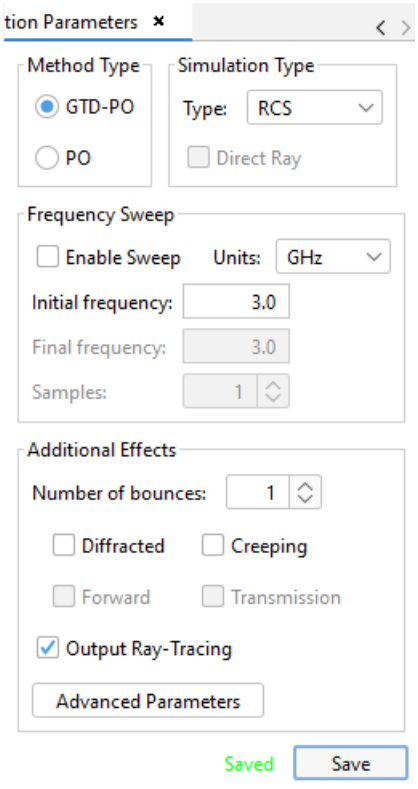


Figure 924: Simulation parameters window

Step 6

Click on RCS -> Parameters to set de RCS parameters of the project.

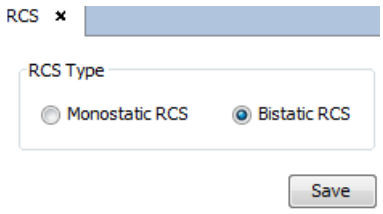


Figure 925: RCS parameters window

Step 7

Click on Source -> Parameters to modify the source parameters for this project.

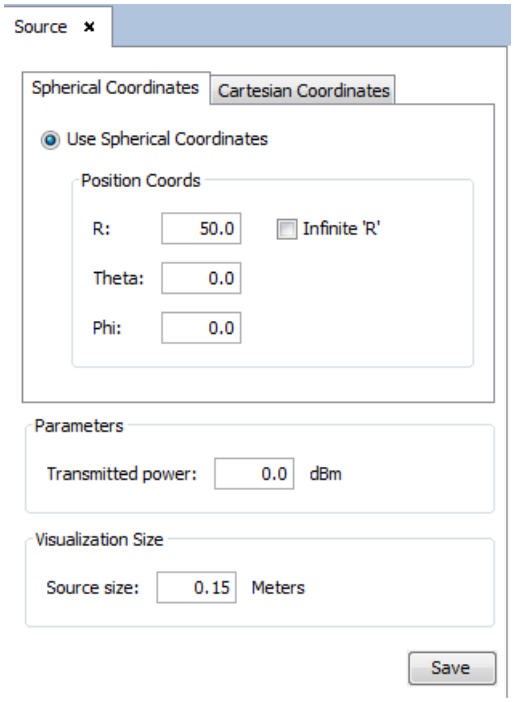


Figure 926: Source parameters panel

Step 8

Click on Output -> Observation directions.

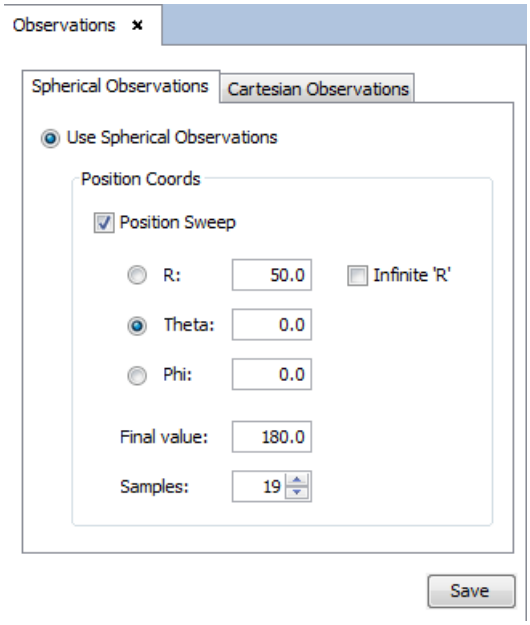


Figure 927: Observation directions panel

Step 9

Before running this example, create the project mesh by doing click on Meshing -> Create Mesh.

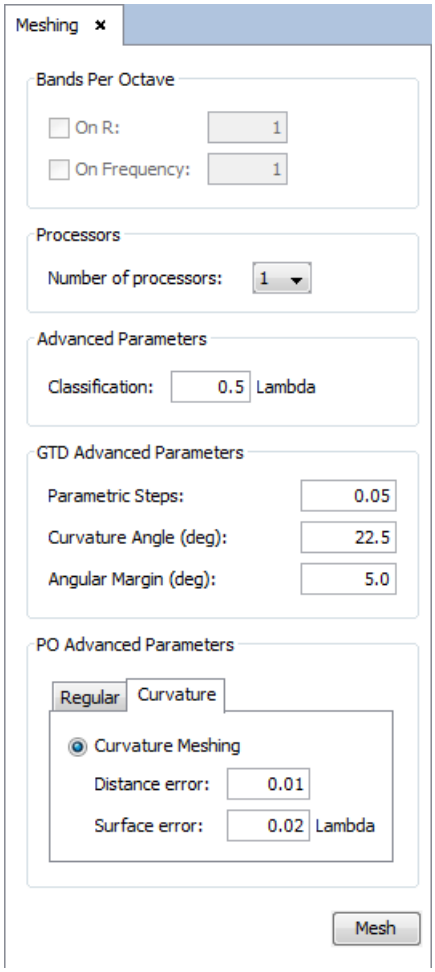


Figure 928: Meshing panel

Step 10

Now we are going to proceed to simulate the project.

Click on Calculate -> Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.

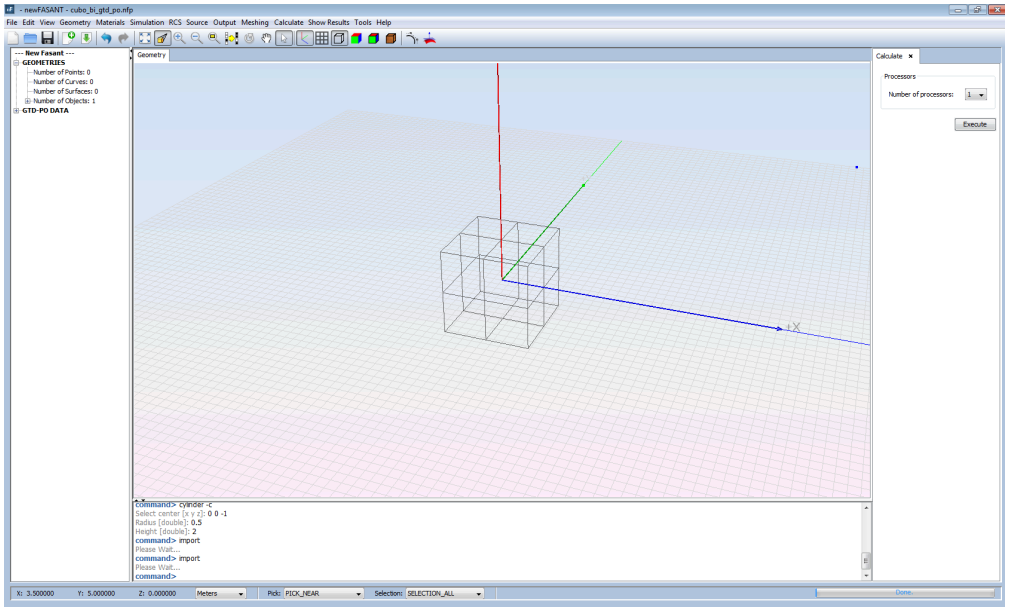


Figure 929: Calculate window

Step 11

Now you can see the results of the simulation by doing click on Show Results and selecting the desired results.

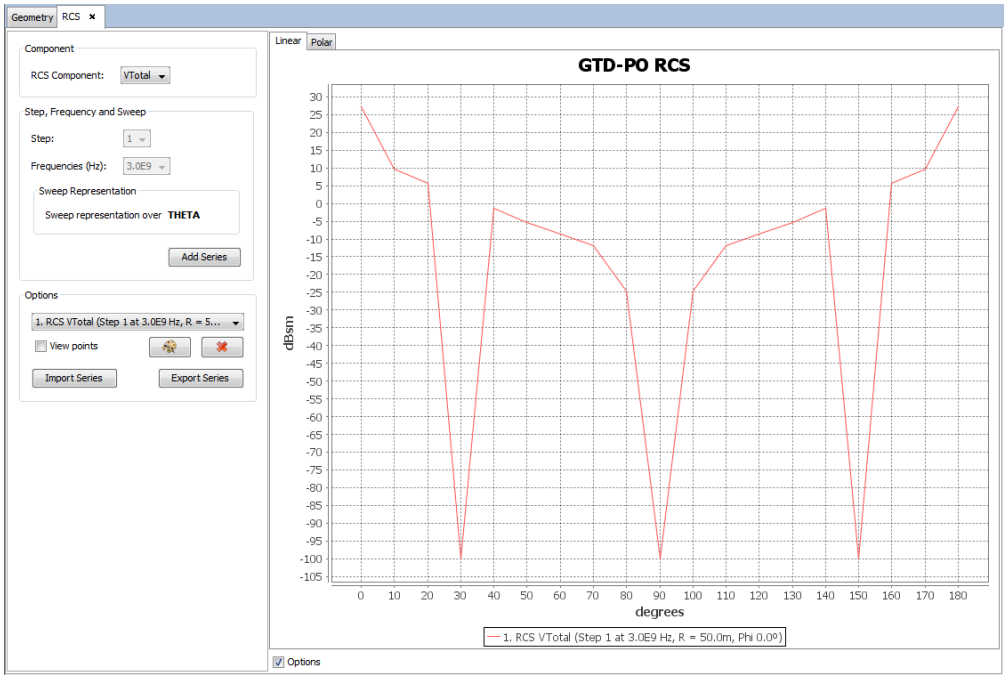


Figure 930: RCS results

9.15.4 Example 4: Coupling

This case explains how to calculate the coupling between two antennas.

Step 1

Start newFASANT and select GTD-PO.

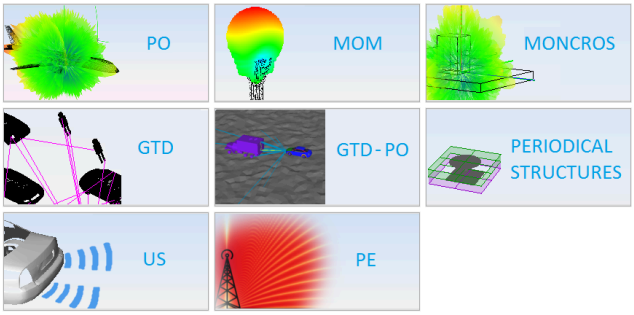


Figure 931: Method type selection.

Step 2

Define the geometry of an ellipsoid

```
command> ellipsoid
Select center [x y z]: 0 0 0
X radius [double]: 1
Y radius [double]: 0.5
Z radius [double]: 0.5
```

Figure 932: Ellipsoid command.

Step 3

Click on Simulation -> Parameters to change the project parameters

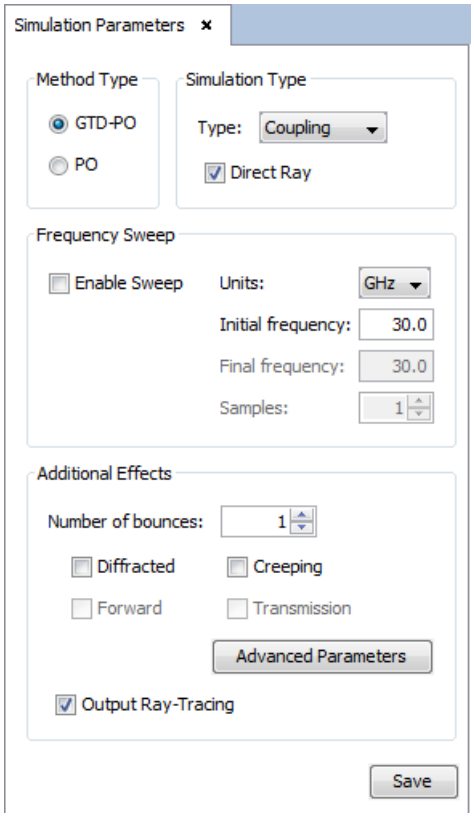


Figure 933: Simulation parameters panel.

Step 4

Click on Source -> Parameters to modify the source parameters for this project

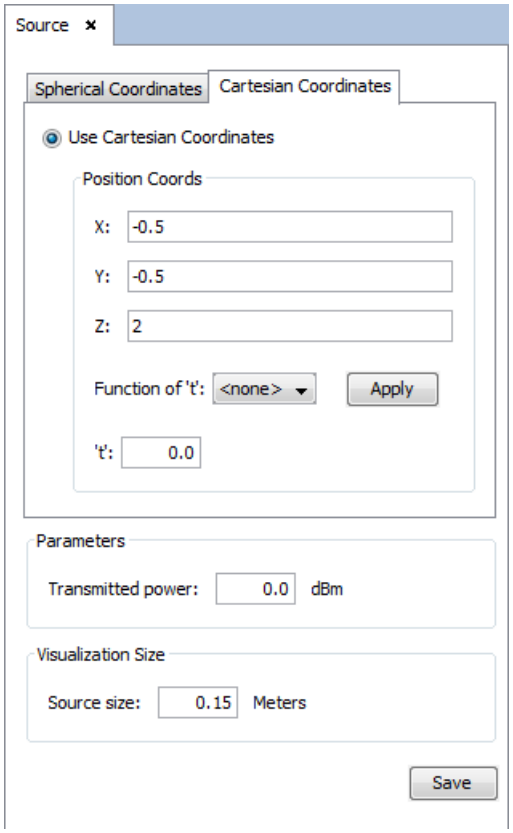


Figure 934: Source parameters panel.

Step 5

Click on Output -> Coupling to define the parameters of the passive source

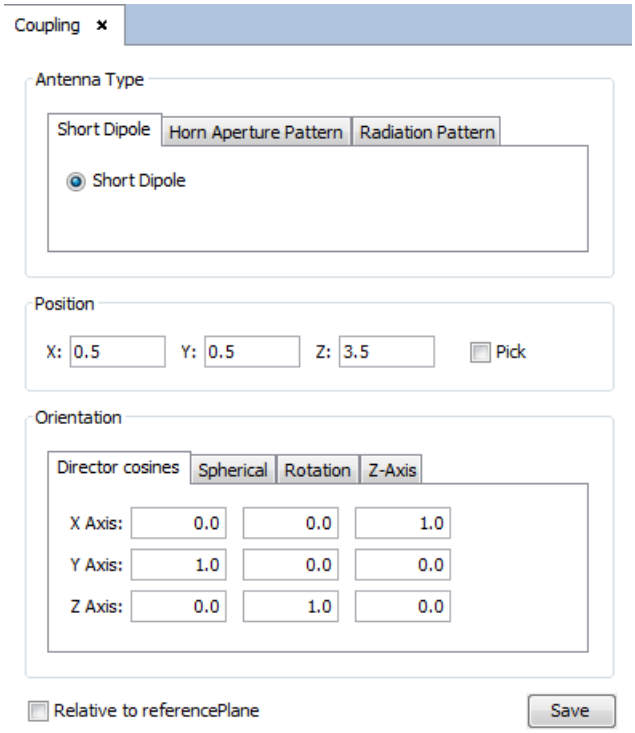


Figure 935: Coupling parameters panel.

Step 6

Before running this example, create the project mesh by doing click on Meshing -> Create Mesh

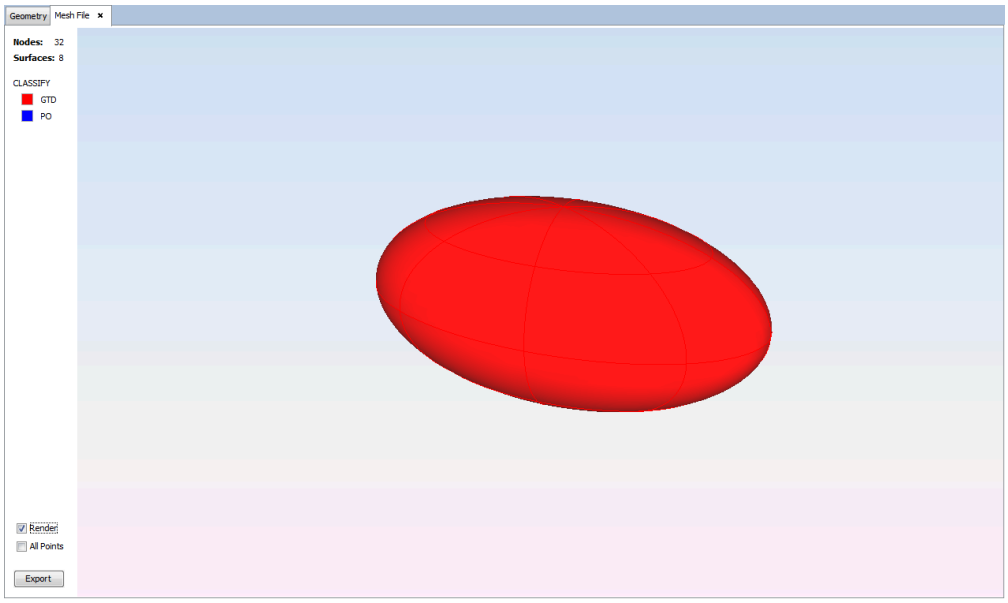


Figure 936: Mesh result.

Step 7

Now we are going to proceed to simulate the project.

Click on Calculate -> Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.

Step 8

Now you can see the results of the simulation by doing click on Show Results -> Coupling -> Source Coupling.

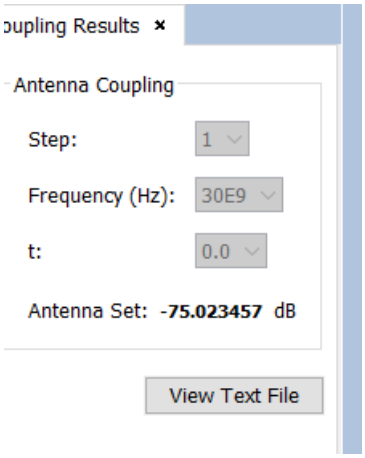


Figure 937: Coupling results.

9.15.5 Example 5: Doppler Spectrum

In this example, the Doppler spectrum of two boxes moving along the x-axis has been analyzed.

Step 1

Start newFASANT and select GTD-PO.

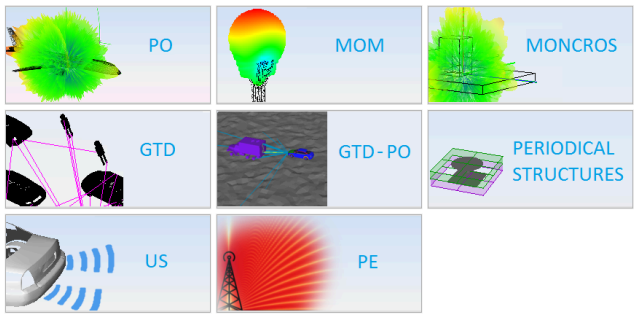


Figure 938: Method type selection.

Step 2

Define the geometry of two boxes.

```
command> box
First corner of base [x y z]: -0.5 -0.5 -0.5
Base size [width depth]: 1 1
Height [double]: 1
command> box
First corner of base [x y z]: -1.5 -0.5 0.5
Base size [width depth]: 1 1
Height [double]: 1
command>
```

Figure 939: Geometry command.

Step 3

Click on Simulation -> Parameters to change the project parameters

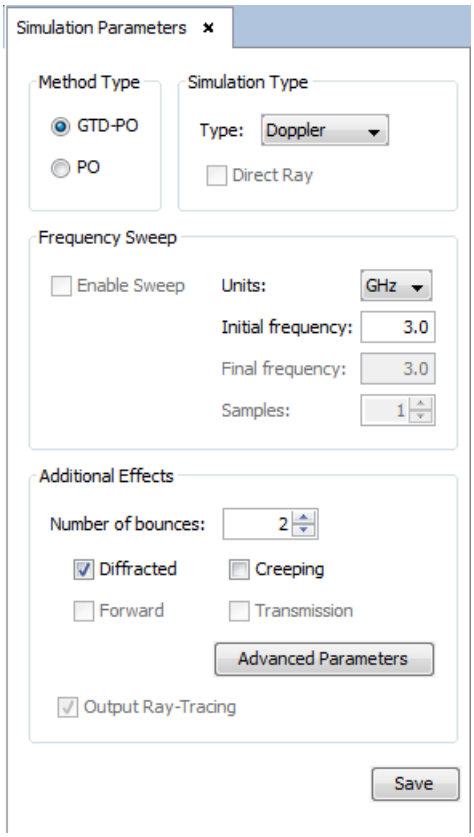


Figure 940: Simulation parameters panel.

Step 4

Now Doppler option in the Simulation menu is available. Click on Simulation -> Doppler. We are going to add doppler effects to the objects in the geometry. Select one box and press the Add button. The selected object is added to the list. Introduce the linear velocity of this object:

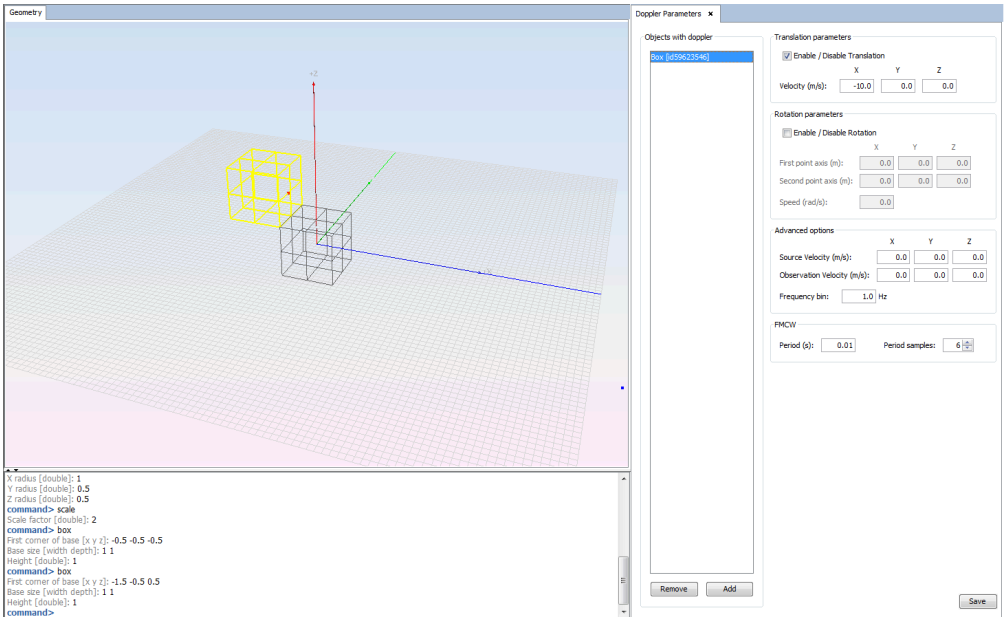


Figure 941: Doppler parameters of the first box.

Select the other box and press Add button. Introduce its linear velocity.

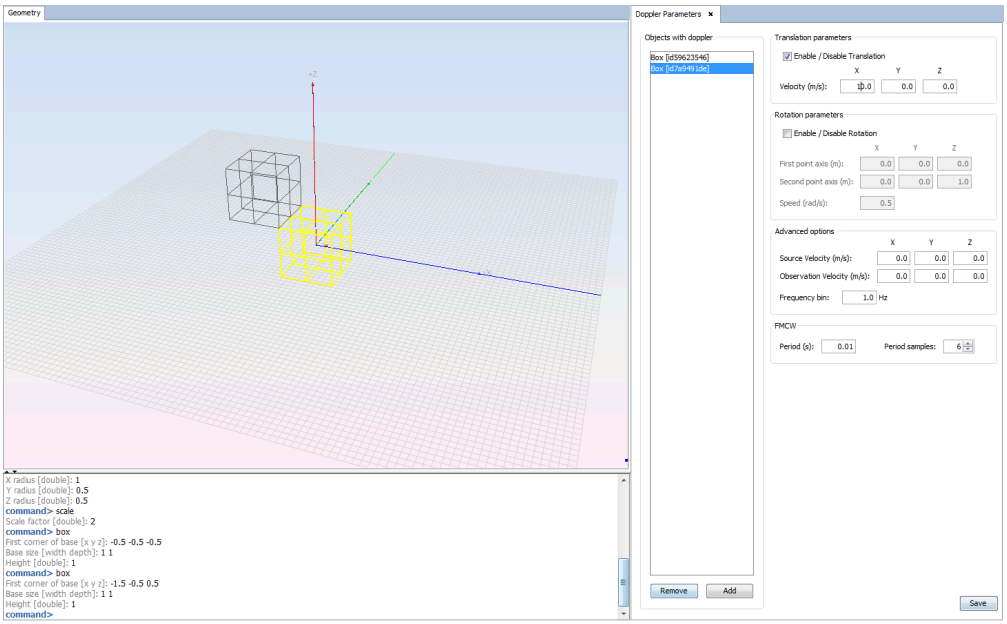


Figure 942: Doppler parameters of the second box.

Step 5

Click on RCS -> Parameters to select Monostatic RCS. As Monostatic RCS has been selected, Output menu is not available.

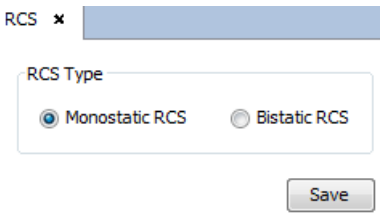


Figure 943: RCS parameters panel.

Step 6

Click on Source -> Parameters to modify the source parameters for this project. Define its position in Cartesian coordinates

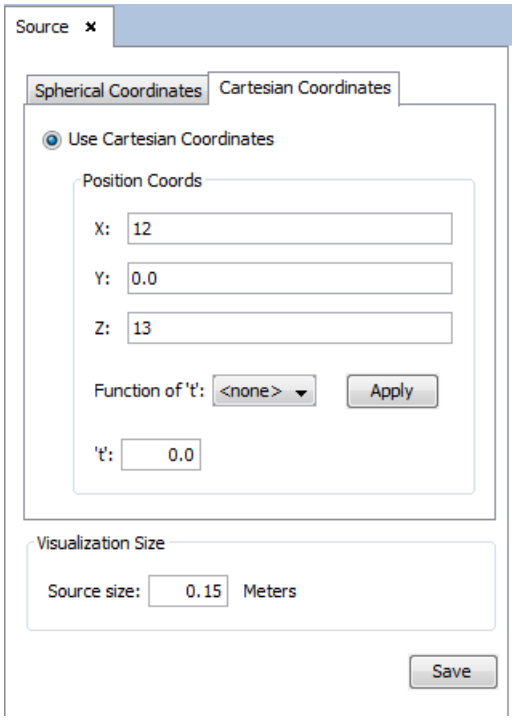


Figure 944: Source parameters panel.

Step 7

Before running this example, create the project mesh by doing click on Meshing -> Create Mesh

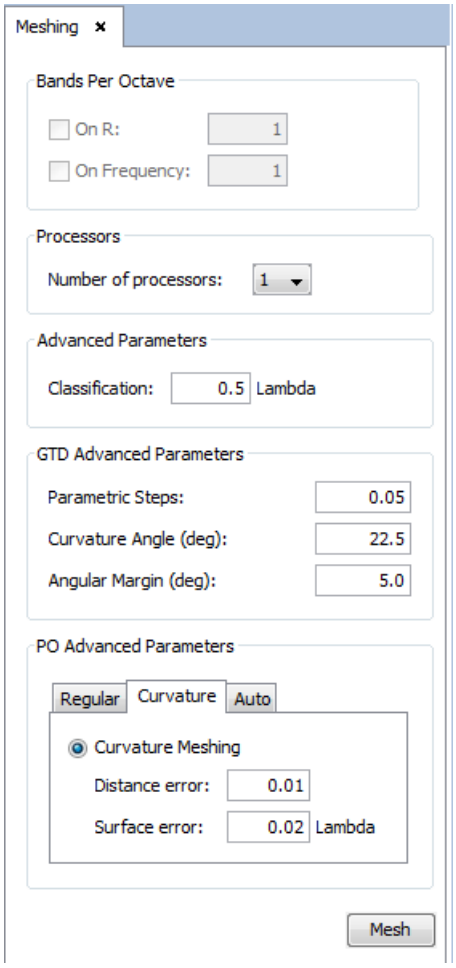


Figure 945: Mesh result.

Step 8

Now we are going to proceed to simulate the project.

Click on Calculate -> Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.

Step 9

Now you can see the results of the simulation by doing click on Show Results -> Doppler -> Doppler Spectrum.

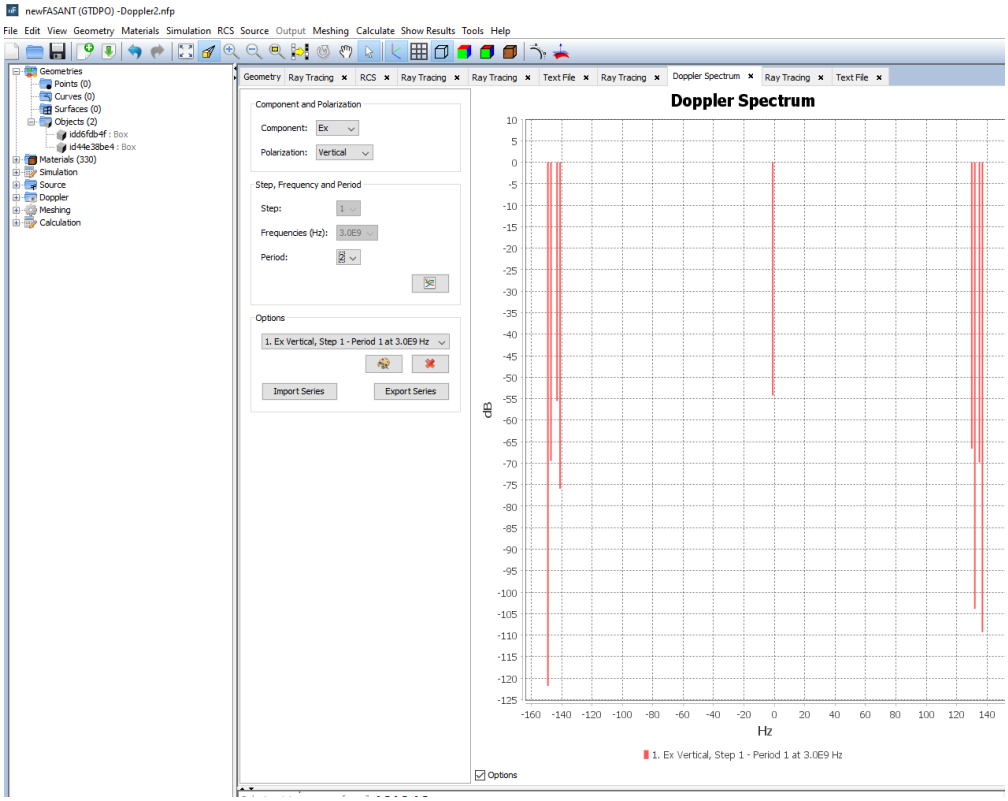


Figure 946: Doppler spectrum results.

Step 10

The ray tracing can be also visualized by doing click on Show Results -> Ray Tracing -> View Ray.

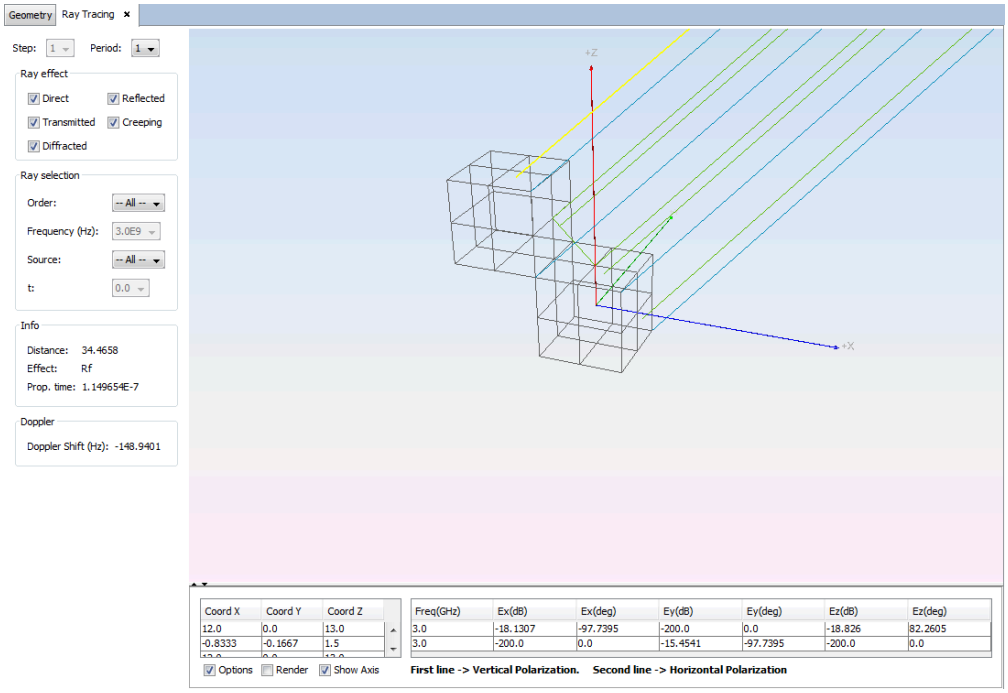


Figure 947: Ray tracing results.

9.15.6 Example 6: Monostatic RCS of a Truck

This case explains how to compute the monostatic RCS of a truck changing the distance of the source.

Step 1

Start newFASANT and select GTD-PO.

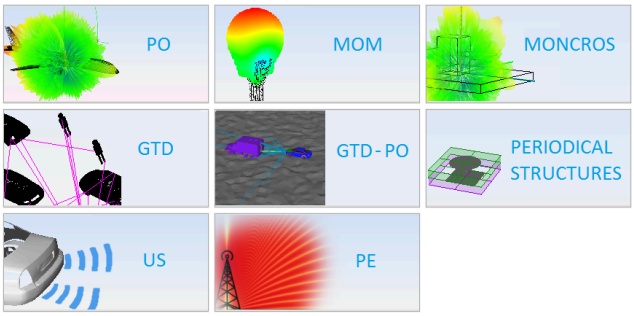


Figure 948: Method type selection.

Step 2

Select the geometry to import by doing click on Geometry -> Import.

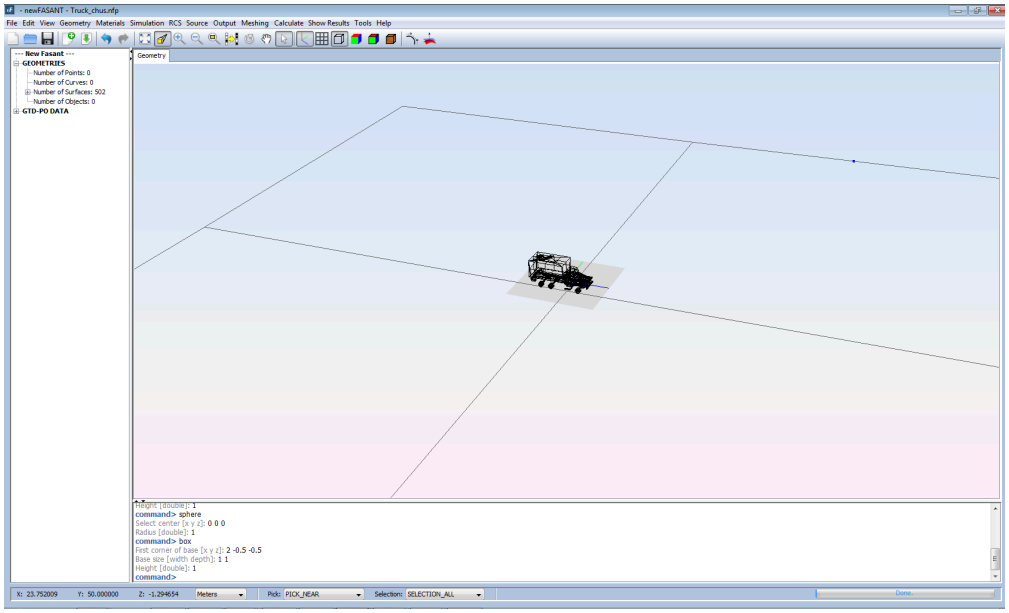


Figure 949: Import geometry.

Step 3

Click on Simulation -> Parameters to change the project parameters

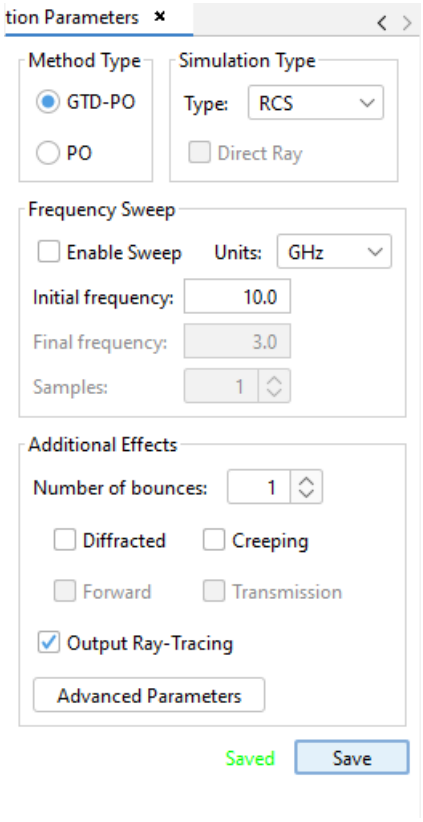


Figure 950: Simulation parameters panel.

Step 4

Click on Source -> Parameters to modify the source parameters for this project

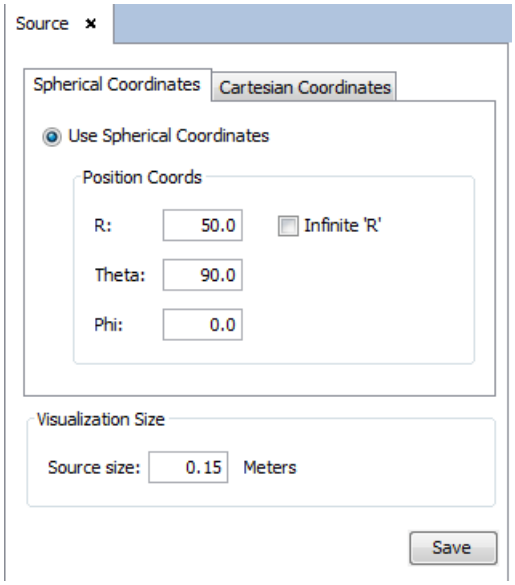


Figure 951: Source parameters panel.

Step 5

Click on Output -> Observation directions to define the parameters.

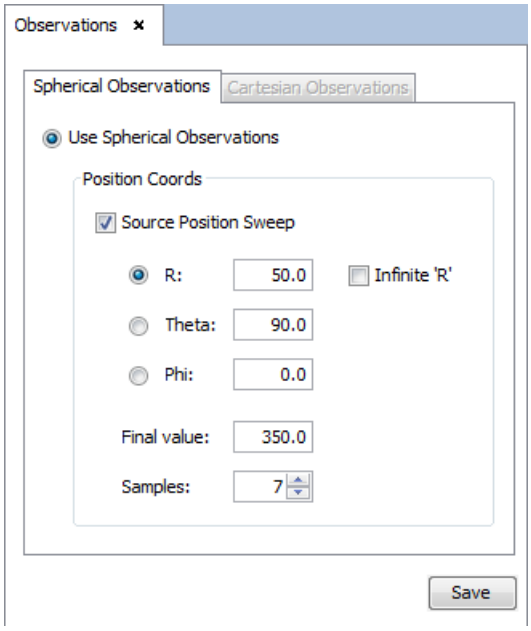


Figure 952: Observation parameters panel.

Step 6

Before running this example, create the project mesh by doing click on Meshing -> Create Mesh

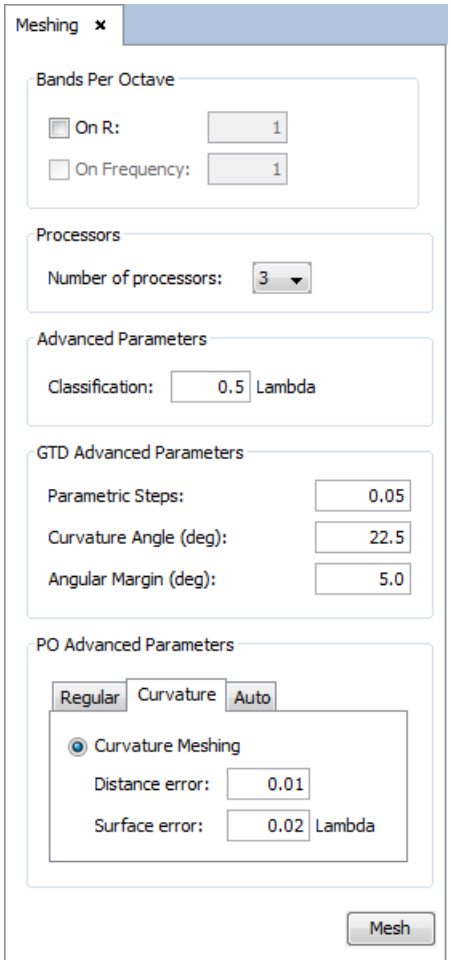


Figure 953: Mesh result.

Step 7

Now we are going to proceed to simulate the project.

Click on Calculate -> Execute and choose the number of processors to use for the simulation. Then click on the Execute button and wait until it ends.

Step 8

Now you can see the results of the simulation by doing click on Show Results -> RCS -> View Cuts.

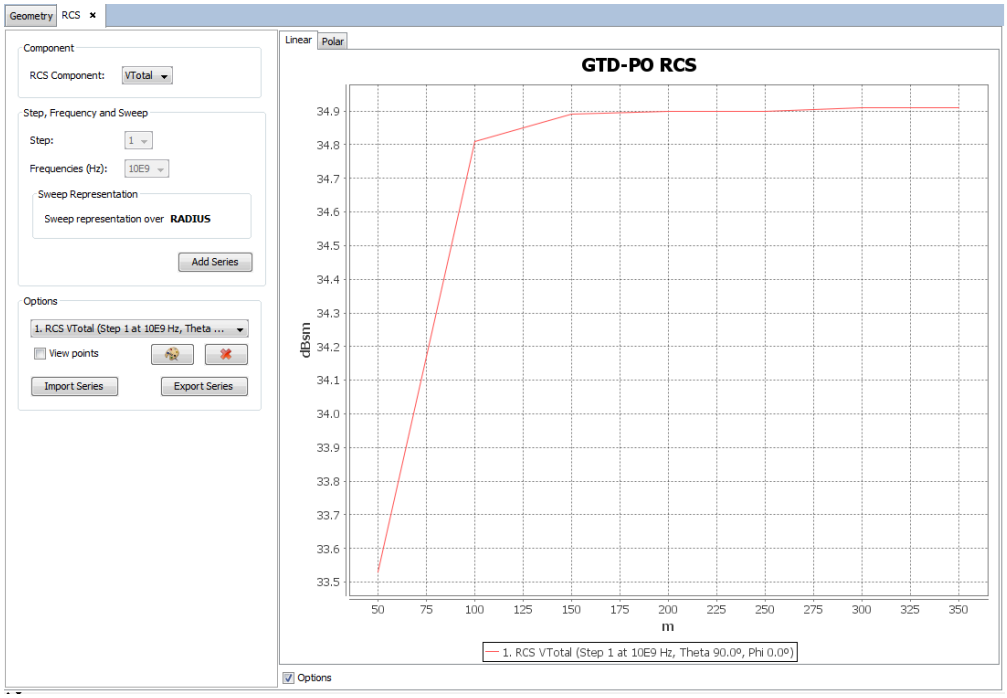
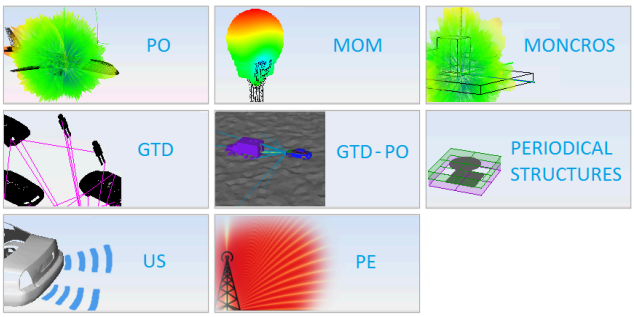


Figure 954: RCS results.

9.15.7 Example 7: FMCW Analysis

This case explains how to compute the FMCW spectrum of a simple geometry.

Step 1: Open newFASANT software and generate a new GTD-PO project selecting the option on the tab that appears when the 'File --> New' option of the menu bar is selected.



Step 2: Build the geometries. For this purpose the user can use all options available to construct geometries (see Geometry Menu). Generate a plane typing 'plane' on the command line panel, and introducing the following parameters when the command line ask for it:

- First corner of base [x y z] 6.0 10.0 0.0
- Box size [width depth] 5.0 2.0
- Height 1.0

Repeat the operation with the following parameters:

- First corner of base [x y z] -15.0 12.0 0.0
- Box size [width depth] 2.0 1.0
- Height 3.0

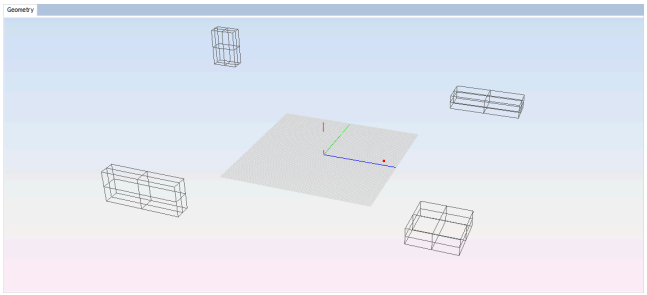
Repeat the operation with the following parameters:

- First corner of base [x y z] 8.0 -8.0 0.0
- Box size [width depth] 3.0 3.0
- Height 1.0

Repeat the operation with the following parameters:

- First corner of base [x y z] -10.0 -10.0 0.0
- Box size [width depth] 5.0 1.0
- Height 2.0

The following figure shows the created boxes:



Step 3: Set the simulation parameters using "Simulation --> Parameters" option on the menu bar.

Set ‘Method Type’ to GTD-PO, to analyze the scene classifying the object to analyze it with GTD or PO methods when corresponds.

Set ‘Simulation Type’ to ‘Doppler’ to analyze the scene considering Doppler effects.

Set the ‘Initial frequency’ to 24.05 and the ‘Units’ parameter to ‘GHz’.

Set ‘Number of bounces’ to 2 for the maximum number of rebounds of the rays.

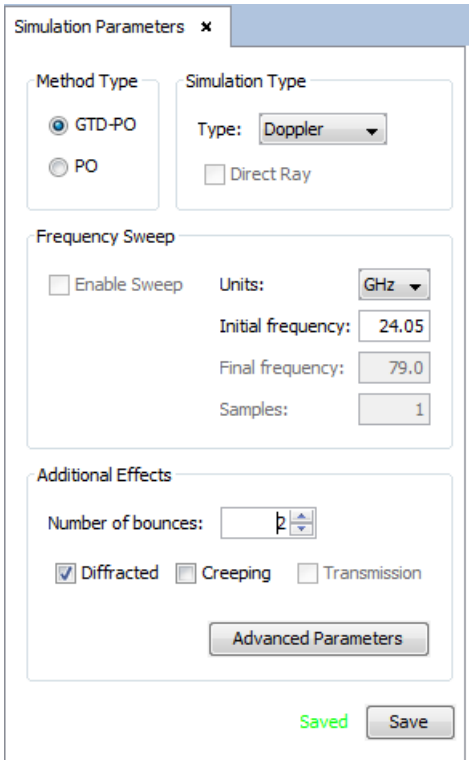
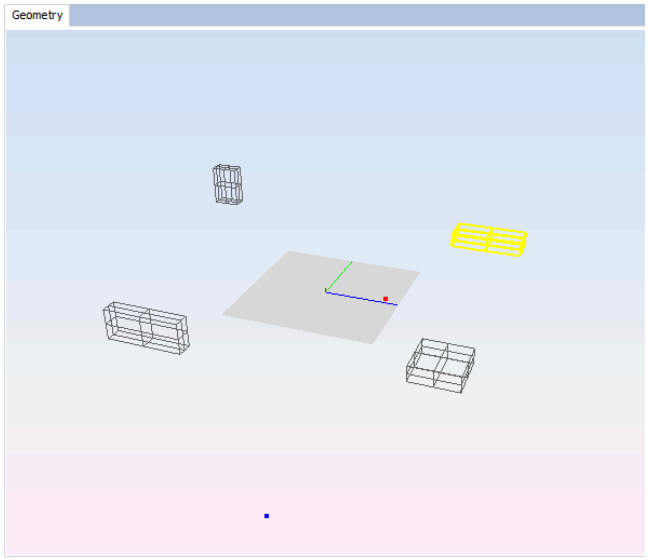


Figure 955: Simulation Parameters panel

NOTE: Selecting 'Doppler' option, "Output" menu is disabled on the menu bar and the observation will be forced to the same point that the source.

Step 4: Set the doppler parameters using "Simulation --> Doppler" option on the menu bar. With this option doppler effect will be added for the desired objects. Add the object selected in the next figure, selecting it and clicking on the “Add” button of the “Objects with doppler” panel.



Then set the parameters as the next figure shows, to add to the object a velocity of 10 m/s on the Y-Axis. Select "Enable/Disable FMCW" to obtain the analysis of 6 period of time with 0.01 seconds of duration. Then we analyze 6 periods of time with the movements on the object calculating the position of each moment with the duration of a period and the velocity.

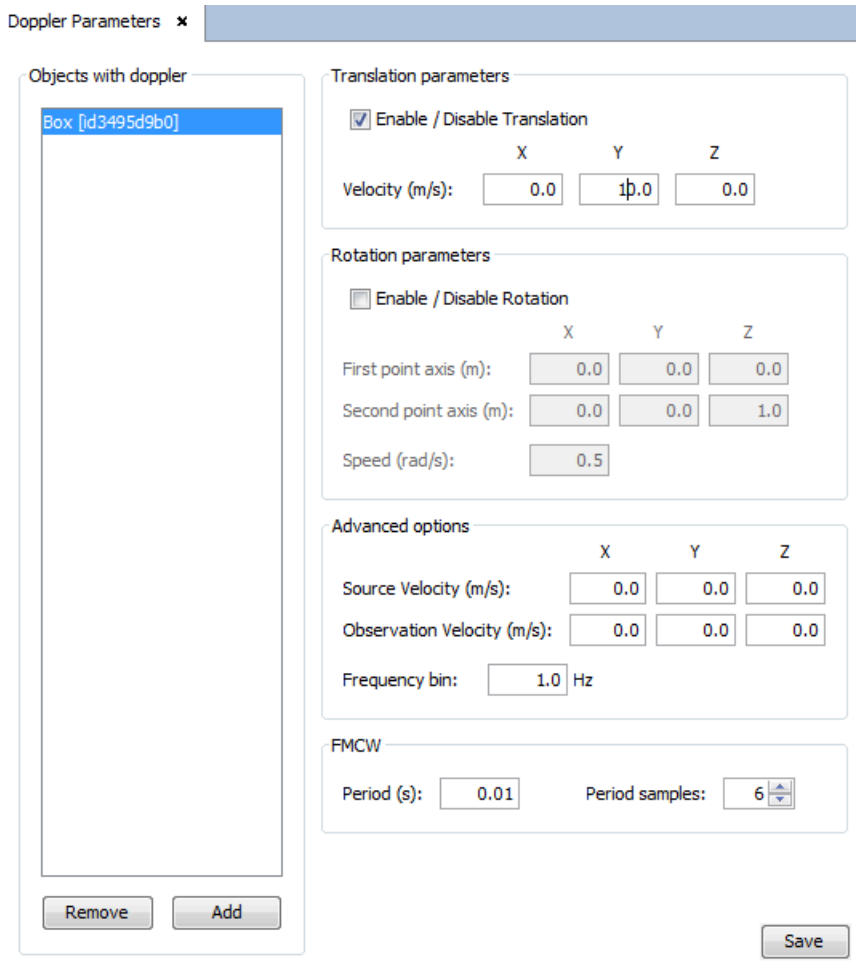


Figure 956: Dopler Parameters panel

NOTE: Selecting 'FMCW' options "RCS" menu is disabled and the RCS will be forced to monostatic.

Step 5: Set source parameters using "Source --> Source Parameters" on the menu bar. This option allows the user to locate the source defining the position by spherical or Cartesian coordinates. For this case select 'Cartesian Coordinates' and set the parameters as the following figure shows, to locate the source at the point 20.0,-6.0,1.0.

Set the 'source size' to 0.5 to optimal visualization of the source.

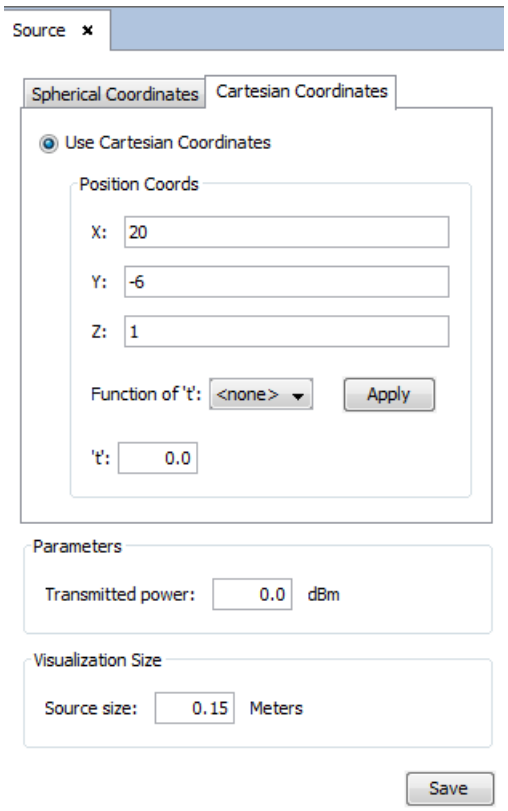
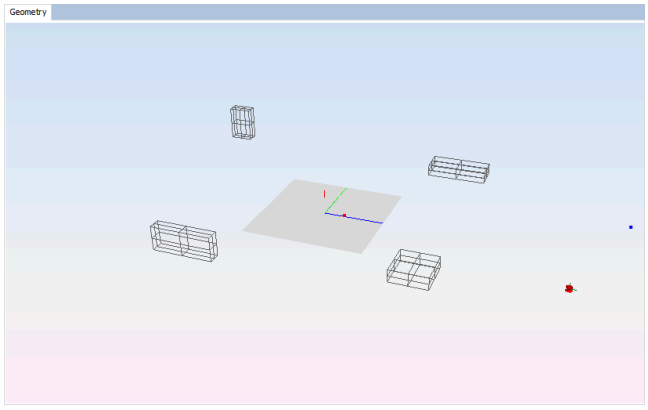
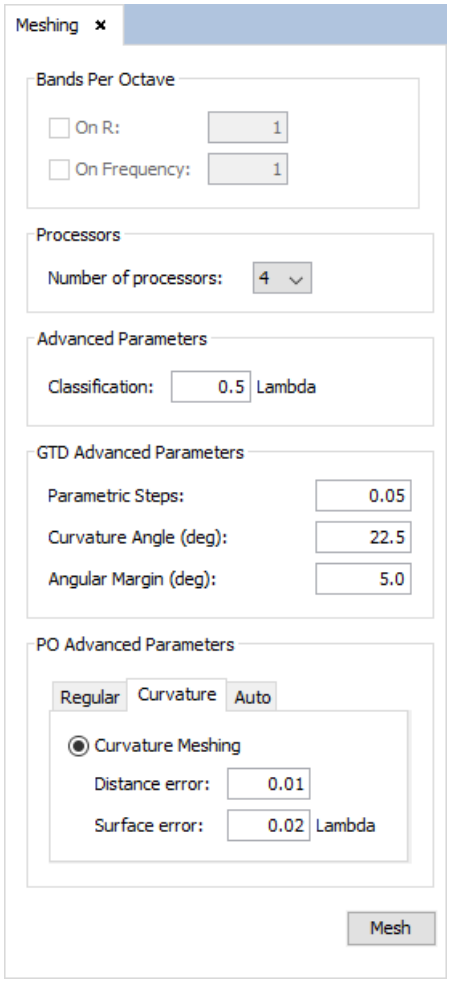


Figure 957: Source panel

Clicking on ‘Save’ button the source appears on the main panel.

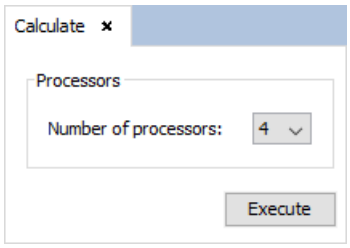


Step 6: Set meshing parameters using "Meshing --> Create Mesh" option on the menu bar. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed. Set the parameters remaining to default values.



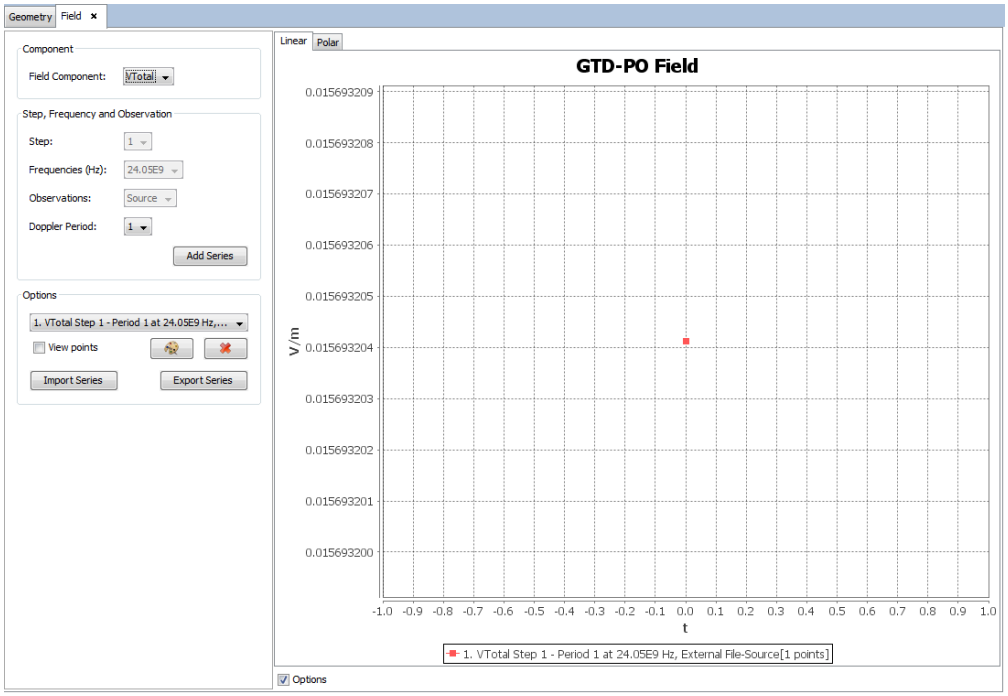
Then, the simulations for each period will be executed. When the execution has finished the result meshes can be opened with "Meshing --> Visualize an existing Mesh" option on the menu bar, and selecting the file from the directory of the selected step, frequency and period.

Step 7: Set calculate parameters using "Calculate --> Execute" option on the menu bar. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed.



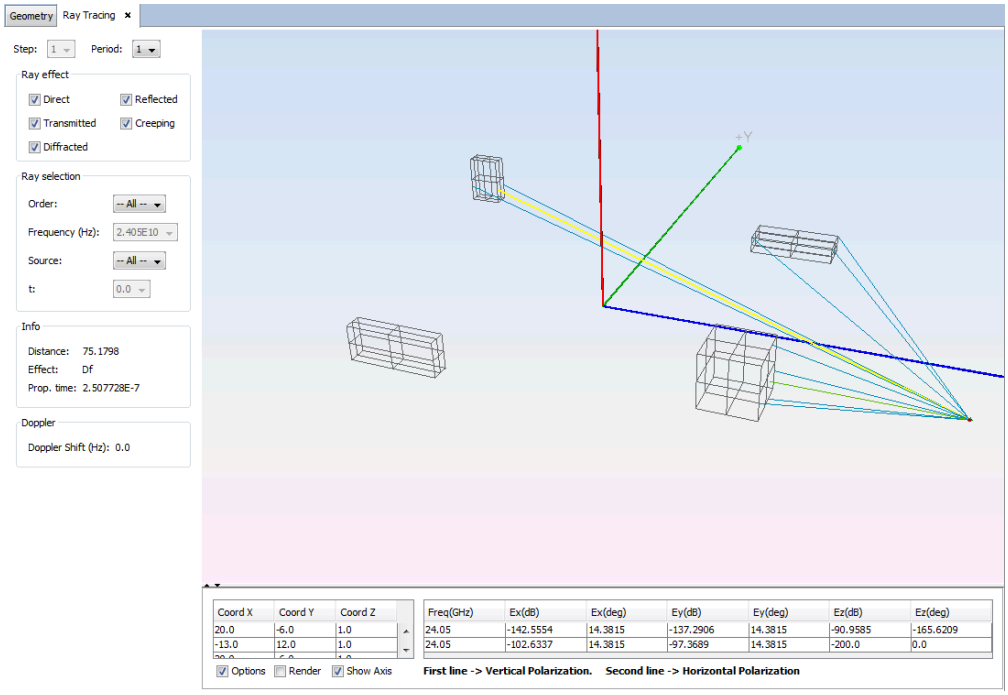
Then, the simulations for each period will be executed. When the simulations finish the results will be enabled to display it.

Step 8: Display the information about the field value with "Show Results --> Field --> View Cuts". The representation indicates the field component value on the observation point, in the case, the same point of the source, for a step, a frequency and period.



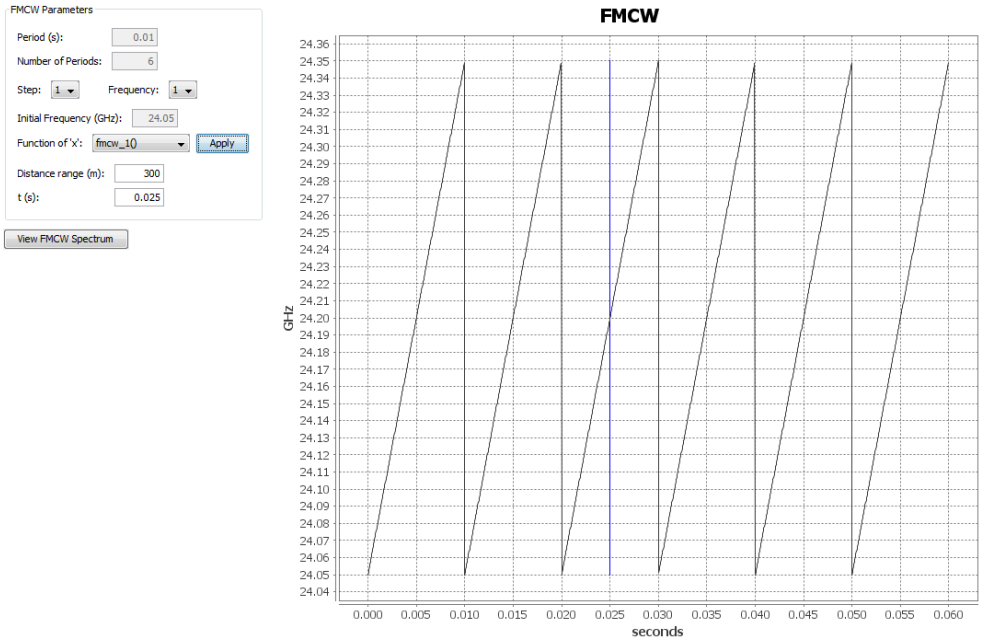
Selecting other field component, step, frequency or period and clicking on “Add Series” button adds other visualization of field.

Step 9: Display the information about the ray tracing with "Show Results --> Ray Tracing --> View Ray". The representation shows the rays for the selected step, period, frequency and source position. The rays can be filtered by effects and orders.

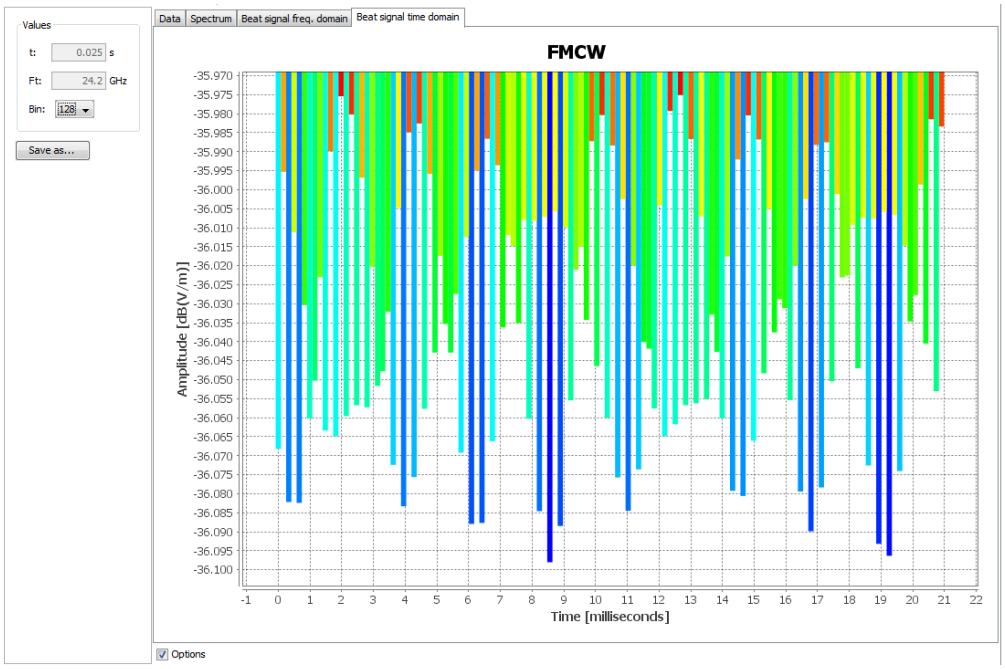


Selecting other step, frequency, period, position of the source or the orders and effects the visualization will be updated printing the rays that have the selected parameters. Selecting a ray into the panel, the information about it will be displayed on the tables at the bottom of the panel and on the 'Info' and 'Doppler' panels on the left panel.

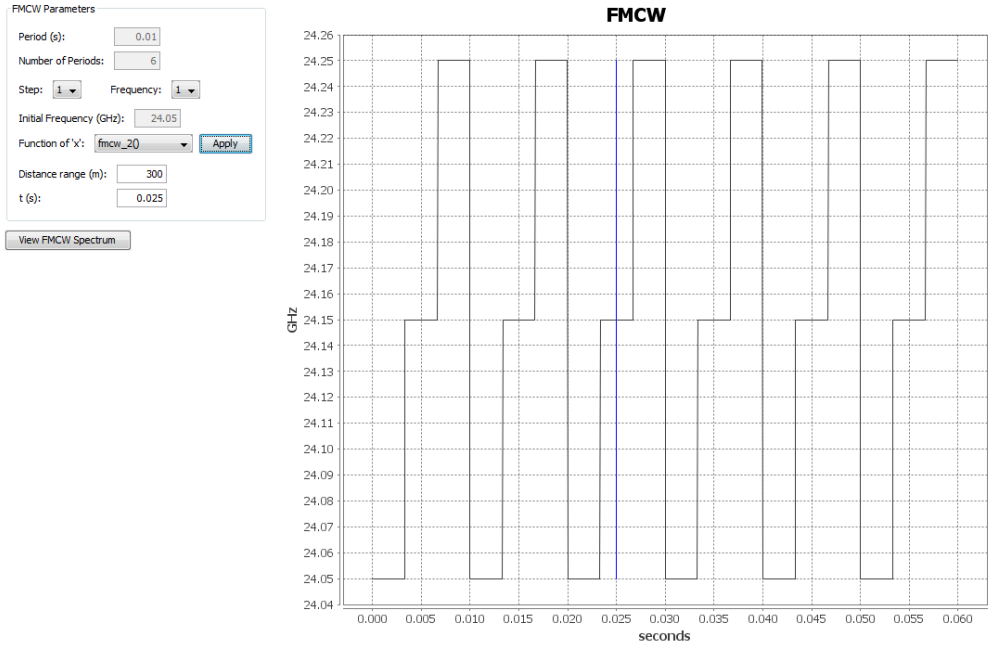
Step 10: Display the information about the FMCW spectrum with "Show Results --> Doppler --> FMCW". When this option is selected, appear a panel to set the parameters to calculate the FMCW Spectrum. These parameters are step, frequency, user function, maximum distance of the spectrum and the time to analyze. Press "Apply" button to visualize the frequency function. For this case, the parameters are correct to visualize the spectrum of the time 0.025 seconds.



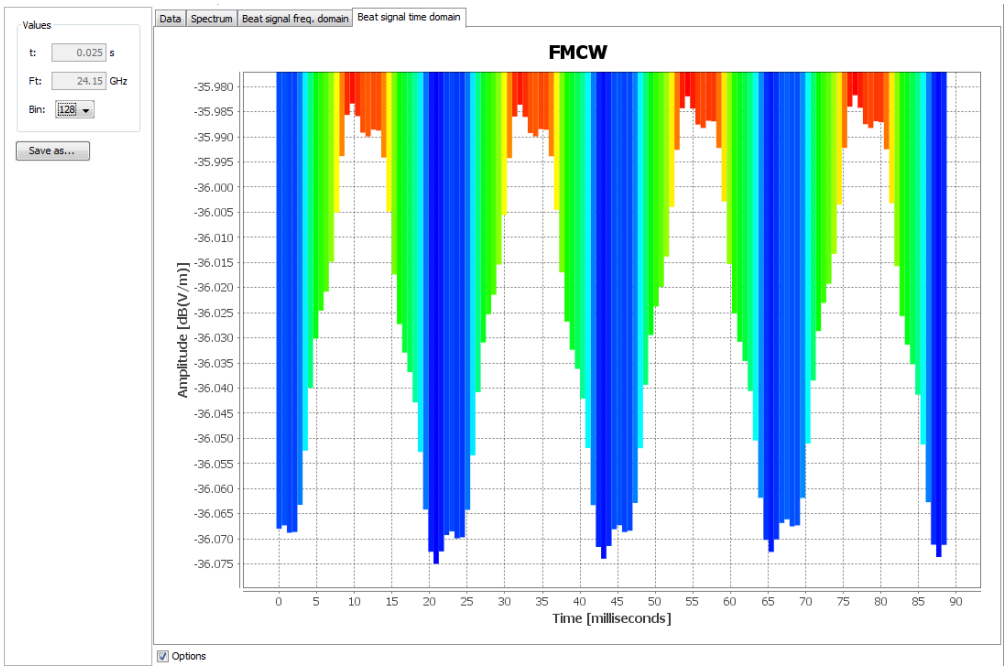
Clicking on "View FMCW Spectrum" button the spectrum will be displayed.



Step 11: The user can change the function to be applied. Change the function on the popup menu to fmcw_2(). Press "Apply" button to view the frequency function.



Clicking on 'View FMCW Spectrum' button the spectrum considering the new function will be displayed.



Step 12: To implement a function that defines the frequency in the time domain, open "Tools --> User Functions". Create a new file and programming the function using the global variables $x and $period. The $x variable defines the time. The $period defines the period and this variable is constant.

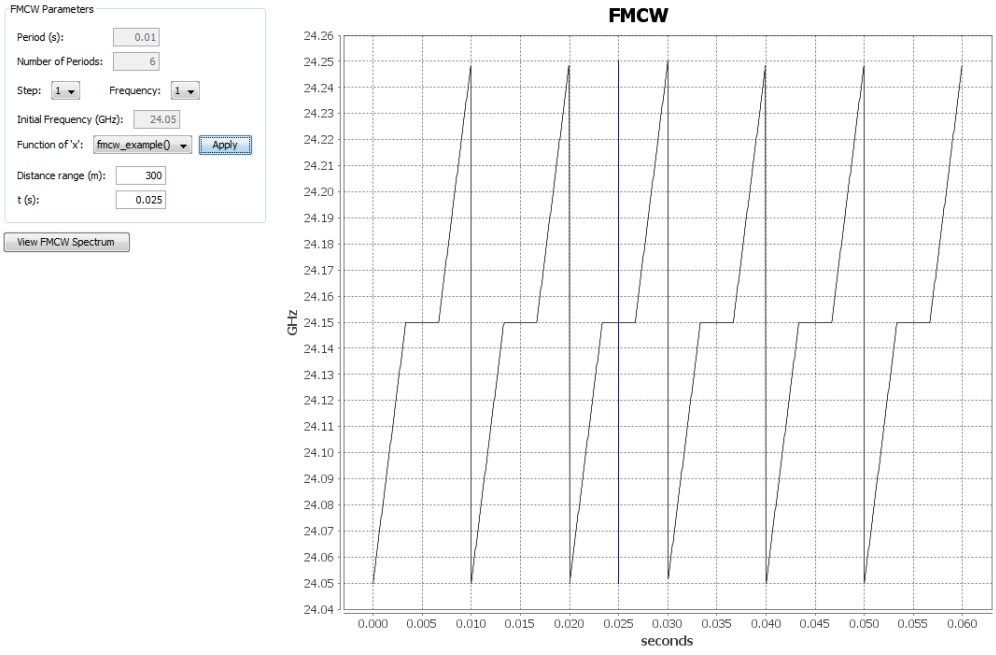
Example:

```
double fmcw_example(){
  // This function returns the frequency increment from time '$x'
```

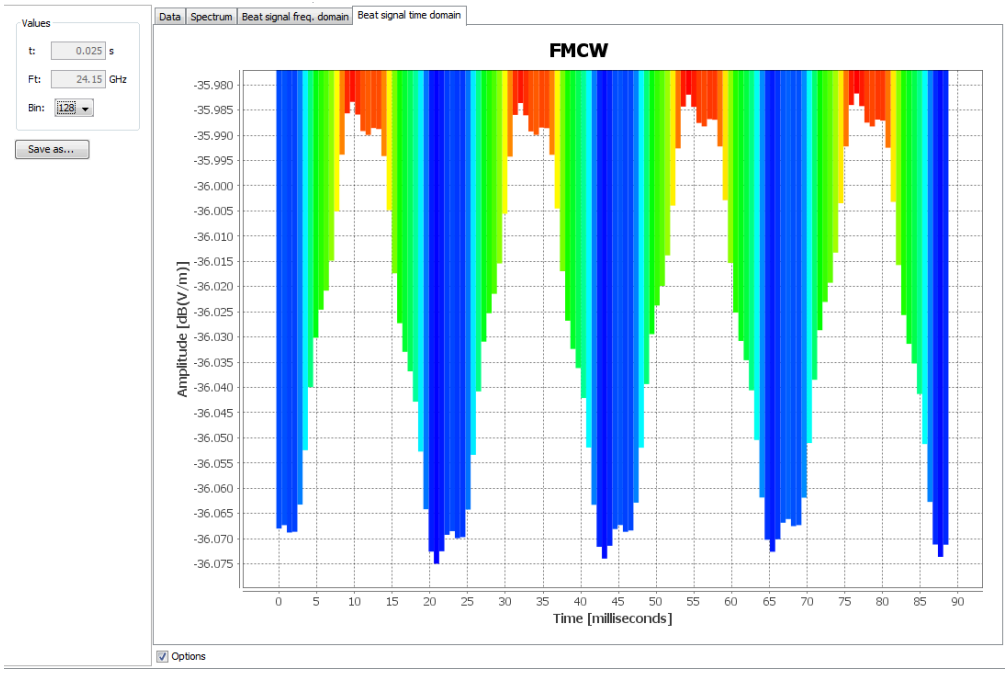
```
  // Global parameters
  // $period period time in seconds.
  // $x time within the period.
  //
  // Note '$x' it is always less than '$period'
  //

  double x1 = 1./3 * $period;
  double x2 = 2./3 * $period;
  double x3 = $period;

  double y1 = 0.1;
  double y2 = 0.2;

  if($x < x1) {
    return y1/x1 * $x;
  } else if($x < x2) {
    return y1;
  } else {
    return y1 + ((y2-y1)/(x3-x2)) * ($x-x2);
  }
}
```

In the above example are defined three ranges, for each range defines a different function.

Save file and display the information about the FMCW spectrum with "Show Results --> Doppler --> FMCW". Select the function in the popup menu and press "Apply" button. The function will be displayed into the plot.



Introduce the time, for example t(s)=0.0225, to analyze and press "View FMCW Spectrum". Select the Bin into the popup menu to change the plots.


Values
t: 0.025 s
Ft: 24.15 GHz
Bin: 128
Save as...
Data
Spectrum
Beat signal freq. domain
Beat signal time domain
FMCW
Amplitude [dB(V/m)]
-35.980
-35.985
-35.990
-35.995
-36.000
-36.005
-36.010
-36.015
-36.020
-36.025
-36.030
-36.035
-36.040
-36.045
-36.050
-36.055
-36.060
-36.065
-36.070
-36.075
0 5 10 15 20 25 30 35 40 45 50 55 60 65 70 75 80 85 90
Time [milliseconds]
Options

MONCROS

10

MONCROS is used for 3D radar cross section (RCS) using the method of moments (MOM) technique.

This chapter covers the following:

- 10.1 File Menu (p. 946)
- 10.2 Edit Menu (p. 947)
- 10.3 View Menu (p. 948)
- 10.4 Geometry Menu (p. 949)
- 10.5 Materials Menu (p. 950)
- 10.6 Simulation Menu (p. 951)
- 10.7 Solver Menu (p. 952)
- 10.8 RCS Menu (p. 961)
- 10.9 Output Menu (p. 963)
- 10.10 Meshing Menu (p. 972)
- 10.11 Calculate Menu (p. 981)
- 10.12 Show Results Menu (p. 983)
- 10.13 Tools Menu (p. 998)
- 10.14 Help Menu (p. 999)
- 10.15 Training Examples (p. 1000)
- 10.16 Annex 1: Post-Processing Currents (p. 1029)

10.1 File Menu

For information about the File menu, refer to File Menu.

10.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

10.3 View Menu

For information about the View menu, refer to View Menu.

10.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

10.5 Materials Menu

For information about the Materials menu, refer to Materials.

10.6 Simulation Menu

This menu contains the following options:

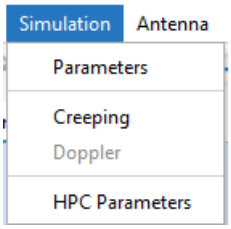


Figure 958: Simulation Parameters menu

10.6.1 Parameters

This menu lets the user modify several parameters related to the simulation process, such as the simulation frequency. Select this option to open the **Simulation** panel on the right area of the main window:

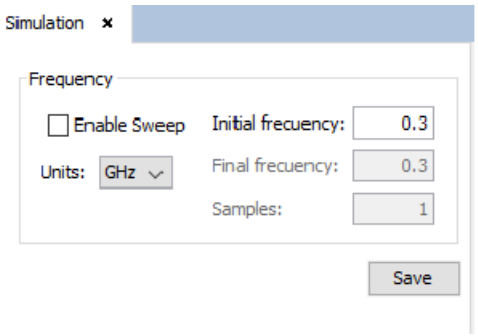


Figure 959: Simulation panel

In this panel, the user can set the main parameters of the simulation:

- **Frequency**: The user can select a single frequency or a frequency sweep to calculate the electrical field. On single frequency simulation, the user will have to introduce the simulation frequency. If frequency sweep is selected, the user will have to introduce the initial and final frequencies, and the number of samples to be simulated in the sweep.

10.6.2 HPC Parameters

For more information on how to set up a queueing system, see High Performance Computing (HPC)/ Queueing.

10.7 Solver Menu

This menu contains options that specify the behaviour of the solver being used during the simulation.

10.7.1 Parameters

Using Solver Parameters the user can specify settings that are used by the solver being used to execute the calculations. When the Parameters menu item is selected from the Solver menu, the following panel appears:

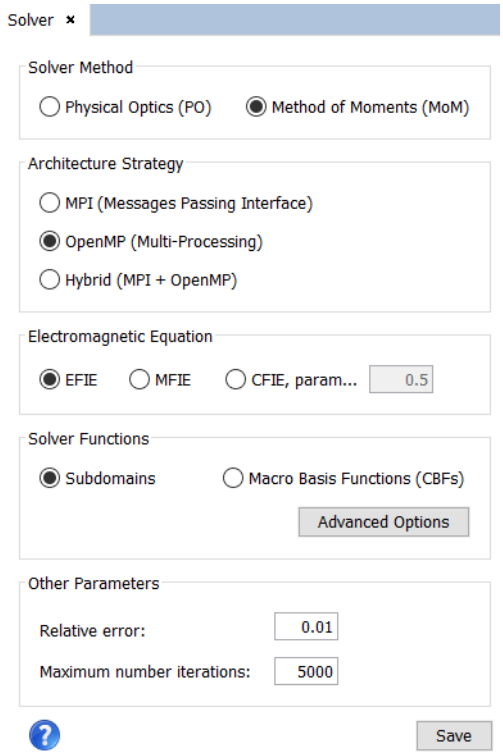


Figure 960: Solver Panel

The **Solver method** is used to specify which solution method is used to execute the calculations. Each method has a different operation mode.

- **Physical Objects (PO)** performs a high frequency approximation based in the combination of optics and full wave methods. It computes the RCS considering only the illuminated subdomains by the incident wave and with the PO current.
- **Method of Moments (MoM)** is an accurate full wave method that can be used to solve many kinds of simulations. The Method of Moments approach is used in subdomains.

A multiprocessing **architecture strategy** can be selected using the appropiate options as well. MPI is an multiprocessing architecture that works on many computer architectures, but it requires more shared memory than OpenMP, which is a memory architecture that can be selected when the constraints for memory requirements are important.

Electromagnetic equation and Solver Functions can only be set when the solver method have been set to MoM. These settings are covered later on its own section.

The user can set the **relative error** that is used by the simulation process. This is the maximum delta error accepted by the simulation. The maximum **number of iteration**s can be set as well. The simulation process makes use of an iterative process. This setting can limit the number of iterations performed by the calculation in order to prevent simulations that don't converge simulating infinititely.

Method of Moments

When the solver method has been set to MoM, there are extra settings that can be selected in order to configure how the analysis is made during the simulation process.

It is possible to set the **electromagnetic equation** as well. This option is only available when the solver method has been set to MOM. This electromagnetic equation defines the integral equation to solve during the simulation process. There are three possible equations:

- EFIE makes use of the Electric Field Integral Equation, which solves most of the problems provided a that a good convergence is achieved.
- MFIE makes use of the Magnetic Field Integral Equation. Note that this solver requires geometries to be closed, having their normal vectors pointing outside the objects, in order to obtain valid results.
- CFIE combines EFIE with MFIE -therefore, requirements for MFIE also apply on CFIE. CFIE uses a weighted combination of EFIE and MFIE. The CFIE Parameter sets this weight. CFIE uses the following equation:

CFIE = **EFIE** · α + **MFIE** · (1 - α)

Tip EFIE is best recommended on projects with open surfaces, although it works on closed surfaces as well. CFIE is best suited for closed metallic surfaces. You can use a combined approach using EFIE for some surfaces and CFIE for closed surfaces.

Warning if the solver uses the CFIE approach, it is mandatory for the normal vectors to be facing outside. Unexpected results and likely errors will happen if normal vectors point inside of volumes.

The selected **solver function** is ued to set the electromagnetic technique being used through the simulation process:

- If the **subdomains** option is chosen, the MLFMA-MoM (Multi-Level Fast Multipole Algorithm) will be used. This is the most conventional technique.
- If the **macro basis functions** (CBFs) option is selected, then the CBFM-MLFMA (Characteristic Basis Function Method - Multi-Level Fast Multipole Algorithm) is used instead. This option improoves the convergence of the MLFMA algorithm, reducing CPU consumption and time since the number of unknowns is reduced.

Tip: CBFs approach is more efficient than the subdomains approach. However, the CBF method has accuracy issues in cavities.

Advanced Solver Options

As covered before, it is possible to set the Solver Functions when the solver method has been set to MoM. Solver functions have custom settings that can be changed by selecting the Advanced Options button in the Solver Functions. Pressing this button shows a dialog with two visible tabs:

- **Main Properties**: this tab has settings for changing how the solver works, including which kind of algorithm will the solver use.
- **Preconditioner**: this tab contains available preconditioners for the selected solver.
- **CBFs Properties**: this tab is only enabled when the solver functions are set to CBFs.

The **Main Properties** tab looks like this:

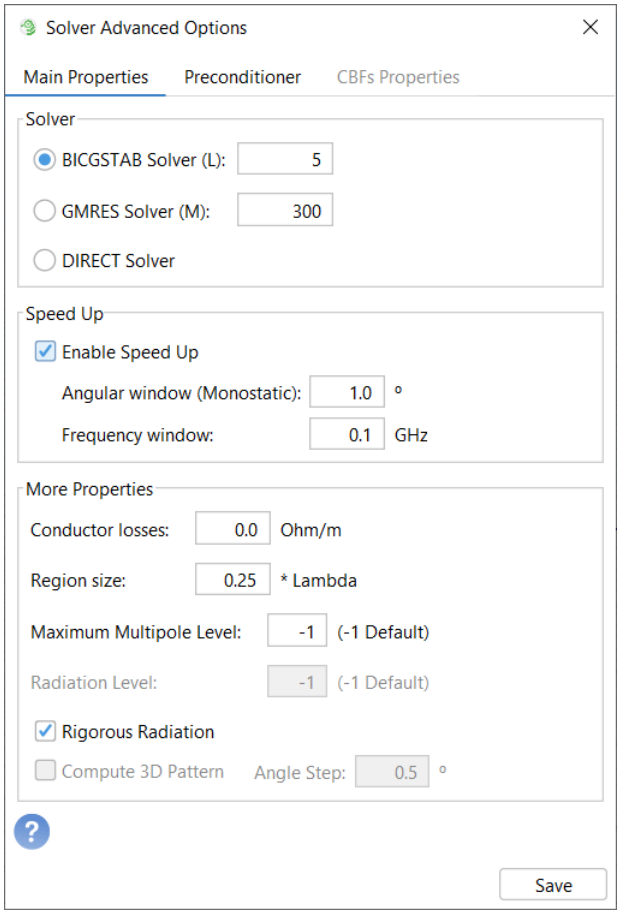


Figure 961: Main Properties

The following settings are available:

- **Solver**: Three algorithms to solve the problem are available. We can choose between two iterative methods, like BICGSTAB (BiConjugate Gradient STAbilized method) and GMRES (Generalized Minimal Residual method), and the DIRECT solver. If no convergence is achieved by using any of the iterative methods, it is recommended to try to use the other one.

  Direct Solver improves the efficiency in time when the problem is less than the number of unknowns introduced. Direct Solver is enabled only if OpenMP Architecture is selected on the main Solver window, and is not compatible with Speed Up and preconditioners.

> **Note:** The direct solution method may require huge memory and time resources when a large number of unknowns is considered.

- **Speed Up**: If the user enables the speed-up, this option reduces the analysis time for problems that have several incident angles at different frequencies.
- **More Properties**: This group lets the user set other special parameters used by the solver, such as the conductor losses, region size or maximum multipole level. Rigorous Radiation improves efficiency of the solved solution, but is not compatible with the 3D Radiation computation. The 3D Radiation computation is not compatible with Monostatic RCS.

The Preconditioner tab contains the following options:

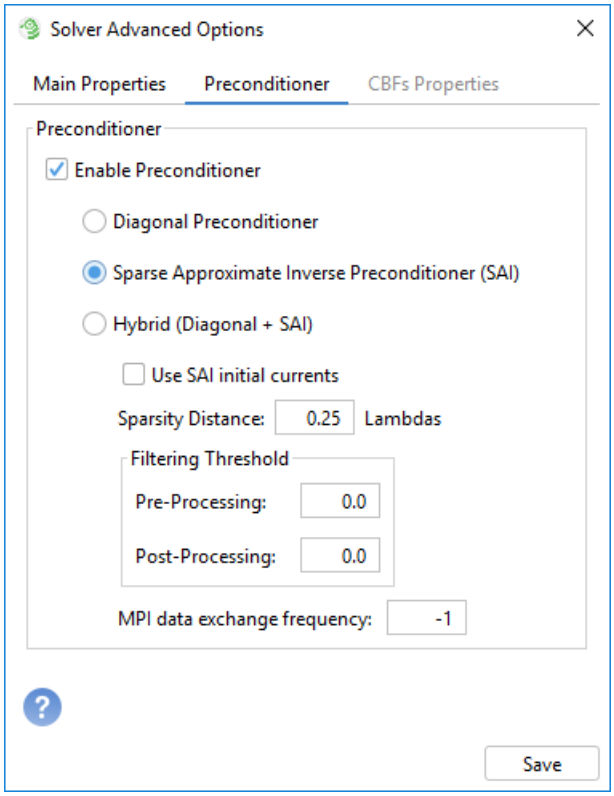


Figure 962: Preconditioner

This tab mainly covers the usage of the Preconditioner, which speeds up the resolution of the problem by selecting one of the possible preconditioners provided.

Diagonal Preconditioner

Use this preconditioner only when the mesh density is 10 div/lambda or higher.

Sparse Approximate Inverse Preconditioner (SAI)

- **Use SAI initial currents**: This option set the currents computed using the SAI preconditioner as the inicial vector of the iterative method. It may be useful if not convergence is achieved in the solution of the problem.
- **Sparsity Distance**: Set this parameter between 0.2 and 2. Usually hight values increase convergence speed and cpu / memory needs.

- **Filtering Threshold**: These parameters should contain a value between 0.0 and 1.0. The default values should be adequate in most cases.
- **Pre-Processing**: This parameter controls the amount of data considered to generate the preconditioner. Lower values entail a more accurate generation, while higher values entail a faster computation.
- **Post-Processing**: This parameter controls the amount of data to be stored after the generation of the preconditioner. Lower values entail better convergence, while higher values entail less RAM required to store the preconditioner.
- **MPI Data Exchange Frequency**: This parameter sets up how often the MPI nodes request more coupling terms to generate the preconditioner. Larger values require less interactions speeding up the simulation, although more memory will be needed to store these terms. A negative or 0 value indicates that the coupling terms are only exchanged once.

Hybrid (Diagonal + SAI)
This preconditioner is suited for memory-shared machines, in which case will use both Diagonal and SAI preconditioners.

When CBFs are enabled, the following options are available as well:

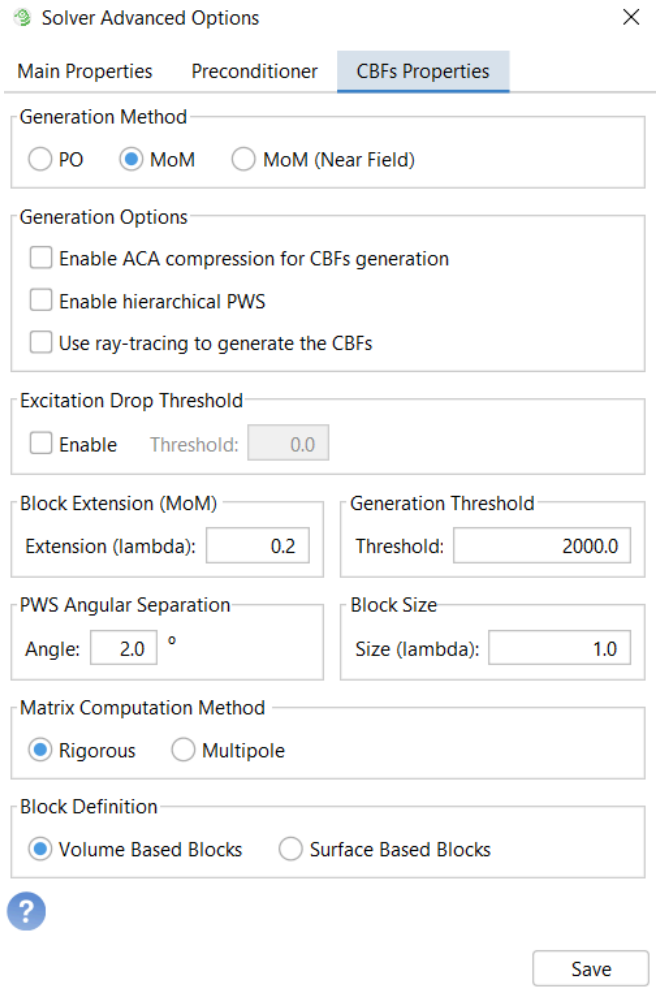


Figure 963: CBFs Properties

These settings let the user configure different parameters of the CBFM method. For most of the analysis, default parameters are suitable.

- **Generation method**: CBFs can be generated using PO currents or MoM currents (default MoM). In this last option, it is desirable to extend the size of the block to avoid the edge effect. The extension is selected in the **Block Extension (MoM)** panel (by default, 0.2 lambda).
- **Generation Options**: There are several techniques that could speed up the generation of CBF. In this panel, you can select the compression of the PWS using the ACA technique, the hierarchical computation of the CBFs by dividing the PWS into groups, or the use of Ray Tracing techniques to establish the number of CBFs to retain at any block.
  - **Enable ACA Compression for CBFs generation**: This check box enables a compression algorithm using ACA in order to reduce the number of excitations used to obtain the CBFs. This will decrease the total simulation time.
- **Excitation Drop Threshold**: With this method, a second stage is implemented to discard cbfs. In this case, only CBFs that get a significant impressed field due to external sources are preserved. If used, a threshold of 0.01 is recommended. Lower values give more accuracy.

- **Generation Threshold**: It is used to set how many cbfs should be retained to solve the problem (Default 2000).
- **PWS angular separation**: Defines the angular separation between two plane waves in the PWS, which determines how many plane waves will be used in generating the PWS. (By default, 10º).
- **Block size**: This parameter defines the size of the block of the CBFM method in lambdas.
- **Matrix calculation method**: You can define the method to calculate the reduced matrix of CBFM, using rigorous calculation or multiple approximation.
- **Block definition**: This option allow to choose the definition of the block. "Surface-based blocks" define a block that includes only subdomains inside of a MLFMM region that belongs to the same surface. "Volume based blocks" define a block as the set of all subdomains inside of a MLFMM region.

In order to save the solver configuration press the **Save** button.

10.7.2 Advanced Options

This submenu lets the user manage advanced options that might be required for some specialized simulations.

Specify EFIE can be used to override the selected electromagnetic equation for some geometries when the global electromagnetic equation is set to CFIE. CFIE requires geometries to be closed in order to get the best results. This menu can be used to select some geometries that should be analyzed using EFIE despite having other objects already in the simulation using CFIE.

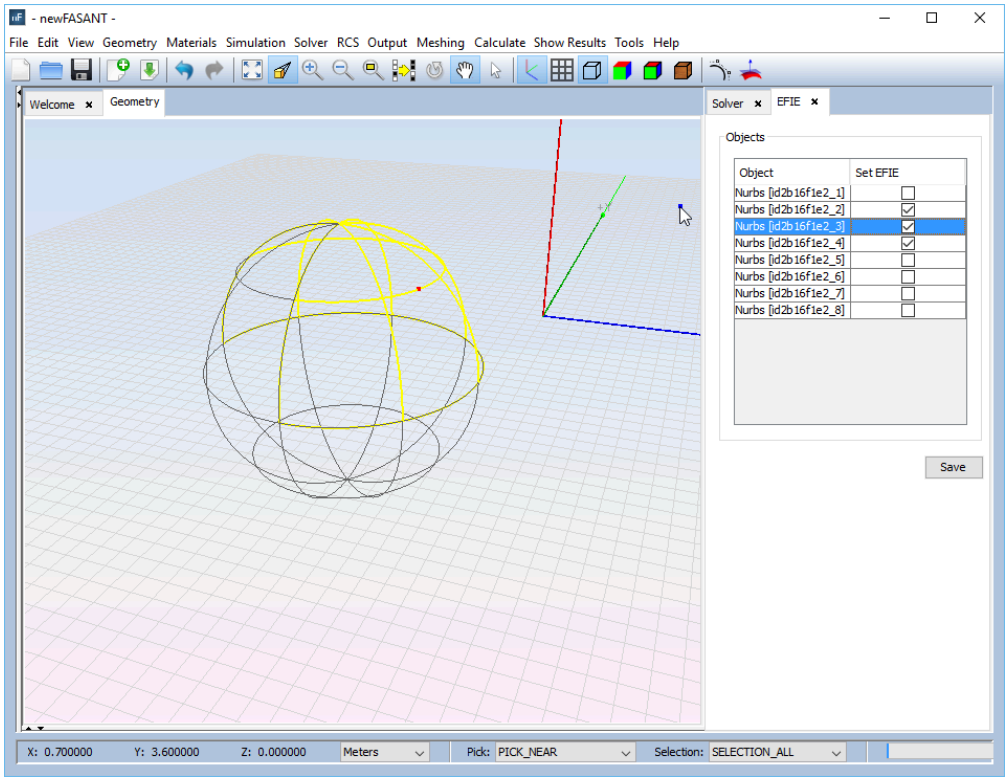


Figure 964: CFIE Parameters

In order to use this tool, open this menu when the solver equation has been set to CFIE. A list with every object defined in the user interface, surfaces and objects, will be displayed in the EFIE Panel that will appear. Use the Set EFIE check box near every row to mark that object to be analyzed using EFIE equation. Press Save to save changes after every object has been selected.

Specify Divisions can be used to force the sampling density and the divisions for a particular object in the user interface. This can be used to override the global sampling and division values for a particular object or surface.

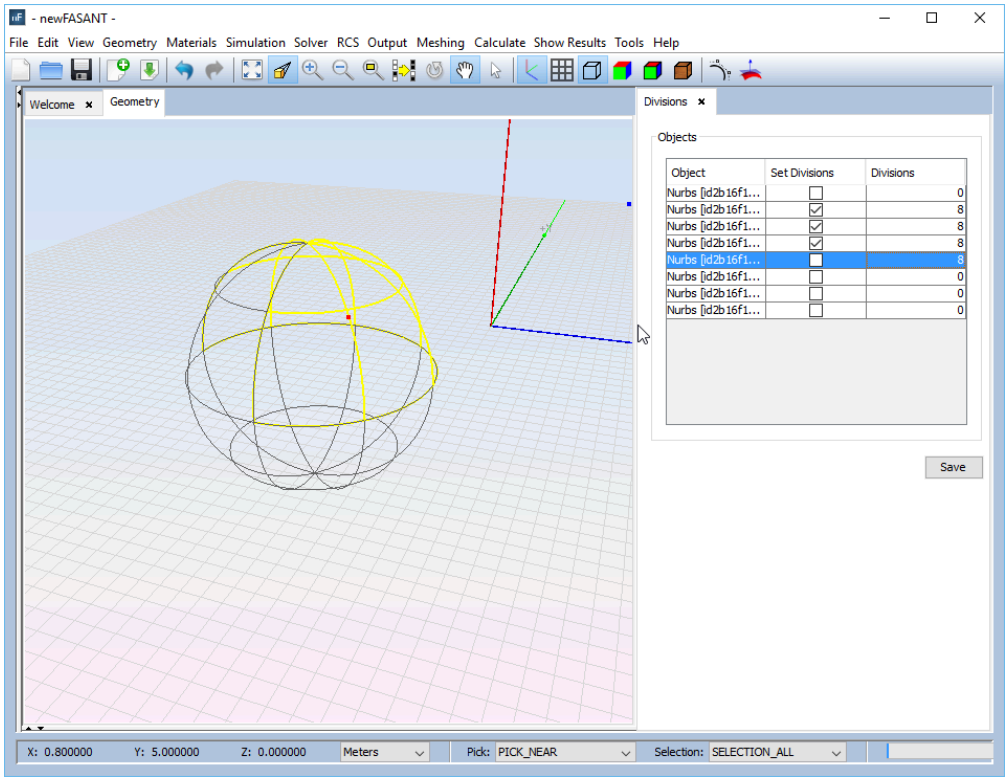


Figure 965: Specify Divisions

In order to use this setting, open this menu item. The Divisions tab will appear on the screen listing in the table every object and surface defined in this project. To override the global divisions value for a particular object or surface, enable the check box that appears in the same row, and change the divisions value near the check box. Press Save when all the desired settings have been changed in order to save the settings in the project.

10.8 RCS Menu

This menu lets the user modify the settings for the RCS radar being used in this simulation, as well as other parameters related to the plane wave. To enter RCS settings, go to RCS > Parameters.

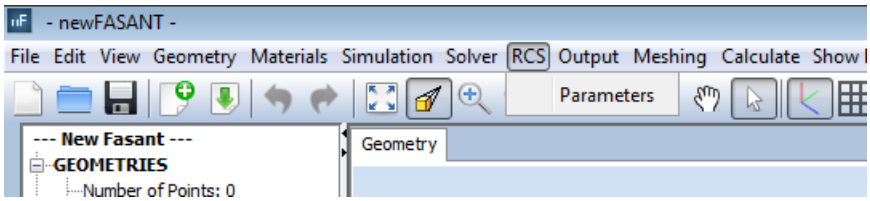


Figure 966: RCS Menu

Selecting the RCS → Parameters option will show the following panel.

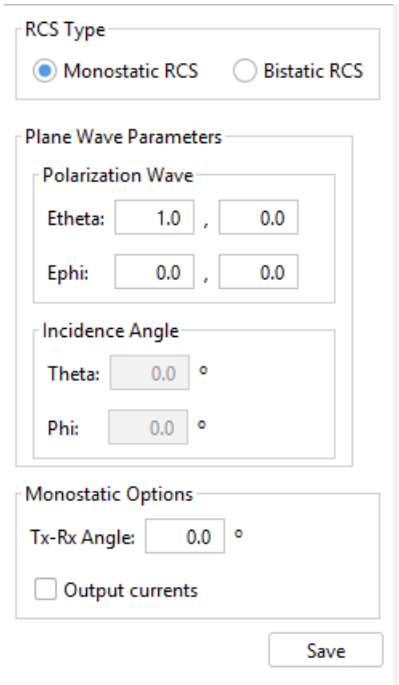


Figure 967: RCS Parameters Panel

The following settings are available:

- **RCS Type**: The RCS can either be monostatic or bistatic depending on whether the radar transmitter and the radar receiver are co-located or not.
- **ETheta** and **EPhi**: These parameters define the polarization wave for the plane wave.
- **Tx-Rx Angle**: The user is able to define this value when the RCS is set to monostatic.
- **Angle of incidence**: The user is able to define this value when the RCS is set to bistatic.
- **Output currents**: This option configures that the current distribution over the surfaces could be shown in the Result menu when the analysis finish.

The kind of observations available in the Output menu depends on the RCS Type selected in this window. Bistatic RCS allows Observation Points where as Monostatic RCS doesn't. Therefore, if RCS is

changed from Bistatic and Monostatic the user wil have to confirm the operation, since any previously defined Observation Points will be removed.

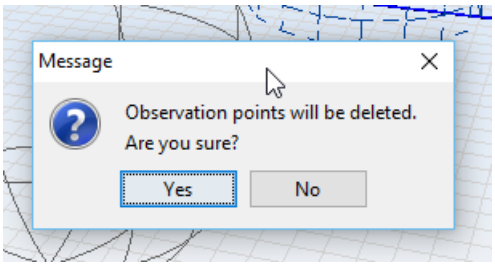


Figure 968: Confirm observation points

10.9 Output Menu

The Output menu lets the user modify observation directions and other settings related to the output generated by the simulation. The visible menu items will depend on the type of RCS simulation previously selected in the RCS menu.

If in the RCS menu, the user selected the monostatic RCS option, the following menu appears:

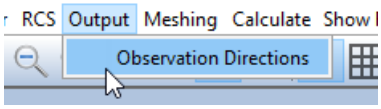


Figure 969: Output Menu on Monostatic RCS

However, if the RCS type is set to Bistatic, the user will be able to enable and disable 3D Diagram generation as well:

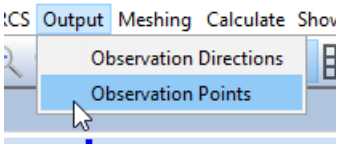


Figure 970: Output Menu on Bistatic RCS

10.9.1 Observation Directions

Observation directions are used when inspecting the generated results at the user interface, such as the generated Far Field results. This option is used for letting the user add, modify and delete these cuts. Select Output > Observation Directions to show the panel described in the figure:

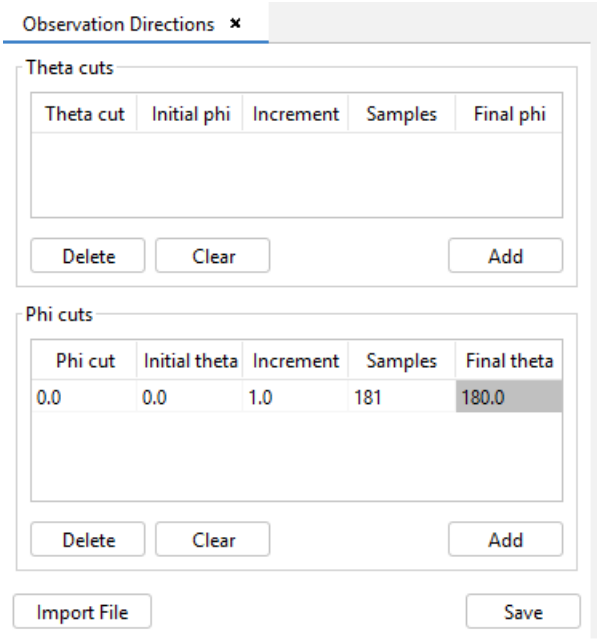


Figure 971: Observation Directions panel

In order to make the calculations useful, the user should configure observation directions where the electrical field will be calculated. These directions will be selectable by the user after the calculations have been done when inspecting the results. To add a cut, either a theta or a phi cut, press the **Add** button to add a row to the associated table, and fill the values in the row.

- Theta Cut / Phi Cut this is the value at which theta will be fixed in a conical cut (theta cut) or at which phi will be fixed in a planar cut (phi cut).
- Initial Phi / Initial Theta this is the initial value for the sweep in that observation direction cut.
- Increment this is the value the angle will be incremented in each iteration for the sweep.
- Samples the number of samples to take for the cut.

Final Phi / Final Theta is a non-editable value that will display the final angle for the sweep. This value is actually calculated using the following simple formula **Initial** + **Samples** * **Increment**.

Rows can be deleted by selecting them in the tables and pressing Delete. Press the Save button when done.

Additionally, this information can be included using an external file, pressing the 'Import File' button and uploading the file with the data. The content of this file will have a row for each cut to enter

For the definition of each cut, two formats are possible:

- A 4-field format separated by the space character. Each field refers to the angle theta, phi, steps and samples of the cut.
- A two-field format separated by the space character, detailing the angles theta and phi to be calculated are detailed (in this case, the sample data is assumed to be 1).

10.9.2 Observation Points

This option is enabled when the RCS Type has been set to Bistatic. It allows the user to set a series of observation points that will be used later when inspecting the Near Field. In the following sections inside this chapter they will be covered.

When the menu item is selected, the following panel appears:

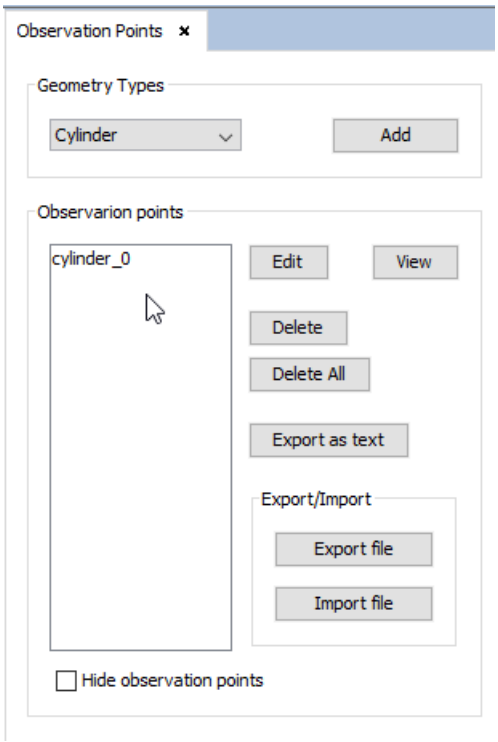


Figure 972: Observation Points panel

The user can add new geometry types by selecting the correct option presented in the Geometry Types list and pressing **Add**. Note that these geometries are just considered as "geometries" inside this panel to make it easier to manage them. When they are added to a project, the simulation will consider every point in the geometry mesh as an actual observation point. The behaviour for each kind of geometry type will be described later.

The **Observation points** list displays a list with every geometry added as an observation point.
By default, every point and geometry added to the project is displayed in the Geometry view. This behaviour can be changed by checking or unchecking the **Hide observation points** check box item.
Buttons near the list are used to modify and delete these observation points.

- **Edit** use this button to modify the data for a geometry using a panel similar to the one that appears when **Add** is pressed.
- **Delete** use this button to delete a previously added geometry from the observation points list.
- **Delete All** use this button to delete every previously added observation point. Note that this will delete **everything**.
- **Export as text** use this button to export the list of points in the selected geometry. The exported file is a text file with the X, Y, Z components for every point in CSV format.

- **Export file** use this button to export the current observation points as an OBS (Observation Points) File. It will be possible to import an OBS file using the **Import file** in other MONCROS project or in other modules that also make use of Observation Points.

It is also possible to view the list of observation points that form a geometry by selecting the geometry item from the list and pressing **View**. A new panel listing every point in the geometry will appear.

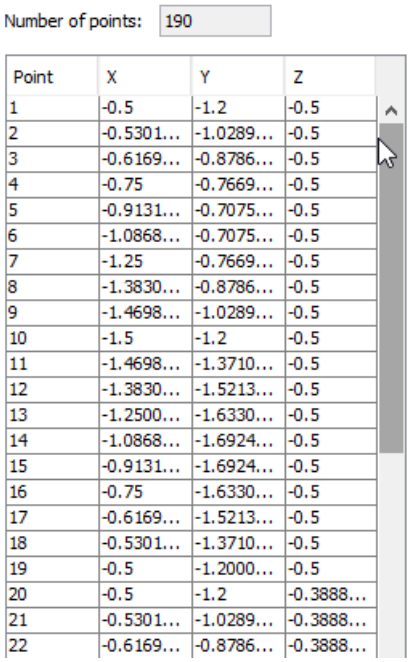


Figure 973: View Points

Cylinder

A cylinder requires a valid name, coordinates for the base cylinder (Base Center), its radius (Radius) and the height and angle coordinates. For the Height Coordinates, the Height and the Number of points must be specified. For the Angle Coordinates, the user must introduce whether a complete or an incomplete cylinder is to be generated by specifying the Initial angle and Final angle parameters. It is also possible to change the Number of points.

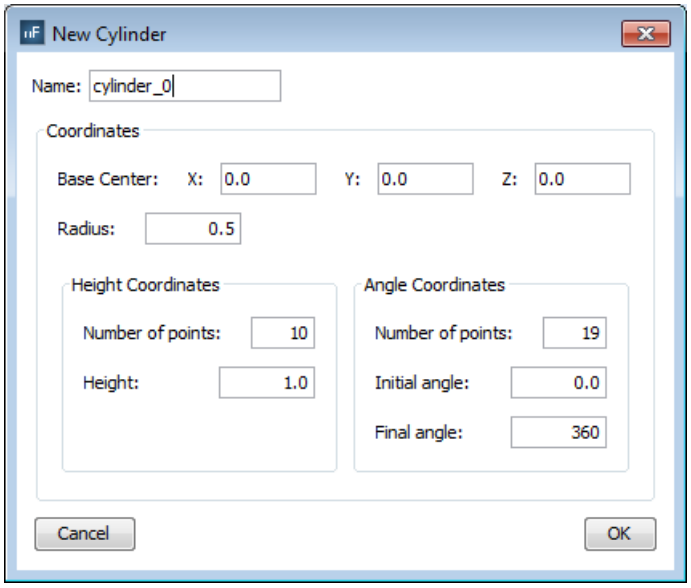


Figure 974: New Cylinder parameters

External File

The user can also export the near field points to a text file (Export as text). This file can then be imported by selecting "External File" into the Geometry Types combo-box. Note that this option imports a file of points, but does not preserve the near field type (i.e. it is imported as a separate type). In order to preserve the near field type, the Export file and Import file must be used instead.

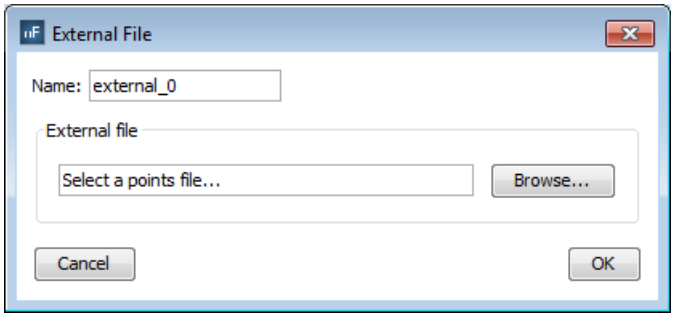


Figure 975: Open External File dialog

From Curve

This option lets you select a line or curve of the main panel and sample it as observation points. The user has to previously select a NURBS Curve. Using the given NURBS curve the number of points to be added is a parameter.

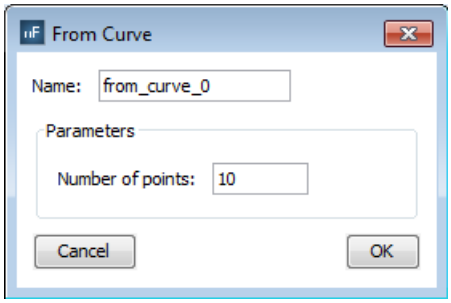


Figure 976: Adding a nurbs curve of observation points

From Surface

Using this option it is possible to create observation points using a surface as a reference. To create one of these geometries, a NURBS Surface has to be previously selected. This operation is usually done by selecting a previously created surface in the project, or exploding an object to select the desired surface. Then, a new geometry type is added. The number of points in U-Dimension and V-Dimension are parameters for this kind of geometry.

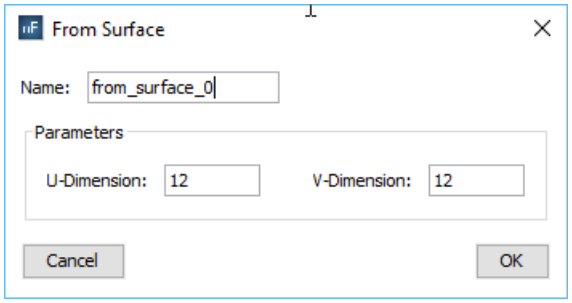


Figure 977: Adding an observation point From Surface

Line

A line is created using the initial and final point as endpoints for the line. As a parameter, the number of points will have to be introduced. The line will be split in as many points as introduced here and every point will be considered as an observation point.

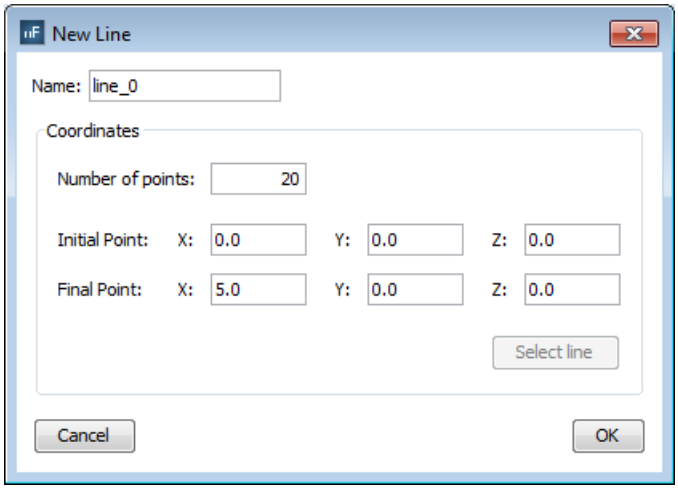


Figure 978: New Line parameters

Parallelogram

To configure the parallelogram, the user can introduce the coordinates of the three points that shape the final figure. Furthermore, the number of points between vertices can be introduced (i.e. Number of points from p1 to p2).

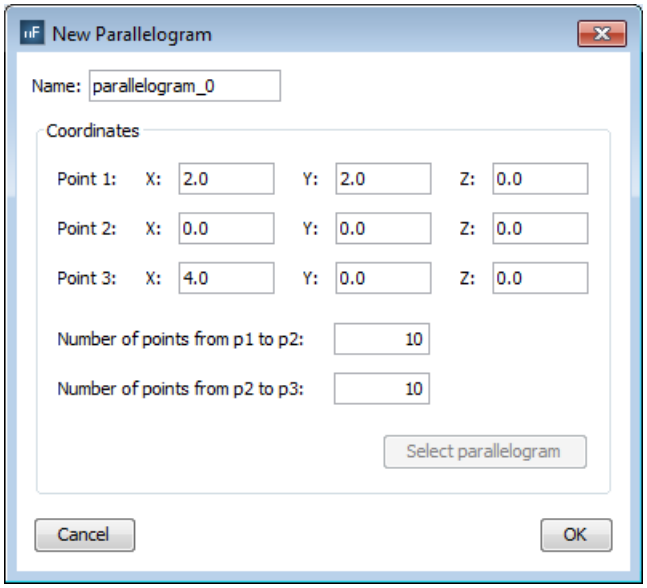


Figure 979: New Parallelogram parameters

Plane

In this window, the user can create a set of observation points laid out in a grid pattern. To do so, the user needs to specify several parameters:

- **Name** A short name that will be used by the user to identify the created set of observation points.

- **Constant axis** The normal axis of the plane that will contain the grid of observation points.
- **Coordinate value** Value of the coordinate of the axis specified in the previous option.

For each of the axes different than the one selected in the **Constant axis** drop-down list, the user needs to specify the following parameters:

- **Initial value** Initial value of the coordinate along each axis.
- **Total size** The total length of the grid of observation points along the axis.
- **Number of points** The number of observation points along the axis.

Note The spacing between each pair of consecutive observation points along an axis will be given by the formula **spacing = (total size) / (number of points - 1)**. This value will be shown next to the label "Distance btw. points".

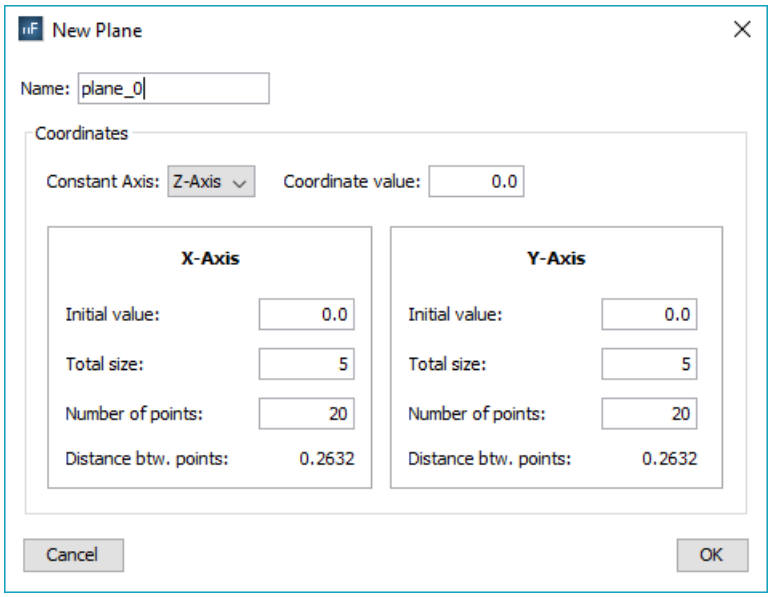


Figure 980: New Plane parameters

Point

The user can introduce the coordinates of the new point.

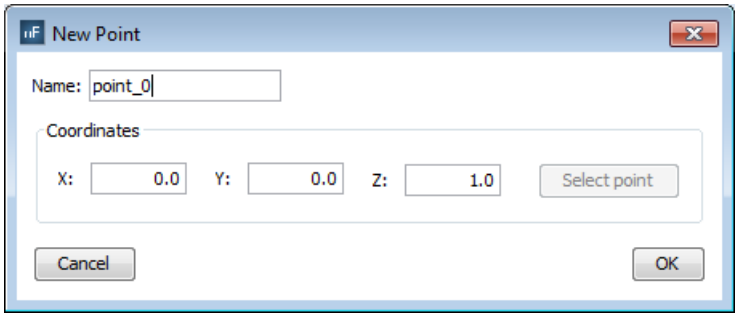


Figure 981: New Point parameters

Sphere

In order to build spheres, introduce the Center Point of the initial sphere. For the radius controls, the user can set the radius for the initial sphere using **Initial radius**. It is possible to create more than one sphere using the **Radius samples**. In that case, the radius increment between two consecutive spheres is controlled via **Increment radius**. Theta coordinates and phi coordinates let the user specify how many points to create from the sphere and whether to use the entire sphere curvature or just a slice of the sphere.

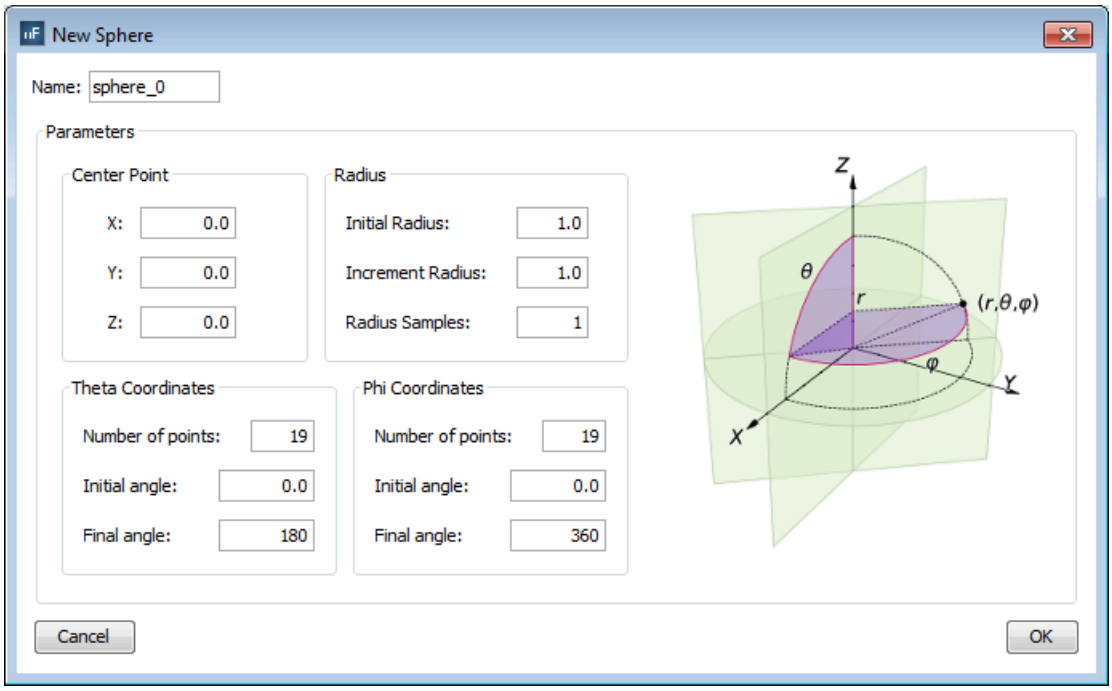


Figure 982: New Sphere parameters

10.10 Meshing Menu

This menu performs operations related to the meshing process. The following operations are available and covered in later sections of this chapter:

- Create Mesh this option is used to mesh the project.
- Visualize existing mesh this option is used to visualize a .msh file.
- View log this option is used to visualize the log that the mesher generates.

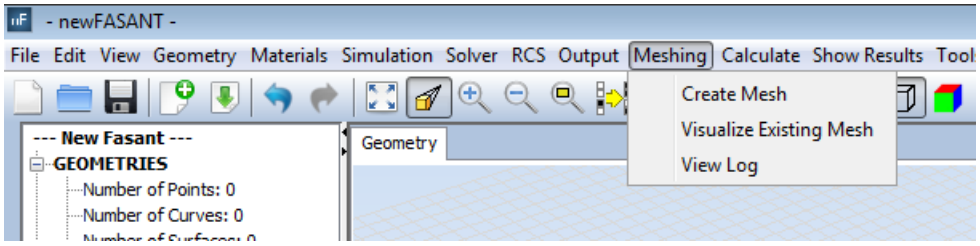


Figure 983: Meshing Menu

10.10.1 Create Mesh

When the Create Mesh option is selected, the following panel will be shown on the right side of the window:

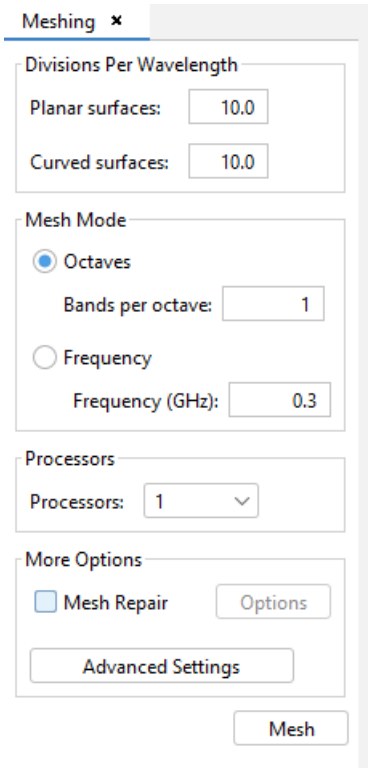


Figure 984: Create Mesh panel

- **Divisions per wavelength**: The user may specify different mesh density in planar and curved surfaces. Planar surfaces are contained in a single plane, otherwise surfaces are curved. Let **c** be the speed of light in the vacuum (meters per second), given a number of divisions **D** and a frequency **f**, the size of the generated elements **L** is given by:

L = lambda / D ; lambda = c / f

Number of bands per octave
: When a frequency swept is enabled, the meshing frequencies are different to the simulation ones. An automatic frequency ranges per octaves is performed, that depends on the initial frequency, the final frequency, and the number of bands per octave specified. The higher is this parameter, the more frequencies are considered for the meshing process.

Frequency
: Select this option to use the same frequency for meshing all the frequencies of the swept. The frequency must be specified in GHz.

Processors
: Use this to set the number of processors used for the meshing process.

More options

- **Mesh Repair** Select this option to evaluate and repair the generated mesh. It is recommended for analyzing complex geometries, specially whenever a good convergence is not achieved. Several problems will be studied and solved isolated and spurious elements are removed, and overlapped regions are repaired. The detection of this defects on the mesh depends on two parameters that are explained below, and can be edited by clicking on **Options** button. Elements wit smaller area than the minimum allowed, isolated elements, or parallel elements closer than a minimum distance are deleted from the output mesh. The minimum area and distance depends on the **wavelength**, the number of **divisions per wavelength** selected, and the **Area** and **Border factors**, and they are determined as:

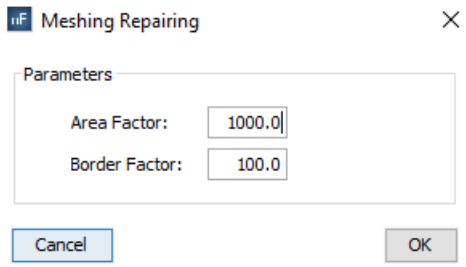


Figure 985: Meshing Repairing options

$$MinimumArea = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)^2}{AreaFactor_{MeshingRepairParameter}} \quad (2)$$

$$MinimumDistance = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)}{BorderFactor_{MeshingRepairParameter}} \quad (3)$$

- Click on **Advanced Settings** button to edit more complex meshing parameters. The available options is detailed in next subsection.
- **Mesh** click on this button to save the parameters and start the meshing process.

Advanced Meshing Settings

When the Advanced Settings button is pressed in the Meshing tab, a window where several advanced parameters on how the messing process work is displayed. The following sections cover these parameters.

General

The General tab has generic advanced parameters that are used by the mesher.

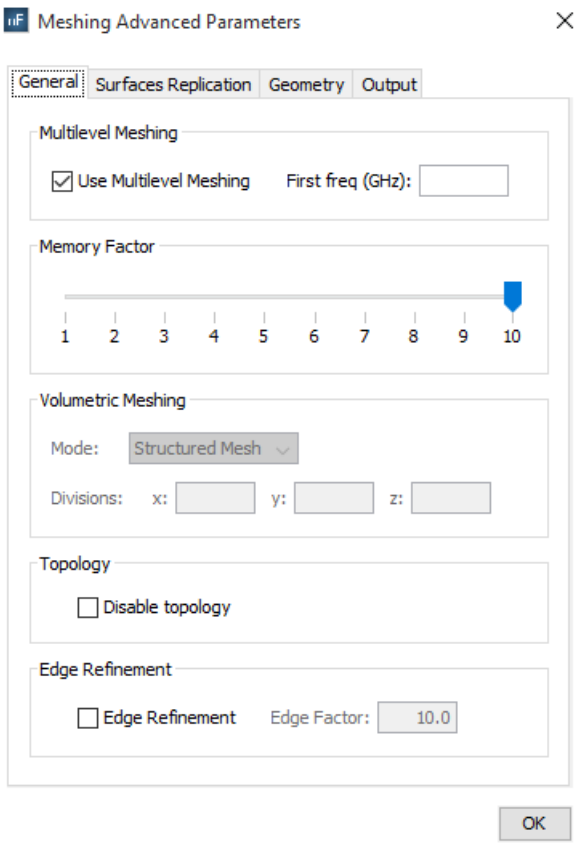


Figure 986: Meshing advanced parameters general tab

- **Multilevel Meshing**. Enable this option to generate the mesh automatically in several steps. This option is usually more efficient (in terms of runtime) than common mesh generation, so it is selected by default. However, minor differences may appear between the meshes obtained with and without multilevel mode. The frequency of the first step may be specified by the user in the **First freq** field, but when this field is empty, the first frequency is automatically computed.

- **Memory Factor**. This option allows a reduction of the memory resources required in the mesh generation process. The memory allocation is an automatic feature in this process, so the highest one is selected by default to ensure that the mesh will be successfully obtained.
- **Volumetric Mode**. This option allows to change the algorithm used in the mesh generation process of dielectric objects (defined as Volumes). Two different modes are available:
  - **Structured Mesh** All elements of the volumetric mesh are perfect parallelepipeds. This method is an approximation of the real shape of the dielectric objects by simple cubes. Some parts of the mesh can be out of the volume and other regions can have incomplete regions, but the average volume of the mesh is very similar to the volume of the original dielectric objects. This algorithm is the fastest one.
  - **Conformed Mesh** Most of elements of the volumetric mesh are perfect parallelepipeds, but only in the completely inner regions of them. The inner structured mesh is joined with the boundaries of the volume by using hexahedrons of variable shapes. This method is more accurate than the Structured Mesh, but is also slower and irregular elements may appear in specific cases.

  Different divisions per wavelength for the X, Y and Z dimensions may be considered for meshing the volumetric objects.
- **Topology**. If the electrical continuity between the surfaces have to be broken, the option Disable topology must be selected. Note that the accuracy may be reduced when the electrical continuities are not correctly analyzed because virtual fissures are introduced.
- **Edge Refinement**. This option allows enabling the modelling of border effect in the mesh generation. The Edge Factor field represents the portion of size of the final elements to model the border effect. It is only enabled when Edge Refinement is selected.

Note that the memory factor is limited by the available memory in the computer. If the memory resources exceed the available memory, a dialog will appear to suggest decreasing this factor:

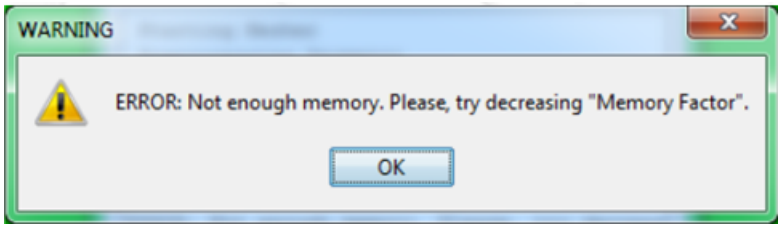


Figure 987: Not enough memory warning dialog.

Surfaces Replication

Surfaces Replication contains parameters that set the behaviour for the mesher when working with surfaces.

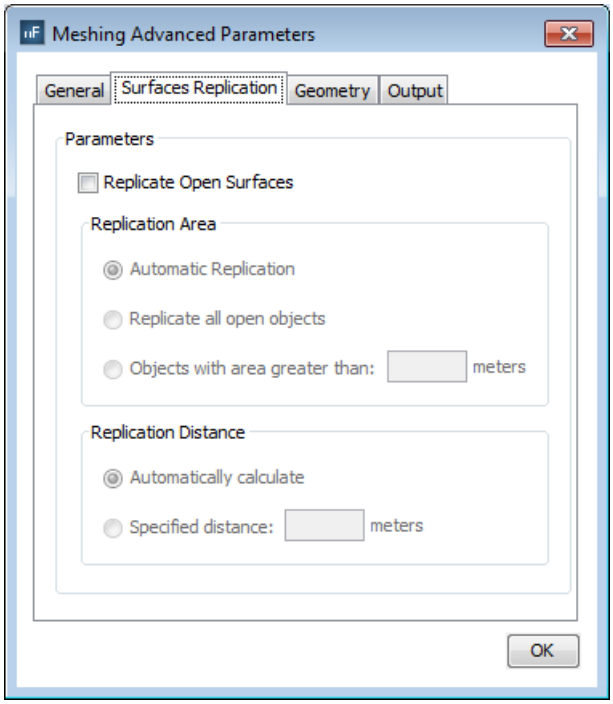


Figure 988: Meshing advanced parameters surface replication tab

There is only one option called Parameters in this tab that is composed by some sub-options. By marking the Replicate Open Surfaces option, a parallel layer of open objects is generated in the mesh generation process. If replication is marked, the elements of all the open objects are replicated to an automatic frequency-dependent distance from the original ones. By clicking on the advanced button, the window shown in the next Figure will appear where additional features can be configured.

- **Replication Area**. The criterion to replicate open objects can be edited.
  - **Automatic Replication**. It is the default option. Let **div_plane** and **div_curve** are the number of **divisions per wavelength** for **planar** and **curved** surfaces, any object with a total area greater than:

$$a \geq 10 \cdot \left(\frac{lambda}{maximum(div_{plane}, div_{curve})} \right)^2 \tag{4}$$

  - **Replicate all open objects**. Any object is replicated, by default.
  - **Replicate objects with area greater than**. To specify manually the threshold of the minimum total area of the objects to be replicated, in square meters.
- **Replication Distance**. The distance between the original objects and their parallel replicas can also be edited.
- **Automatically calculate**. By default, the distance is given by the following expression:

$$d = 0.03 \cdot \left(\frac{lambda}{maximum(div_{plane}, div_{curve})} \right) \tag{5}$$

  - **Specified distance**. To specify manually the desired distance, in meters.

Geometry

The Geometry tab has parameters related to how the geometry is studied during the meshing process.

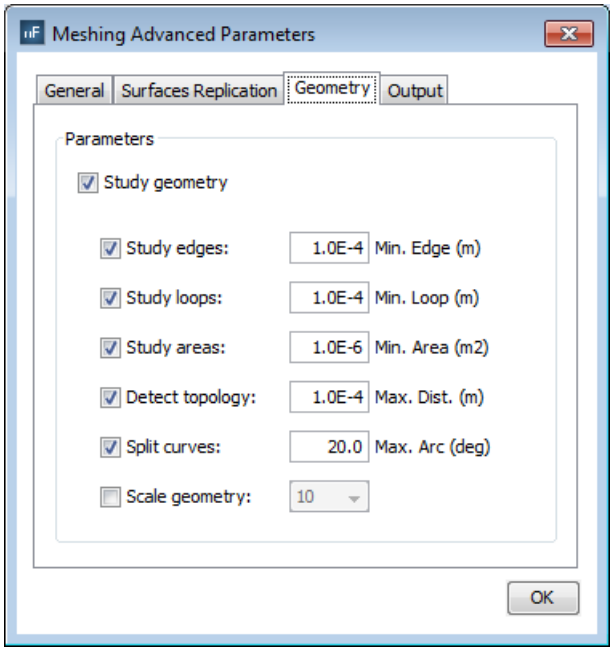


Figure 989: Meshing advanced parameters geometry tab

- **Study Geometry**. To pre-process the input geometry, evaluating some features such as its sizes or electrical continuities.
- **Study Edges**. To delete the edges of the input surfaces that are shorter than the Minimum Edge parameter (in meters), extending its adjacent edges. This option is only enabled when the Study Geometry button is enabled.
- **Study Loops**. To delete the loops of the input surfaces that are shorter than the Minimum Loop parameter (in meters). This option is only enabled when the Study Geometry button is enabled.
- **Study Areas**. To delete the input surfaces that are smaller than the Minimum Area parameter (in square meters). This option is only enabled when the Study Geometry button is enabled.
- **Detect Topology**. To detect automatically electrical continuity between neighboring surfaces that are very close but have not been modeled with precise continuity. The maximum separation allowed to set the electrical continuity is the Maximum Distance parameter (in meters). This option is only enabled when the Study Geometry button is enabled.
- **Split Curves**. To divide the curved borders of the input surfaces that have a curvature greater than the Maximum Arc parameter (in degrees). This option is only enabled when the Study Geometry button is enabled.
- **Scale Geometry**. To scale internally the geometry during the meshing process. When the mesh generation finishes, the output mesh has the same sizes as the original geometry. This option is useful for meshing very small structures. This option is only enabled when the Study Geometry button is activated.

Output

This option is no longer within the **Meshing Advanced Parameters** window. It has been moved to the main **Meshing** window, as it is recommended whenever a good convergence is not achieved.

10.10.2 Visualize Existing Mesh

This option can be chosen by the user to load and render a mesh file. Mesh files are generated by the mesher provided with newFASANT and usually end by .msh.

Select Visualize Existing Mesh item from the menu in order to render a mesh file. It will ask for a `.msh` file:

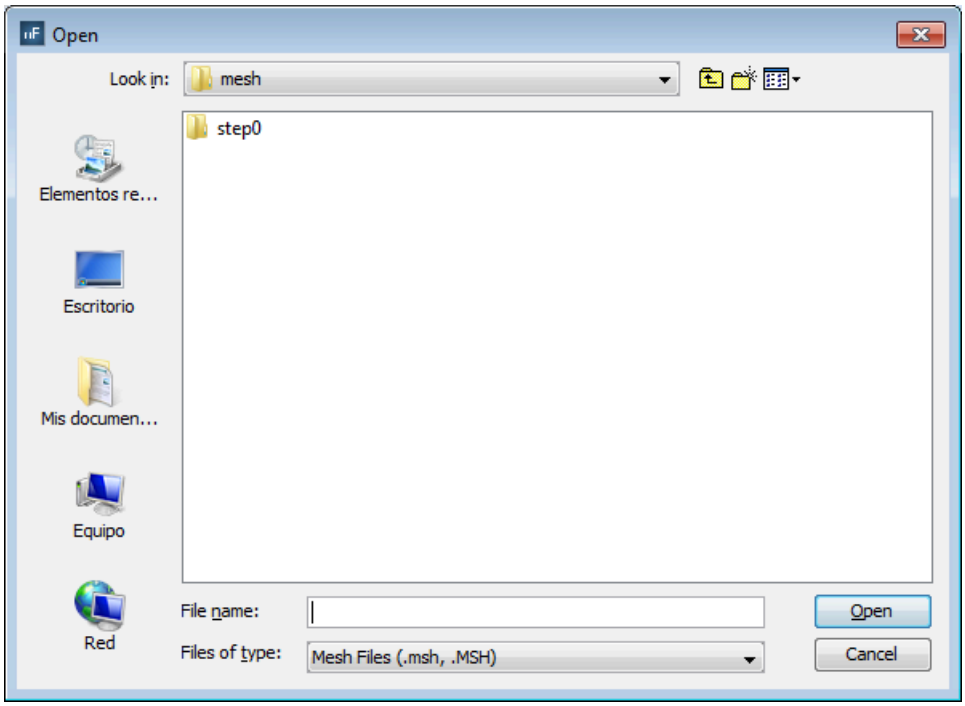


Figure 990: Open a Visualize Existing Mesh

Once a valid mesh file is selected, press OK to load it and render it using the viewer:

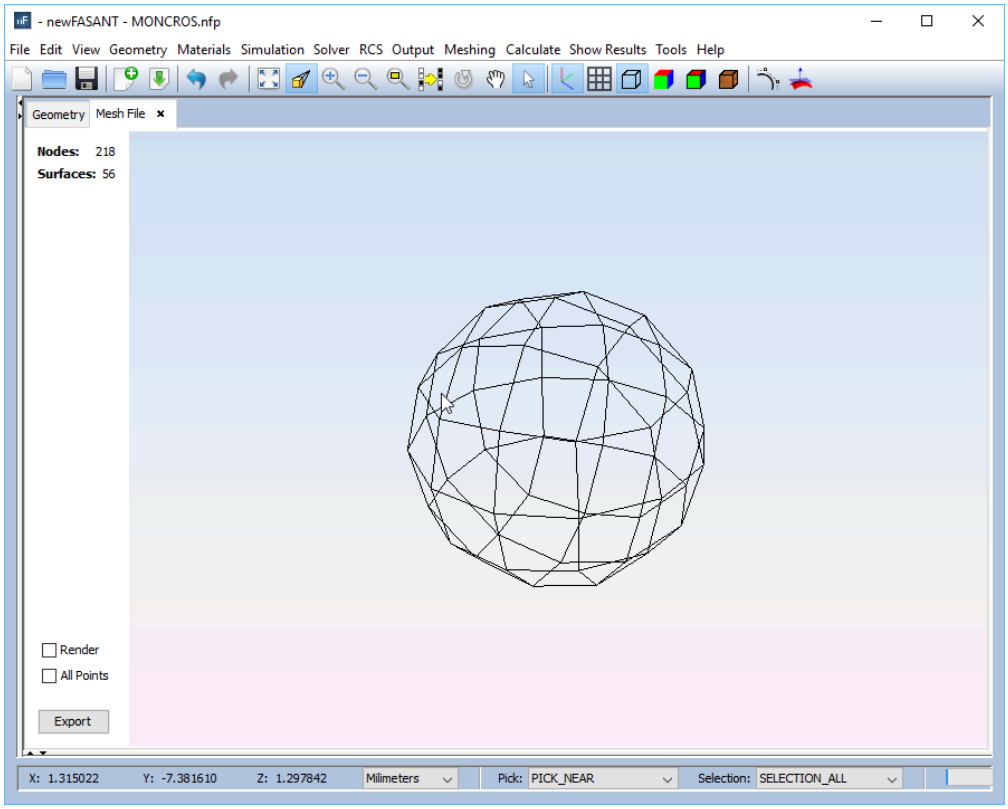


Figure 991: Mesh Viewer

The following two options appear as check boxes that can be enabled in the bottom left area of the panel:

- **Render**: if this option is disabled, the mesh will appear as a wireframe. Enable this check box to actually render a solid yellow mesh.
- **All Points**: selecting this option will increase the number of points displayed for the mesh.

It is possible to export the mesh pressing the Export button and selecting a folder where the mesh should be saved.

10.10.3 Visualize Mesh Log

During the meshing process, information about the meshing execution is generated and displayed in the log. This information can be manually viewed later by selecting this item from the menu. A tab will display the contents of the log file, which is also saved into mesh_log.txt. Press the **Save as...** button to save the log to a file.

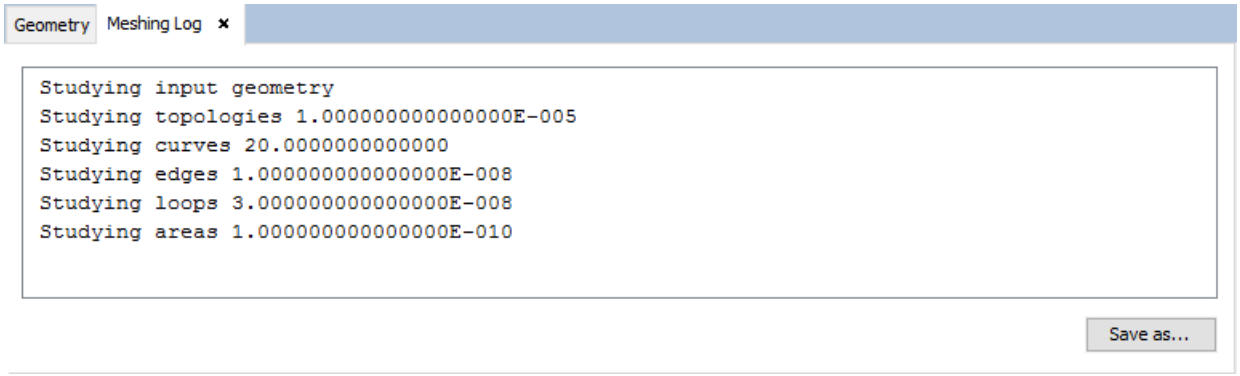


Figure 992: Mesh Log

10.11 Calculate Menu

This menu has options for executing a simulation after every parameter has been correctly defined and every possible geometry has been added to the project. Before executing the simulation, the user needs to make sure that every simulation parameter is enabled and well-defined. If something is wrong (e.g. the geometry is not meshed) a warning message will be displayed to let the user know that it is necessary to properly configure the simulation data.

10.11.1 Execute

This option runs the simulation. The user can set the number of processors the simulation will run on.

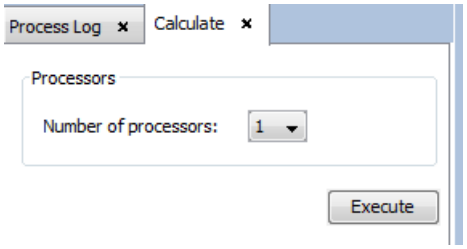


Figure 993: Calculate, execute option

Once the simulation starts, the following panel is shown:

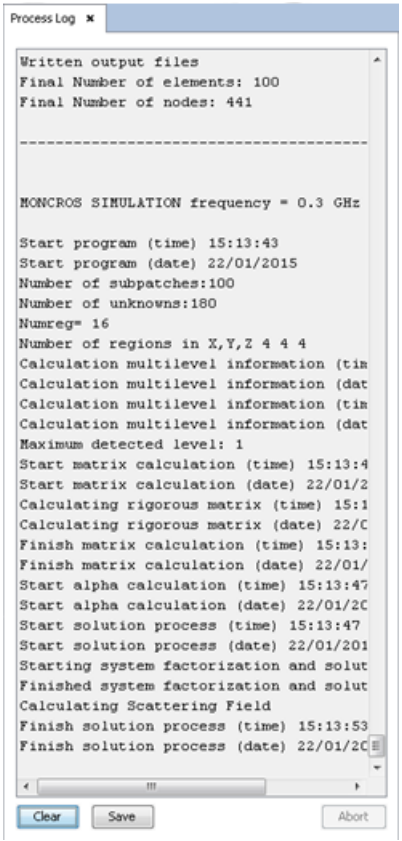


Figure 994: Calculate, process log

In this panel, the program shows the process log of the running simulation. It also offers the possibility of aborting the simulation and saving the contents of the process log to a text file.

10.12 Show Results Menu

This menu contains several options that lets the user view the results generated by the simulation process. There are different kinds of results, some of them presenting tables, some of them presenting numerical results, some of them presenting plots and some of them presenting 3D views where the user can see visual results like the charges and currents analysis through the objects.

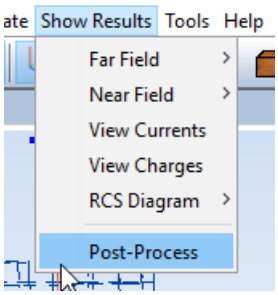


Figure 995: Show Results Menu

Most of plots show 2-D curves of a result field versus an input parameter. This type of graphics share aditional features of zoom, raxis and curves edition, and so on. To read more information about the chart options (when clicking on the rigth button over a chart) see Annex 1: Graphics Advanced Options.

10.12.1 Far Field

Using this menu it is possible to analyze the amplitudes in the Far Field through the observation directions previously defined in the Output section.

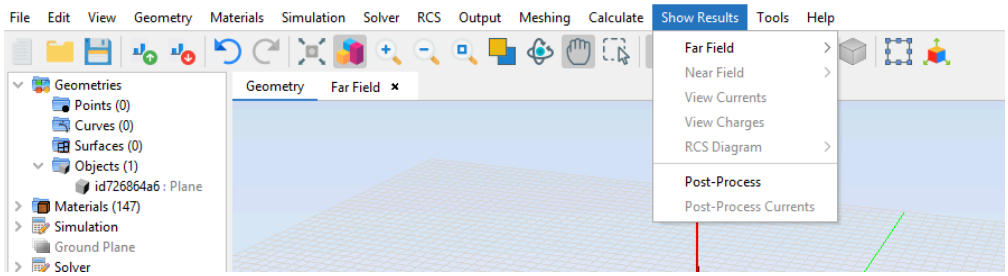


Figure 996: Far Field Menu

The following options are available:

- **View Cuts**: Displays a plot that relates the amplitude against a theta cut or a phi cut.
- **View Cuts by Frequency**: Displays a plot that relates the amplitude against a frequency sweep.
- **View Cuts by Step**: Displays a plot that relates the amplitude against a parametric step.
- **View Text Files**: Displays the numeric results gathered by the simulation process.

View Cuts

This option is used to analyze information about the calculated 2D Far Fields using the observation directions previously created in the Output menu. When selecting this option, a panel similar to the one presented in the figure will be displayed:

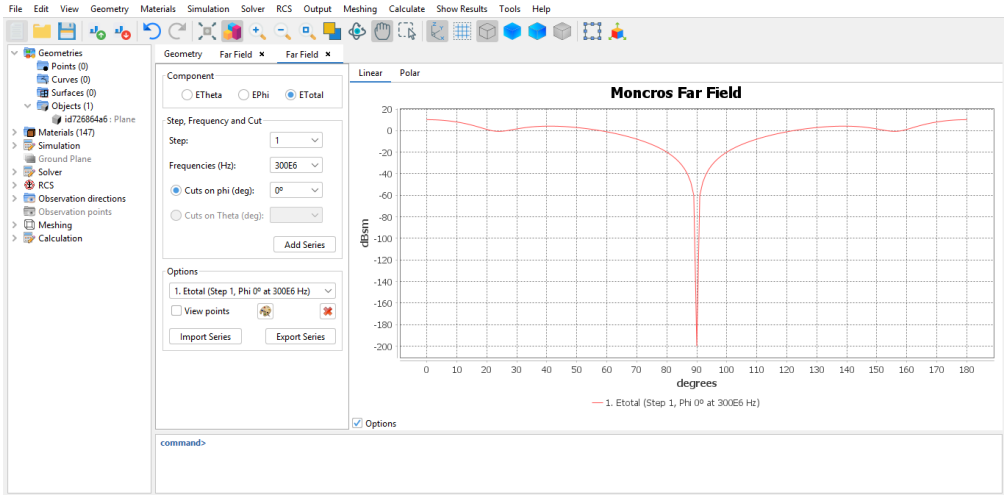


Figure 997: View Cuts panel. Linear plot

This command plots the amplitude or the phase of the RCS computed for each angular cut. Several cuts and frequencies can be plotted in the same graphic. To plot a graph, the user must select the RCS component, the frequency and the cut. The user can press the button “Add Series” to plot the new graph. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete the selected trace of the list with the “Remove Series“. The display option allows to change the color of the series and display points. The buttons “Import Series“ and “Export Series“ are used for importing and exporting the selected series in “List of Series“ to a datatext file.

The user can also switch between the linear graph and the polar graph to visualize far field results.

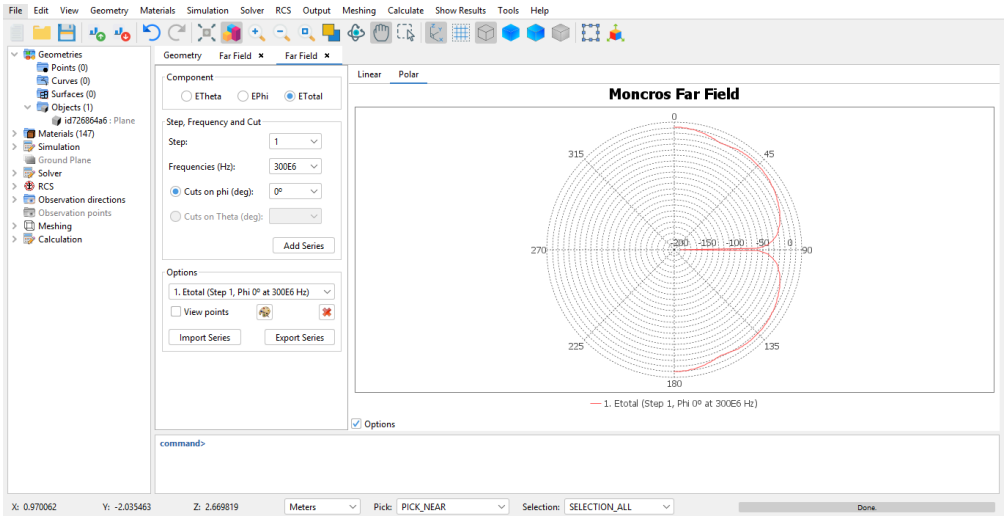


Figure 998: View Cuts panel. Polar plot

View Cuts By Frequency

This kind of plot can be used to compare the amplitude measurements made by the simulation process for a particular component, step, cut and point in the sweep, against a simulation frequency. This option is useful when the user has selected a frequency sweep in the Simulation Settings, in order to inspect how the measurements for a particular observation point change through a frequency sweep.

When this option is chosen from the Show Results > Far Field > View Cuts By Frequency menu, the panel presented in the figure below will appear.

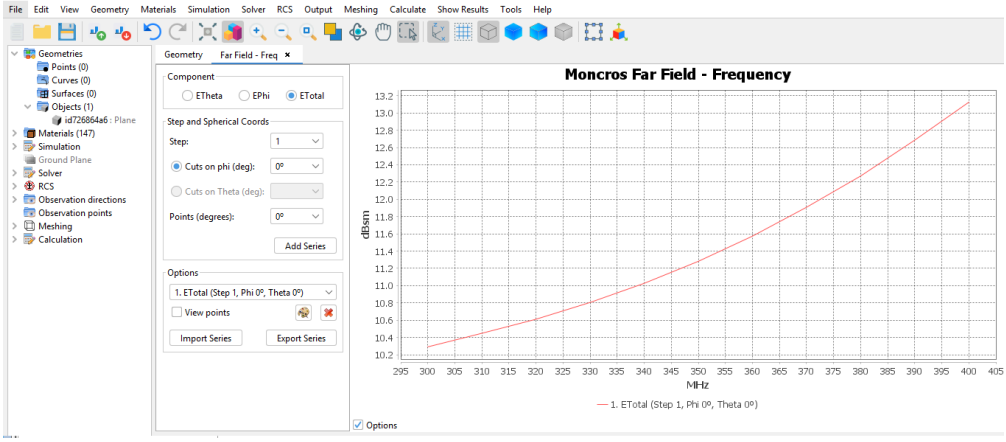


Figure 999: View Cuts By Frequency panel

Similar to the behaviour of the standard Far Field chart, this panel displays a plot with the results, and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, step, and observation point in a phi or theta cut, against

every possible frequency that has been simulated. More series can be added, by selecting component, step and spherical coordinates in the control panel that appears leftside to the plot.

- Component the plane wave component to inspect. Select one of the possible values Eθ, Eφ or Et.
- Step the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- Theta cut / Phi cut select one of the theta cuts or phi cuts that were previously added to the project.
- Point select a particular observation point in the cut selected before.

The selected series can be added to the plot by pressing the Add Series button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the Change Colors. To remove a series from the plot representation, press the Remove series button.

Series can be imported and exported. To save a series press the Export series button. Later it is possible to load previously exported series by pressing the Import Series button and locating the file where the results were saved in.

View Cuts By Step

This kind of plot lets the user analyze RCS measurements made by the simulation proves for a particular component, frequency, theta or phi cut and observation point against the parametric steps defined in the project. This option is useful when the user has parametrized a simulation by adding parameters and making objects such as geometries depend on the parameters themselves.

When this option is chosen from the Show Results > Far Field > View Cuts by Steps menu, the panel presented below will appear:

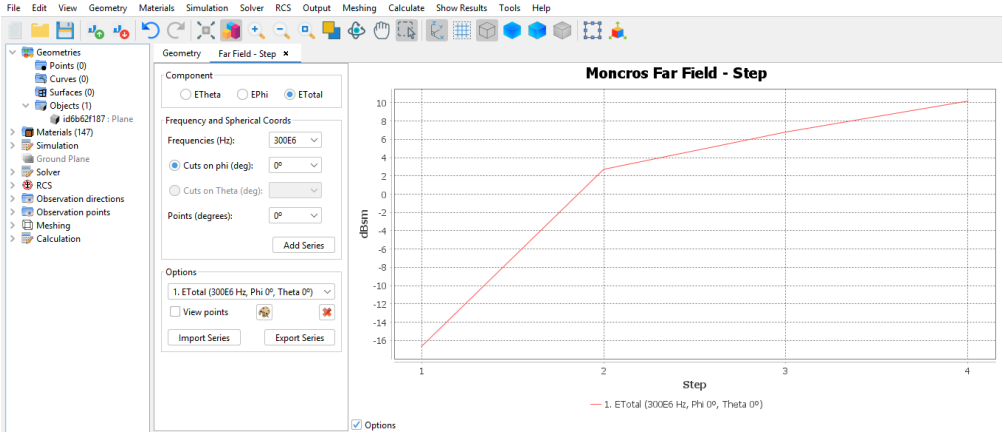


Figure 1000: Far Field by Step

Similar to the behaviour of the standard Far Field chart, this panel displays a plot with the results and a control panel where the user can add, modify and remove series that are then displayed in the chart. A default series will be added automatically to the plot, but the user can add new series by modifying the values in the control panel and pressing Add Series for adding the series to the panel. The following settings can be modified:

- Component the plane wave component to inspect. Select one of the possible values Eθ, Eφ or Et.
- Frequency if a frequency sweep has been made, multiple frequencies may be available for selection.
- Cuts on phi / Cuts on theta select one of the cuts that were previously added as observation points to the project.
- Point select one of the samples made for this observation direction.

The selected series can be added to the plot by pressing the Add Series button. It is also possible to modify the look and delete series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the Change Colors. To remove a series from the plot representation, press the Remove series button.

Series can be imported and exported. To save a series press the Export series button. Later it is possible to load previously exported series by pressing the Import Series button and locating the file where the results were saved in.

View Text Files

This option lets the user inspect the results of the Far Field RCS values in a tabular like structure. Selecting this option will display a panel where the user should select a step and a frequency. To use this section, select a parametric step from the list, and select the frequency to get the results from and press OK to load the data. A table will be presented with the values for this frequency and step. To save the results press the Export button below the table.

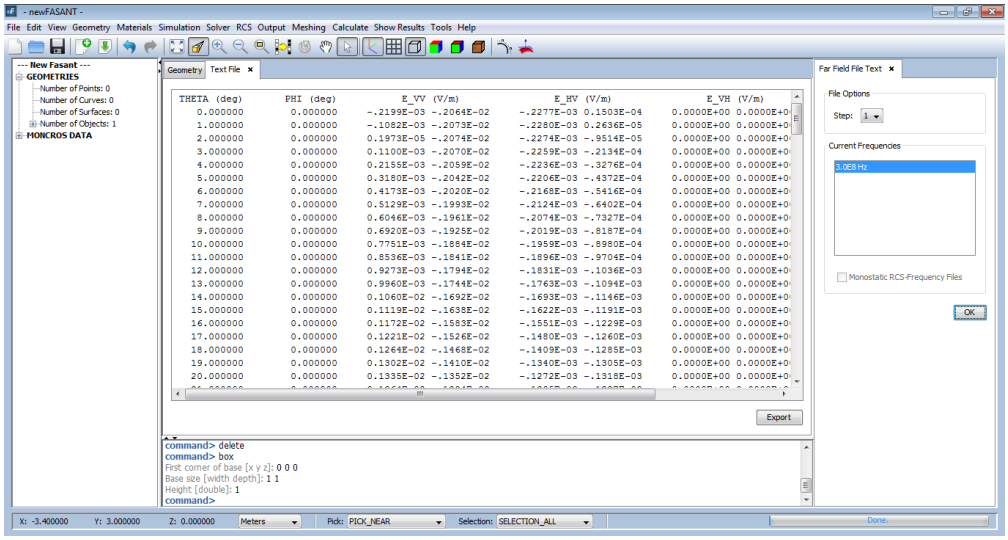
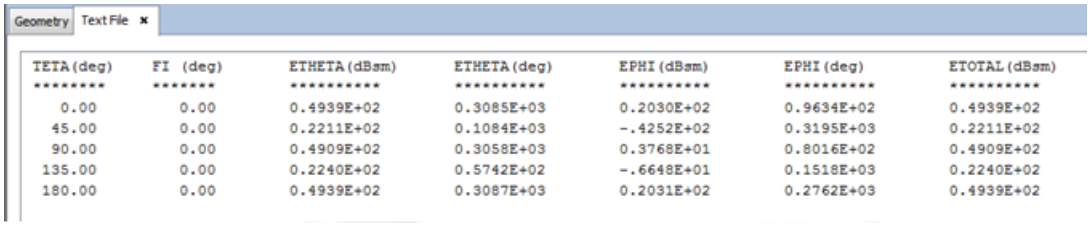


Figure 1001: View Text Files panel

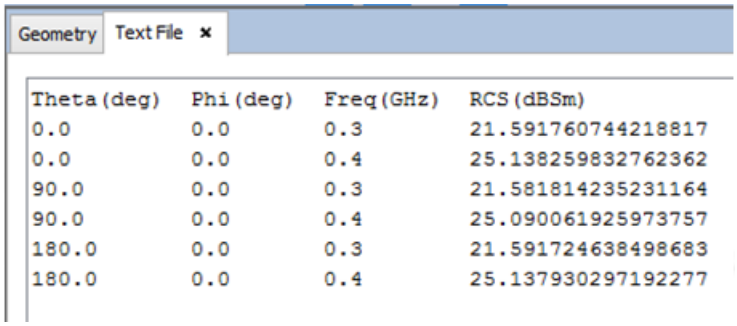
Select a frequency on the list and press Ok. The following tab in the central panel will show up with the results in text format:


Geometry | Text File ×

TETA (deg)	FI (deg)	ETHETA (dBsm)	ETHETA (deg)	EPHI (dBsm)	EPHI (deg)	ETOTAL (dBsm)
0.00	0.00	0.4939E+02	0.3085E+03	0.2030E+02	0.9634E+02	0.4939E+02
45.00	0.00	0.2211E+02	0.1084E+03	-.4252E+02	0.3195E+03	0.2211E+02
90.00	0.00	0.4909E+02	0.3058E+03	0.3768E+01	0.8016E+02	0.4909E+02
135.00	0.00	0.2240E+02	0.5742E+02	-.6648E+01	0.1518E+03	0.2240E+02
180.00	0.00	0.4939E+02	0.3087E+03	0.2031E+02	0.2762E+03	0.4939E+02

Figure 1002: Viewing results in text format

If Monostatic RCS is enabled, the "Monostatic RCS-Frequency Files" option appears. If this option is selected, the program generates the text file showing the result for the RCS-Frequency cuts, visualizing the RCS values for each combination of frequencies and angular cut.


Geometry | Text File ×

Theta (deg)	Phi (deg)	Freq (GHz)	RCS (dBSm)
0.0	0.0	0.3	21.591760744218817
0.0	0.0	0.4	25.138259832762362
90.0	0.0	0.3	21.581814235231164
90.0	0.0	0.4	25.090061925973757
180.0	0.0	0.3	21.591724638498683
180.0	0.0	0.4	25.137930297192277

Figure 1003: Viewing RCS-Frequency cuts data in text format

10.12.2 NearField

These settings are used to inspect the near field using the observation points previously defined in Observation points. It is necessary therefore to define at least one observation points structure in the corresponding panel at the Output menu. Observation points also require the RCS to be configured in bistatic mode.

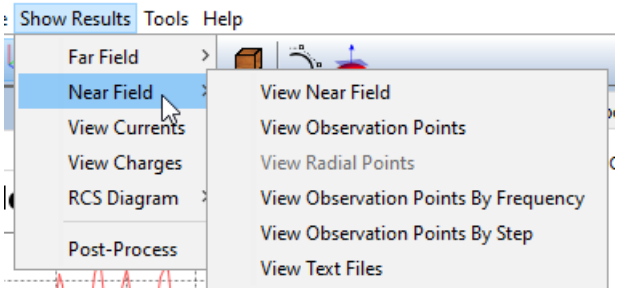


Figure 1004: Near Field menu.

The following menu items are available:

- View Near Field display a 3D View with the measurements for every observation point in the near field.

- View Observation Points display a plot with the measurements for each observation point in the near field.
- View Radial Points displays a radial plot with measurements made in the different radii for spheres.
- View Obervation Points by Frequency displays a plot with the measurements for a particular observation point against a frequency sweep.
- View Observation Points by Step displays a plot with the measurements for a particular observation point against parametric steps.
- View Text Files displays numerical results as given by the simulation process.

View Near Field

This option can be used to present in a 3D View the value for each user defined observation points.

After selecting this option, the following panel will be displayed.

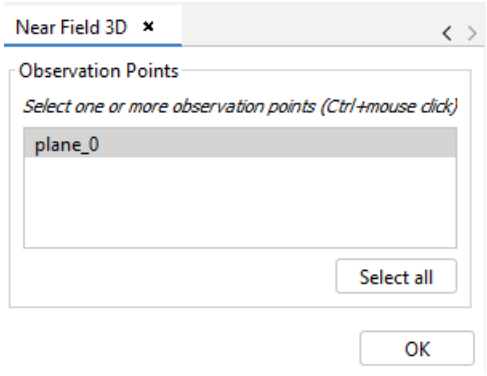


Figure 1005: View Near Field panel

The diagram shows, for every structure defined as an observation point, the near field measurement in every point of the structure. For each geometry, values will be rendered with a diferent color that is smoothed, creating a grid matrix where every observation point is assigned to a color. Colors are part of a measurement system where blue is asigned to the lowest values and red to the highest values.

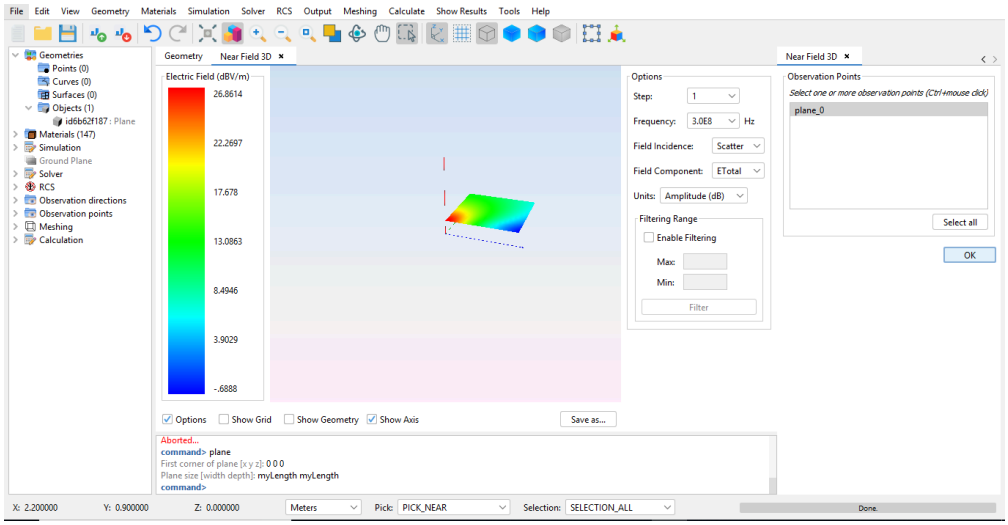


Figure 1006: Near Field diagram

The following options are available:

- **Incidence**: possible values are scatter incidence, direct incidence or total incidence.
- **Magnitude**: which units to use in measurements. Units affect to the color scale being used.
- **Field Component**: which field component to inspect, from either the E-field or the H-field, and whether to inspect the X, Y or Z component or the total field.
- **Filtering Range**: if filtering is enabled, a custom range can be set in the measurement system, setting maximum and minimum values in the rendering view to the values provided by the user.

Additionally, the following options are available at the bottom bar:

- **Options**: hides and shows the options panel that appears on the left.
- **Show Grid**: if this option is selected, a grid with every observation point will be overlayed to the rendered figure, making possible to precisely see which value has every observation point defined in the geometry.
- **Show geometry**: use this option to make the geometry visible or not.
- **Show Axis**: use this option to hide or show the axis.

It is also possible to export a graphical representation of the image. Press **Save as...** to save the image as a PNG file.

View Observation Points

This option shows the observation points as a plot.

Select the theta or phi cut using the controls and press OK. After selecting the desired options and pressing OK, the following screen will be shown:

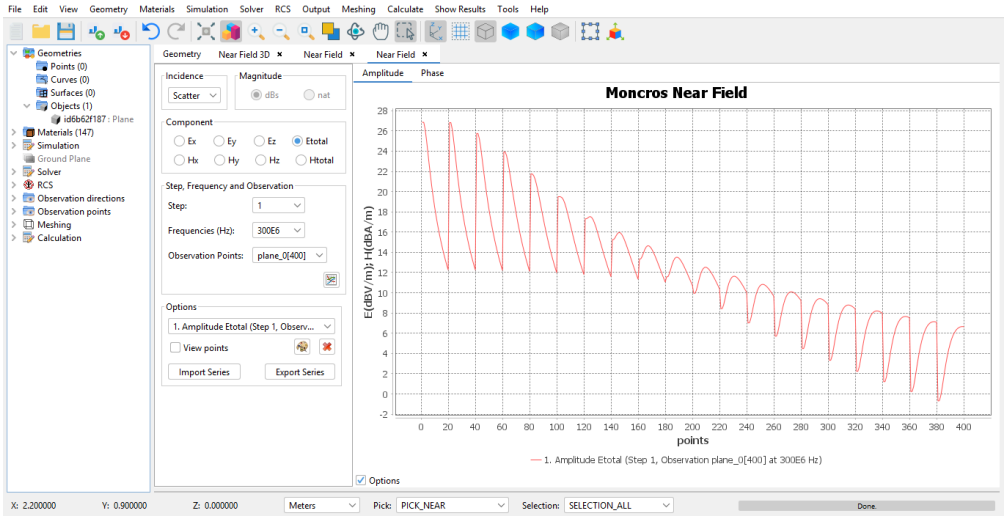


Figure 1007: View Observation Points panel

The controls on the left let the user add, modify and delete series. Using the visible controls the user can select a series to add using the following settings:

- **Incidence**: which kind of near field incidence to add scatter incidence, direct incidence or total incidence.
- **Field Component**: which field component to inspect, from either the E-field or the H-field, and whether to inspect the X, Y or Z component or the total field.
- **Step**: on simulation projects that make use of step, the user can select the step number to add.
- **Frequencies**: on simulation projects that make use of multiple frequencies, the user can select one particular frequency to add.
- **Observation points**: on simulation projects that make use of multiple observation point structures, the user can select one of the geometries.

Press the Add Series button to add the series to the graph. Note that if multiple cuts exists, a dialog similar to the one that appeared before will be presented to the user again asking to select a particular theta cut or phi cut.

It is possible to modify already existing series too. The user cans et the color for the series, which is useful when multiple series have been added to the plot. Select the series using the List of series control, and press Change colors to modify the color used to render the series. Additionally, a series can be removed if it's not relevant anymore to the purpose of the plot.

It is also possible to export already existing series and import series previously imported, which is useful for comparing series made using different projects or simulations.

View Radial Points

This option is available when observation spheres have been set. This plot is used to represent the different values for the Near Field on each radius sample defined in a sphere. Therefore, to make this plot useful, it is suggested to generate an observation sphere that makes use of multiple radius using samples.

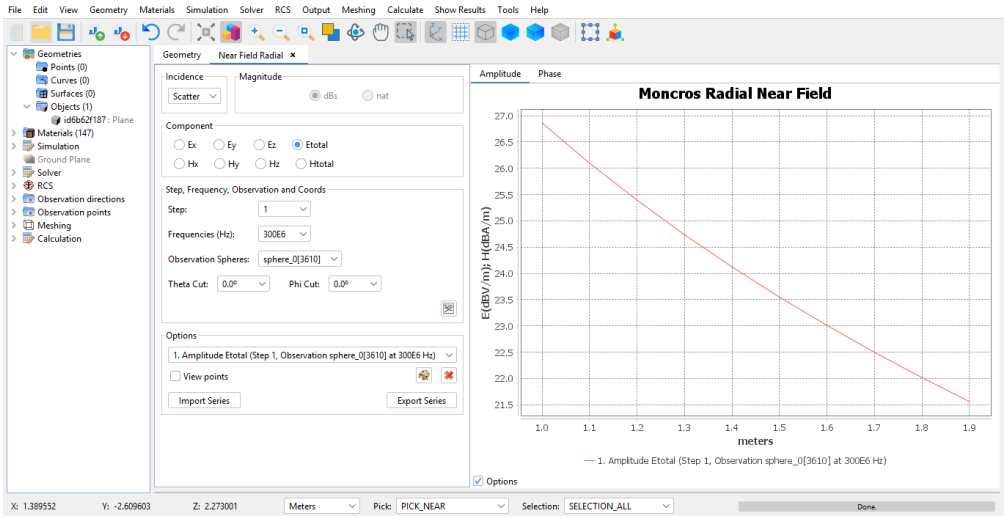


Figure 1008: Radial Near Field

When this option is selected, the panel previously depicted is shown. The following options are available to the left.

Incidence
: which kind of near field incidence to add scatter incidence, direct incidence or total incidence.

Component
: which field component to inspect, from either the E-field or the H-field, and whether to inspect the X, Y or Z component or the total field.

Step
: on simulation projects that make use of step, the user can select the step number to add.

Frequencies
: on simulation projects that make use of multiple frequencies, the user can select one particular frequency to add.

Observation points
: on simulation projects that make use of multiple observation point structures, the user can select one of the geometries

Theta and phi cuts
: when multiple observation directions are available, this option is used to select one particular cut.

Press the Add Series button to add the series to the graph.

It is possible to modify already existing series too. The user can set the color for the series, which is useful when multiple series have been added to the plot. Select the series using the List of series control, and press Change colors to modify the color used to render the series. Additionally, a series can be removed if it's not relevant anymore to the purpose of the plot.

It is also possible to export already existing series and import series previously imported, which is useful for comparing series made using different projects or simulations.

View Observation Points By Frequency

This option lets the user analyze the values for the near field through a frequency sweep. To make this section useful, a frequency sweep must have been previously calculated. The user can use this result view to generate plots that compare the values at near field for a particular Observation point in a given component and incidence and step through every frequency.

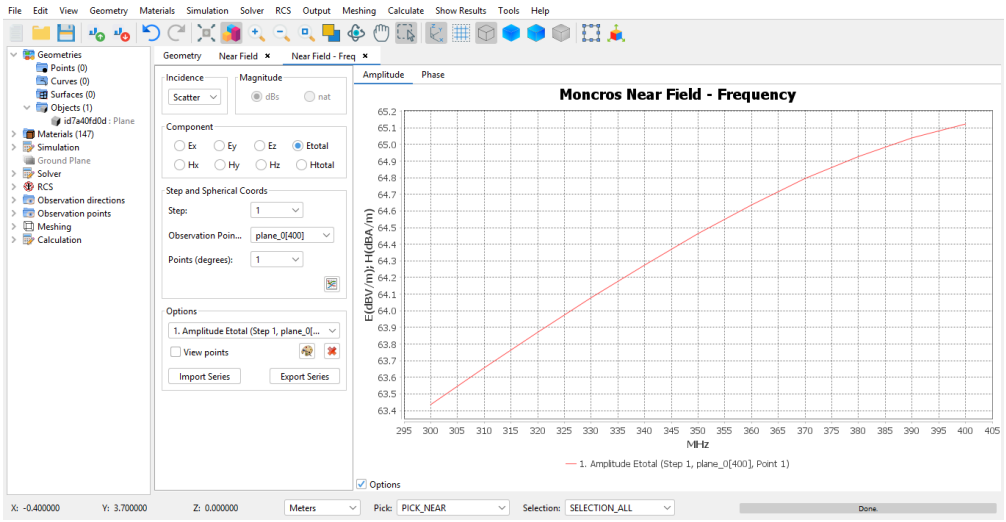


Figure 1009: Near Field - Frequency

The controls on the left let the user add, modify and delete series. Using the visible controls the user can select a series to add using the following settings:

Incidence
: which kind of near field incidence to add scatter incidence, direct incidence or total incidence.

Field Component
: which field component to inspect, from either the E-field or the H-field, and whether to inspect the X, Y or Z component or the total field.

Step
: on simulation projects that make use of step, the user can select the step number to add.

Observation points
: on simulation projects that make use of multiple observation point structures, the user can select one of the geometries.

Observation Point
: the user can select a particular observation point from the list.

Press the Add Series button to add the series to the graph.

It is possible to modify already existing series too. The user can set the color for the series, which is useful when multiple series have been added to the plot. Select the series using the List of series control, and press Change colors to modify the color used to render the series. Additionally, a series can be removed if it's not relevant anymore to the purpose of the plot.

It is also possible to export already existing series and import series previously imported, which is useful for comparing series made using different projects or simulations.

View Observation Points By Step

This option lets the user analyze the values for the near field through multiple steps. To make this section useful, at least two steps must exist in the project. The user can use this result view to generate plots that compare the values at near field for a particular Observation point given component, incidence and frequency through every step.

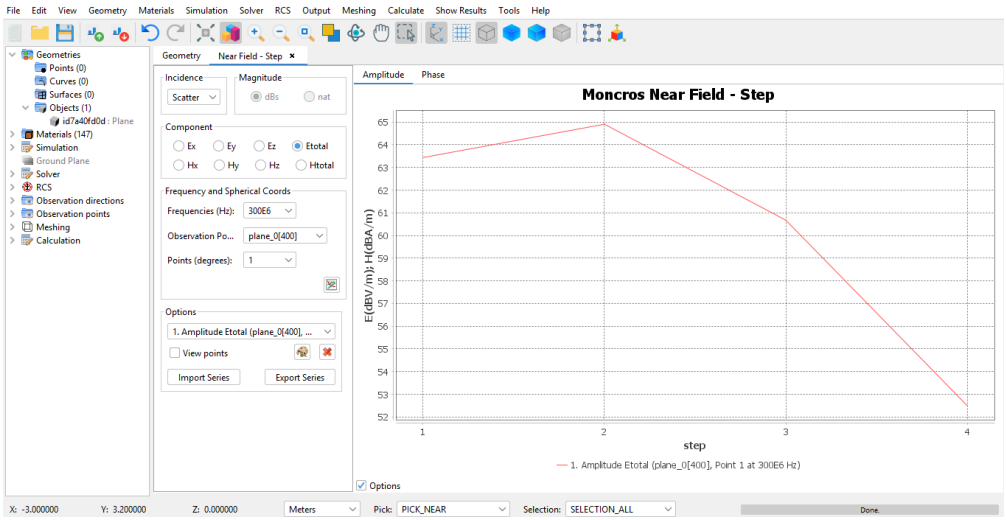


Figure 1010: Near Field - Step

The controls on the left let the user add, modify and delete series. Using the visible controls the user can select a series to add using the following settings:

Incidence
: which kind of near field incidence to add scatter incidence, direct incidence or total incidence.

Field Component
: which field component to inspect, from either the E-field or the H-field, and whether to inspect the X, Y or Z component or the total field.

Frequencies
: on simulation projects that make use of multiple frequencies, the user can select one particular frequency to add.

Observation points
: on simulation projects that make use of multiple observation point structures, the user can select one of the geometries.

Observation Point
: the user can select a particular observation point from the list.

Press the Add Series button to add the series to the graph.

It is possible to modify already existing series too. The user can set the color for the series, which is useful when multiple series have been added to the plot. Select the series using the List of series control, and press Change colors to modify the color used to render the series. Additionally, a series can be removed if it's not relevant anymore to the purpose of the plot.

It is also possible to export already existing series and import series previously imported, which is useful for comparing series made using different projects or simulations.

View Text Files

This option can be used to view the numerical results for the near field for a particular step and frequency. Select this menu item to display the Near Field Text panel. In this window, it is possible to select a step if more than one step exists, and one of the simulated frequencies. There will only be one frequency available for selection. However, if multiple frequencies are available as part of a frequency sweep simulation, they will be available for selection.

Press the OK button to display the numerical results. It is possible to export these results to a text file by pressing the **Export** button.

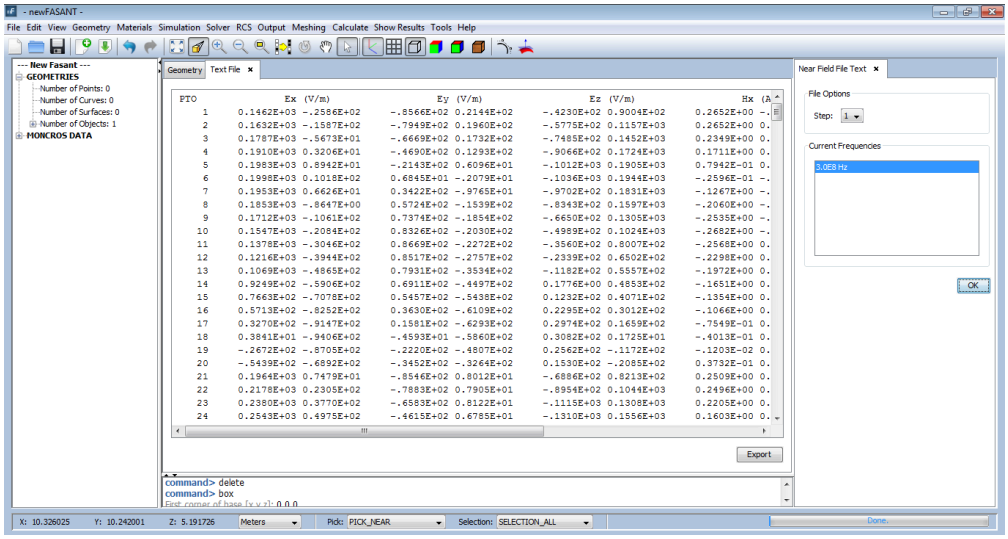


Figure 1011: Viewing near field values in text format

10.12.3 View Currents

This visualization option allows the user to see a representation of the induced currents. A color legend shows the magnitude of the current values corresponding to each color shown on the geometry (at the right side of the screen).

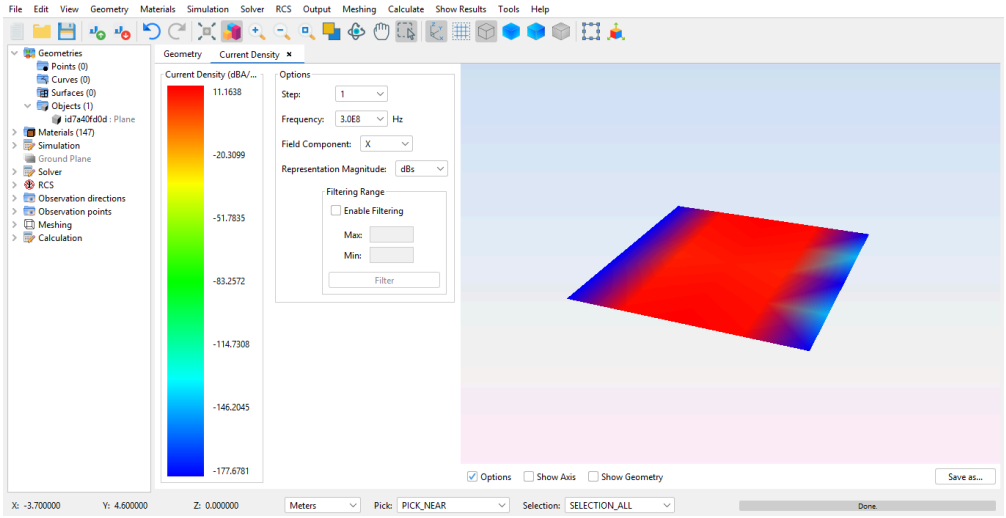


Figure 1012: Currents diagram

The following options are available at the left side:

- **Step and Frequency**: on projects that make use of multiple steps and frequencies, it is possible to select a particular step and frequency.
- **Magnitude**: the magnitude that is used to represent the values.
- **Field Component**: use this option to select which kind of field values to represent. Either the total field or particular X, Y or Z values can be selected.
- **Filtering range**: by default the values are assigned a color scale where blue is the lowest value and red is the highest value. It is possible to change this to use a custom filter scale by enabling this option and setting the minimum and maximum value.

Additionally, the following options are available at the bottom bar:

- **Options**: hides and shows the options panel that appears on the left.
- **Show geometry**: use this option to make the geometry visible or not.
- **Show Axis**: use this option to hide or show the axis.

It is also possible to export a graphical representation of the image. Press **Save as...** to save the image as a PNG file.

10.12.4 View Charges

This visualization option allows the user to see a representation of the charge density. A color legend shows the magnitude of the charge values corresponding to each color shown on the geometry (at the right side of the screen).

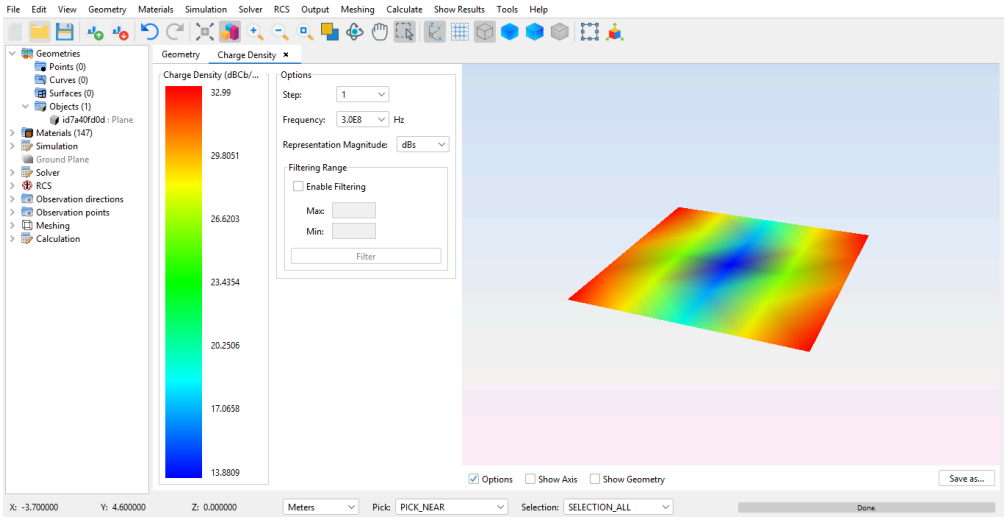


Figure 1013: Charges diagram

The following options are available at the left side.

- **Step and Frequency**: on projects that make use of multiple steps and frequencies, it is possible to select a particular step and frequency.
- **Magnitude**: the magnitude that is used to represent the values.
- **Filtering range**: by default the values are assigned a color scale where blue is the lowest value and red is the highest value. It is possible to change this to use a custom filter scale by enabling this option and setting the minimum and maximum value.

Additionally, the following options are available at the bottom bar:

- **Options**: hides and shows the options panel that appears on the left.
- **Show geometry**: use this option to make the geometry visible or not.
- **Show Axis**: use this option to hide or show the axis.

It is also possible to export a graphical representation of the image. Press **Save as...** to save the image as a PNG file.

10.13 Tools Menu

For information about the Tools menu, refer to Tools.

10.14 Help Menu

For information about the Help menu, refer to Help Menu.

10.15 Training Examples

10.15.1 Example 1

Step 1

First, start newFASANT.

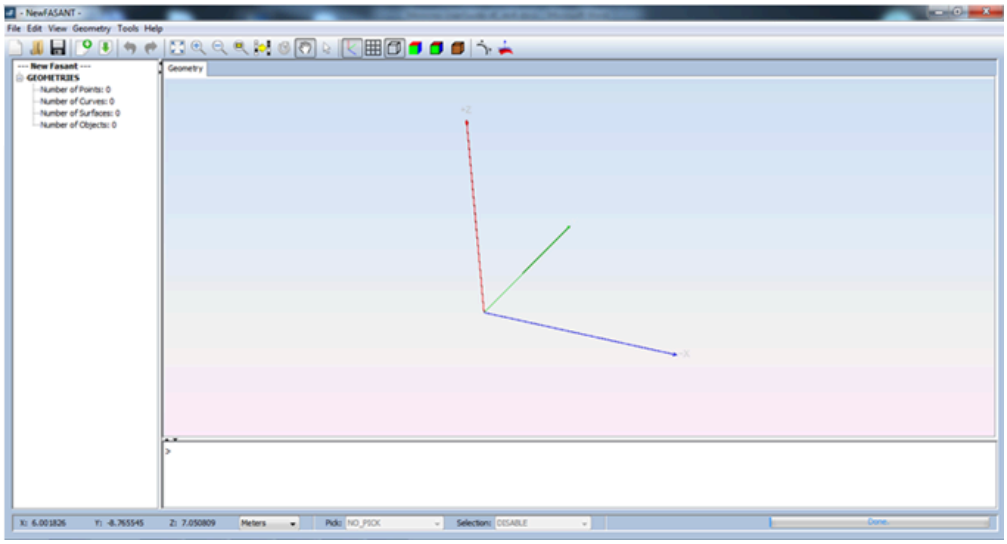


Figure 1014: Starting window

Step 2

Create a new newFASANT project. To do this, select the File → New option (alternatively, the press Ctrl +N).

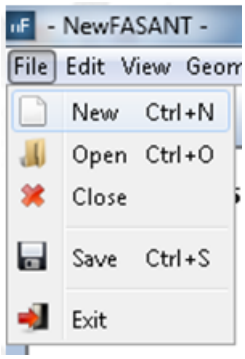


Figure 1015: File menu

Step 3

After selecting the New option, a list of the available modules is presented. The user needs to select a module from the available module list. Select MONCROS for this training example.

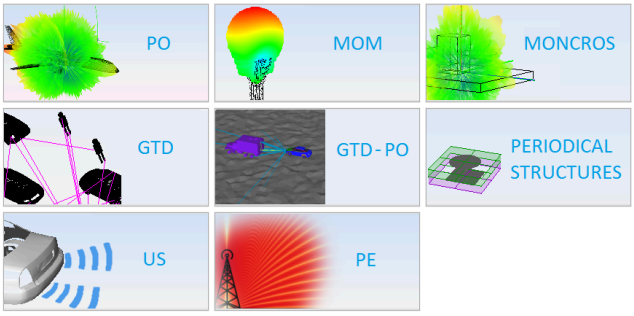


Figure 1016: Module selection screen

Step 4

Select the Geometry → Surface → Plane option.\

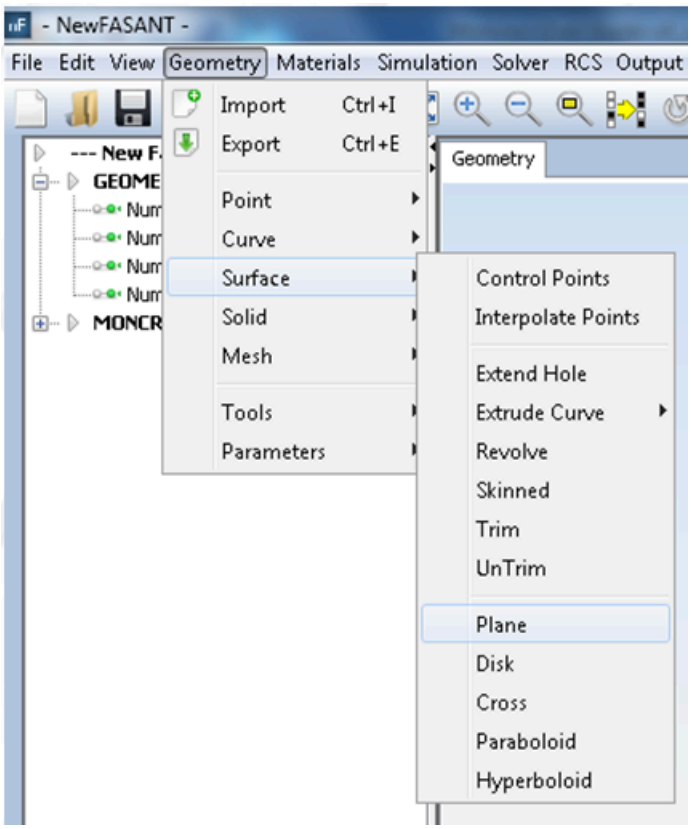


Figure 1017: Geometry → Surface menu

Step 5

The command line panel requires some parameters at this point. Enter "-0.5 -0.5 0" as the first parameter and "1 1" as the second parameter so we create a 1x1 plane centered at the origin.

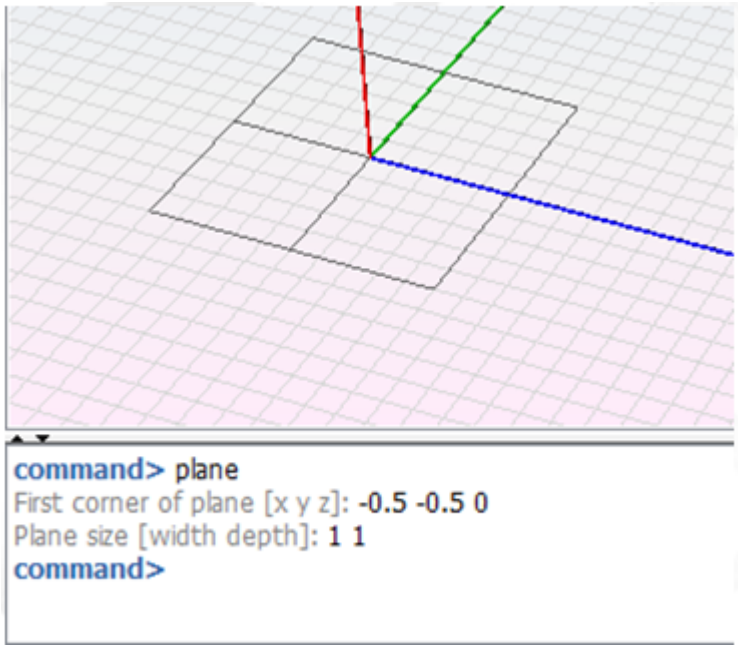


Figure 1018: Creating a 1x1 plane

Step 6

Select the simulation parameters option.

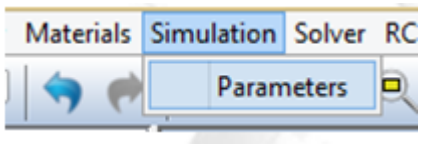


Figure 1019: Simulation menu

Step 7

Select a frequency of 0.3 GHz (default options) and press Save.

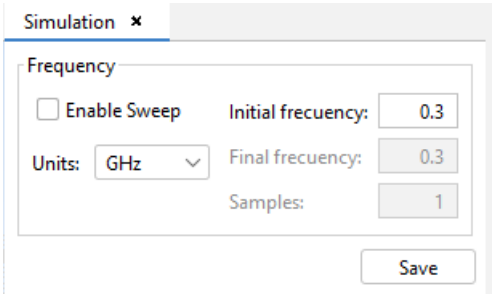


Figure 1020: Simulation options

Step 8

Select the Solver → Parameters option:

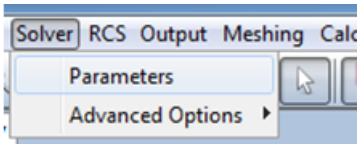


Figure 1021: Solver menu

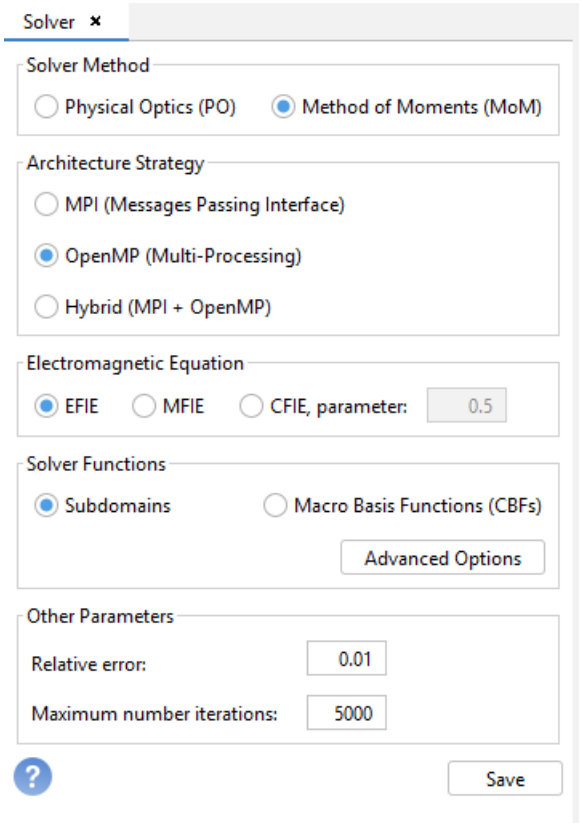


Figure 1022: Solver parameters

Check the default parameters as the follow figure.

Click on the “Advanced Options” button to set up other solver parameters:

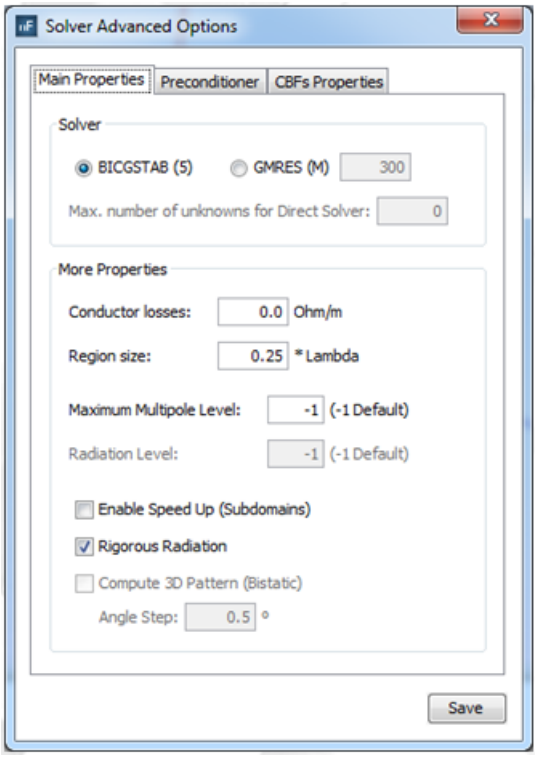


Figure 1023: Advanced parameters of solver, 'Main Properties' tab

Save the values for the advanced parameters of the solver.

Step 9

Select the RCS → Parameters option:

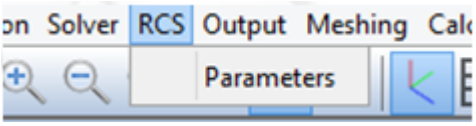


Figure 1024: RCS menu

Step 10

Select Monostatic RCS and leave the default parameters. Press the Save button.

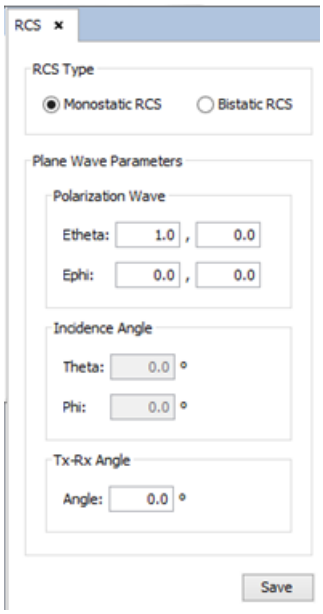


Figure 1025: RCS parameters

Step 11

Select Output → Observation directions.

Figure 1026: Output menu

Step 12

Leave the default observation directions (0.0 phi cut with theta angle varying between 0.0 and 180.0 using 181 samples).

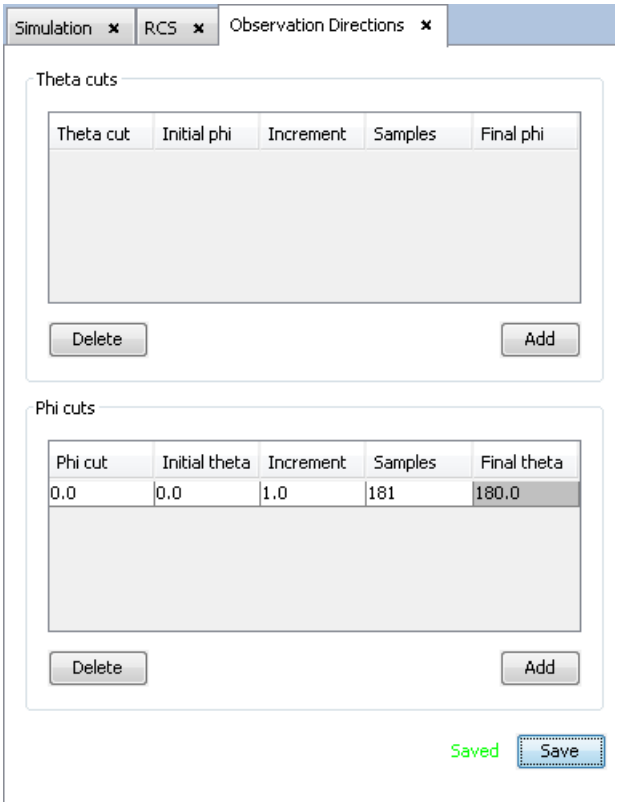


Figure 1027: Observation directions options

Step 13

Before running the simulation, select the Meshing → Create Mesh option.

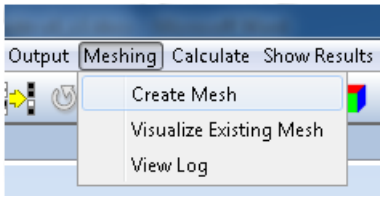


Figure 1028: Meshing menu

Step 14

Set 10 divisions on planar and curved surfaces, set Processors to 1 and the default number of bands per octave. Press the Mesh button.

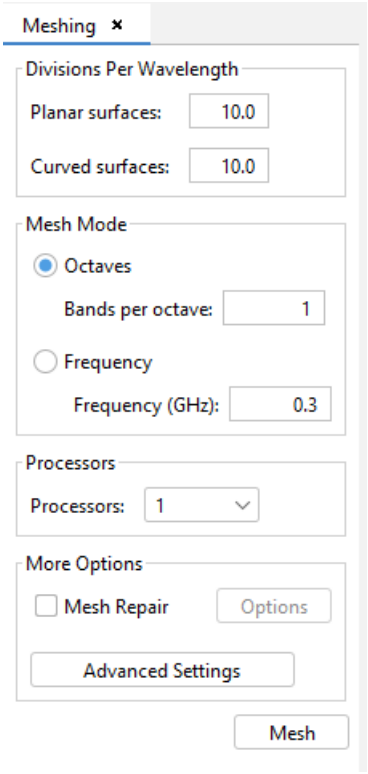


Figure 1029: Meshing options

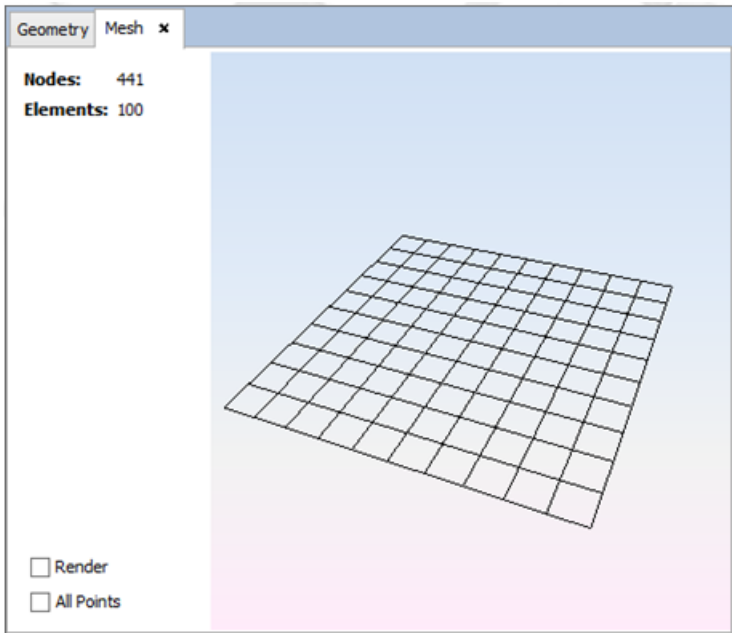


Figure 1030: Mesh visualization

Step 15

Select Calculate → Execute and set the number of desired processors to run the simulation. Press the Execute button to start the calculation.

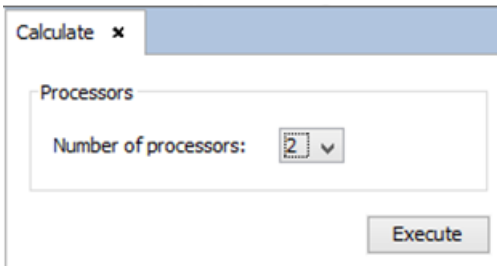


Figure 1031: Calculate options

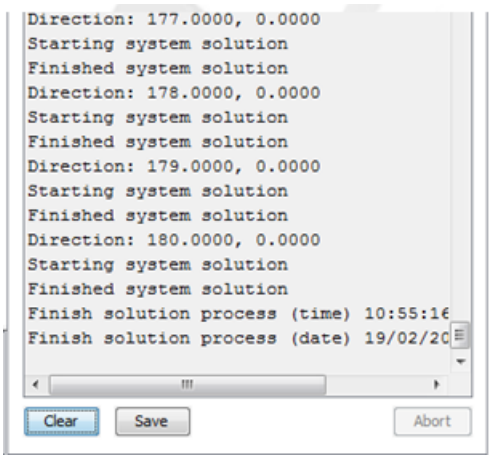


Figure 1032: Process Log showing the progress of the calculation

Step 16

When the simulation finishes (the progress bar on the bottom-left corner displays "Done"), we are able to see the results. Select the Show Results → Far Field → View cuts option to show the RCS plot.

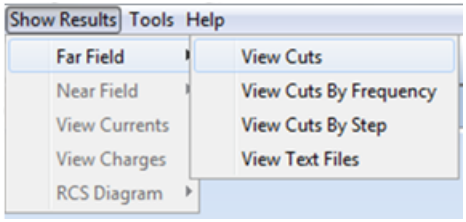


Figure 1033: Show Results → Far Field menu

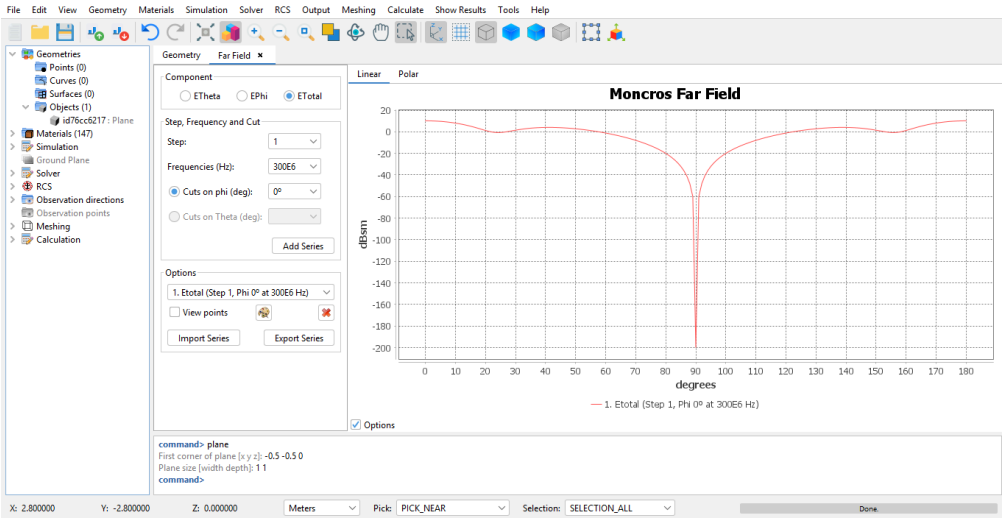


Figure 1034: RCS graphic

Step 17

Select Show Results → Far Field → View Text Files to show the RCS data in text format.

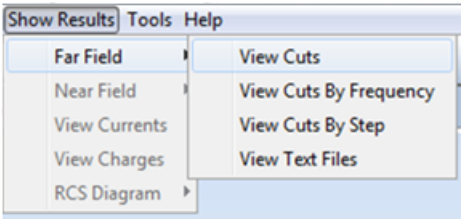


Figure 1035: Show Results → Far Field menu

Select the desired frequency and press OK.

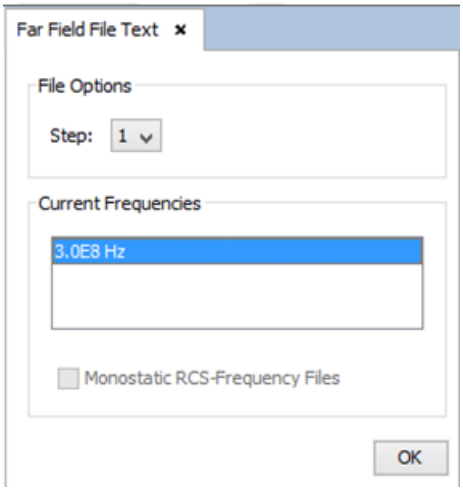


Figure 1036: Far Field View Text File options

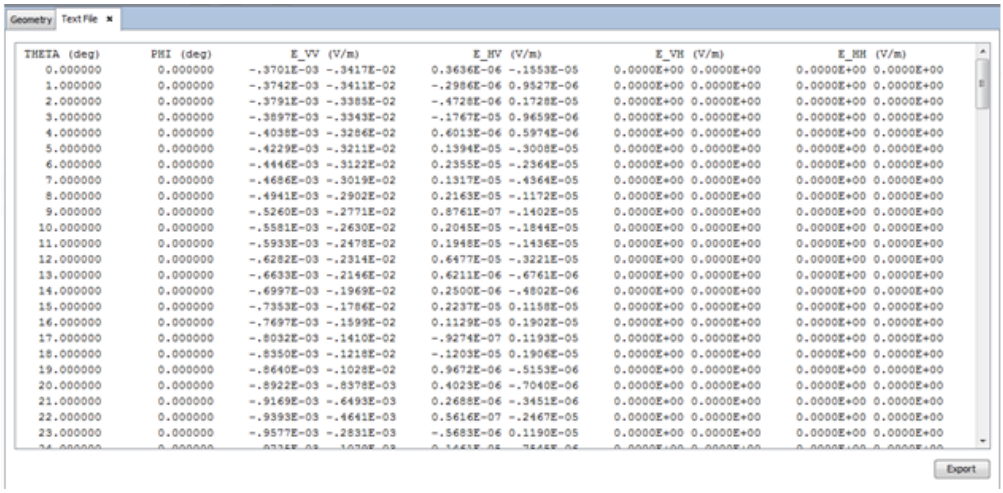


Figure 1037: Viewing results in text format

10.15.2 Example 2

In this example, we will calculate the bistatic RCS of an imported geometry using a frequency sweep from 0.4 GHz to 0.6 GHz.

Step 1

Create a new newFASANT project (using the MONCROS module) following Step 1 to Step 3 in Example 1.

Step 2

Select Geometry → Solid → Sphere.

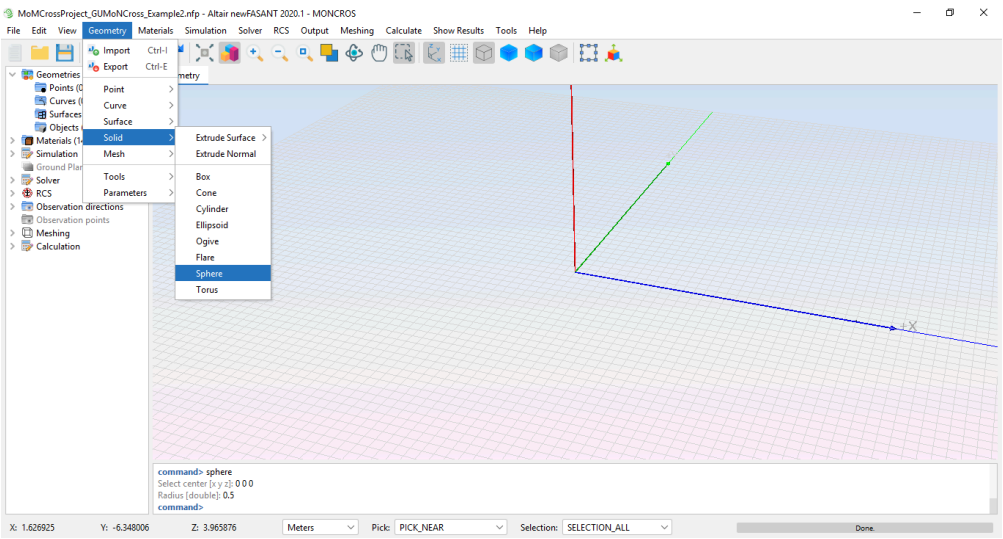


Figure 1038: Geometry menu

Step 3

Insert the parameters shown in the figure.

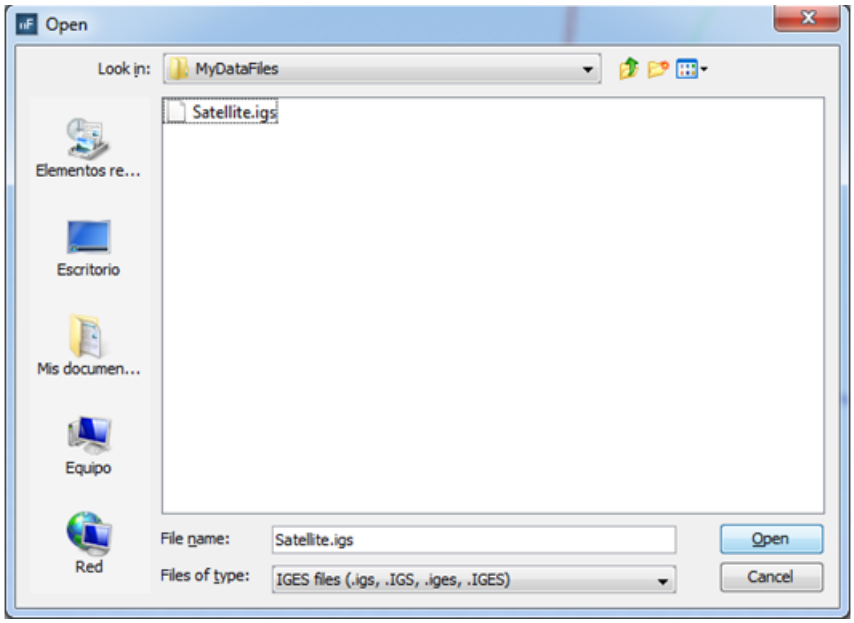


Figure 1039: Importing geometry

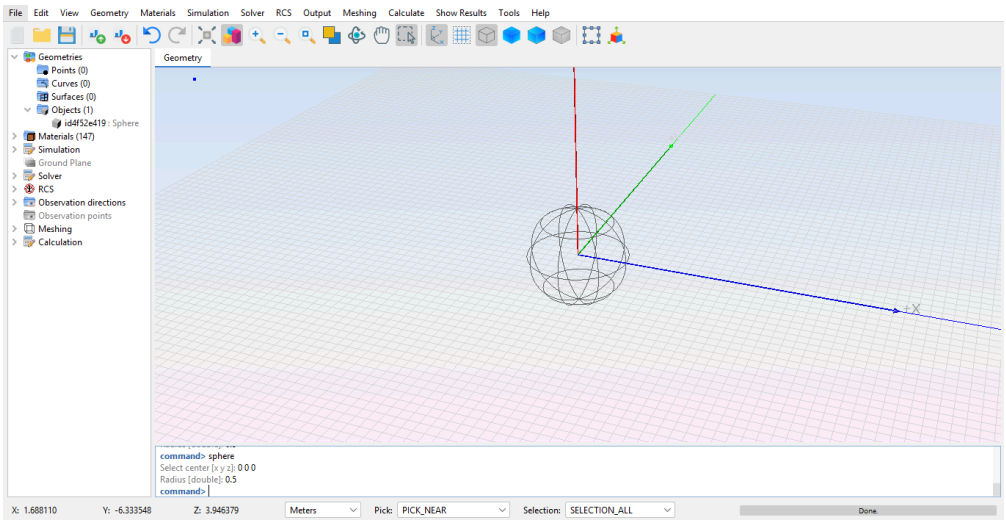


Figure 1040: Imported geometry

Step 4

Select the Simulation → Parameters option. Enable the Frequency Sweep option and set the initial frequency to 0.4 GHz and the final frequency to 0.6 GHz. Set the number of samples to 3 (the frequencies will be 0.4 GHz, 0.5 GHz and 0.6 GHz) and left-click the Save button.

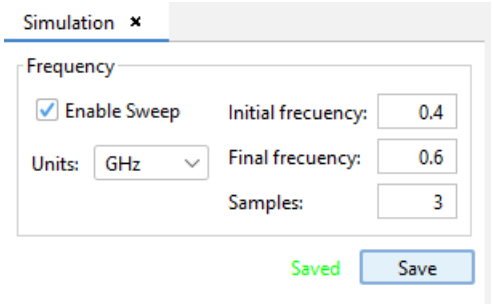


Figure 1041: Simulation parameters

Step 5

To set up the solver parameters, follow Step 8 of the Example 1 and follow the next step, making sure to comply with the final warning.

Step 6

Select the RCS → Parameters option. In the panel that appears, select the Bistatic RCS option and leave the default values for the remaining options. Press the Save button.

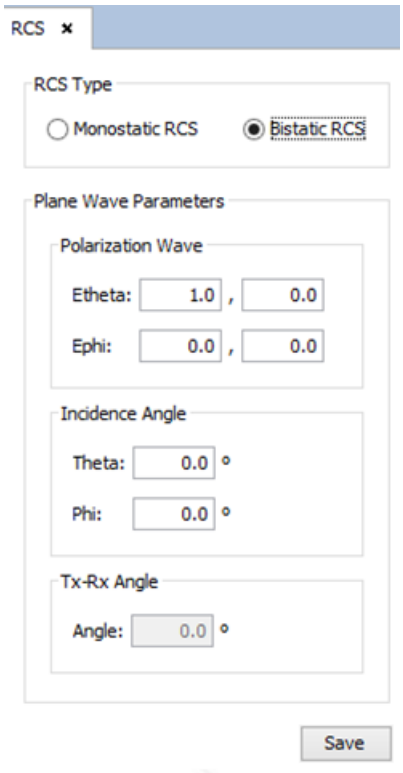


Figure 1042: RCS parameters

WARNING: return to 'Advanced Options' of the solver, disable 'Rigorous Calculate' option and enable 'Compute Pattern 3D' option with the angle value for 0.5 degrees as default.

Step 7

Select the Output → Observation Directions option. Modify the existing phi cut and set it to 45º. Leave the remaining fields intact. Press the OK button.

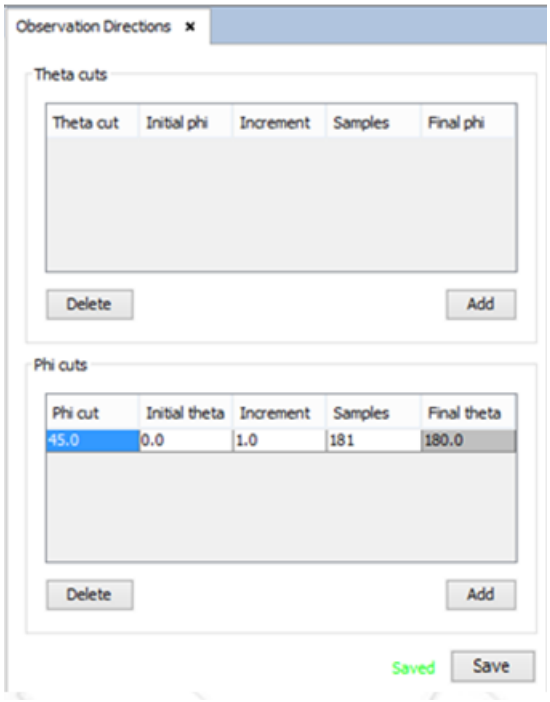


Figure 1043: Observation Directions panel

Step 8

Select the Output → Observation Points option. The following panel will be shown:

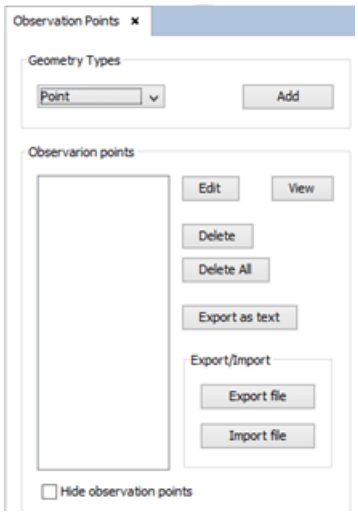


Figure 1044: Observation Points panel

Select the Line option from the Geometry Types combo-box and left-click the Add button. In the dialog that appears, configure the parameters as shown in the next figure and press the OK button:

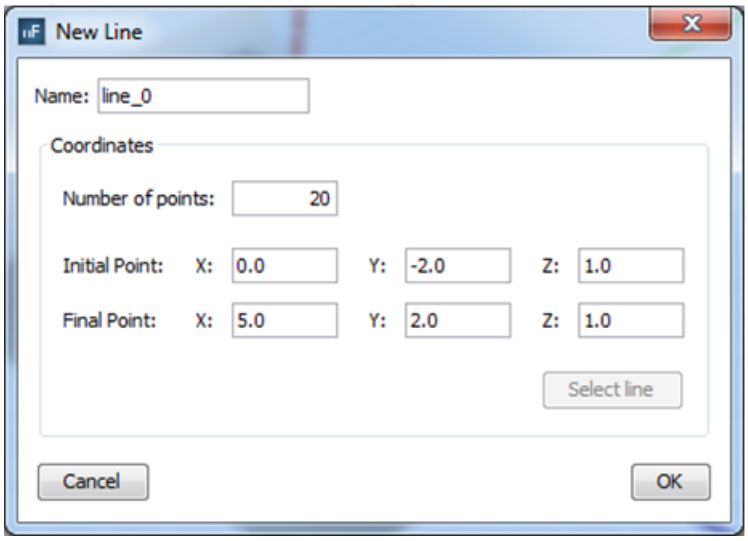


Figure 1045: Observation Line parameters

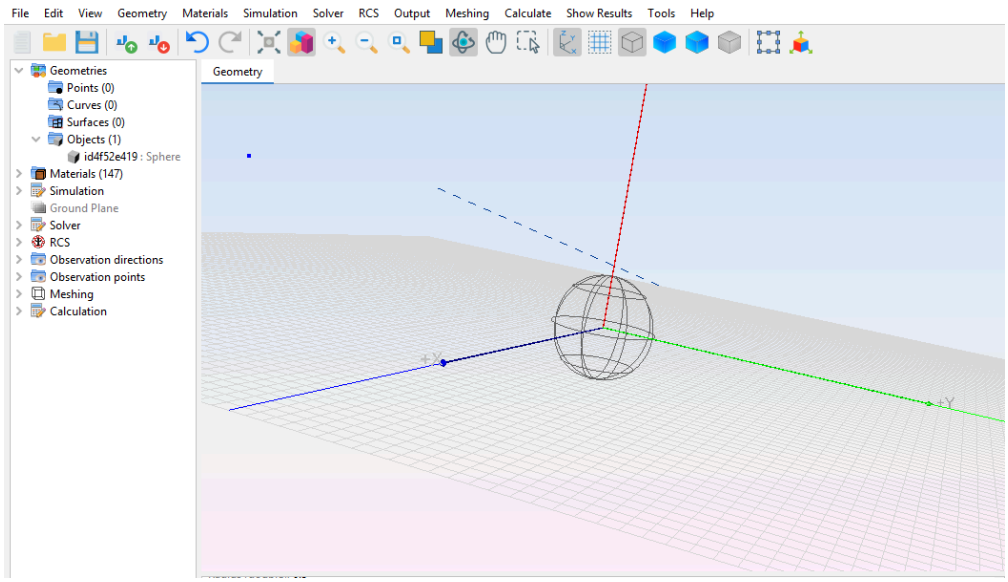


Figure 1046: Observation line visualization

Step 9

Select Meshing → Create Mesh. Set the options as shown in the following figure:

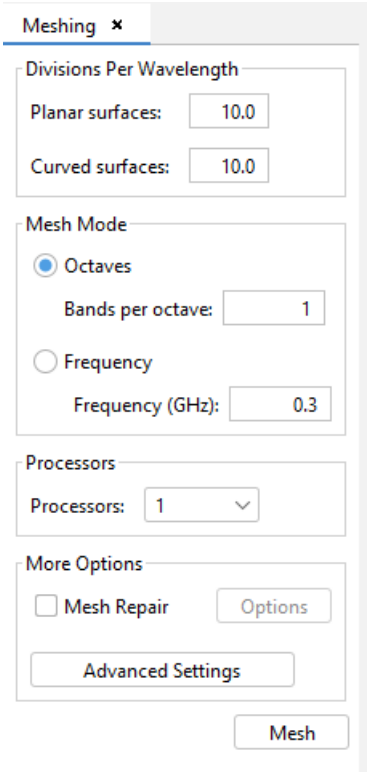


Figure 1047: Meshing options

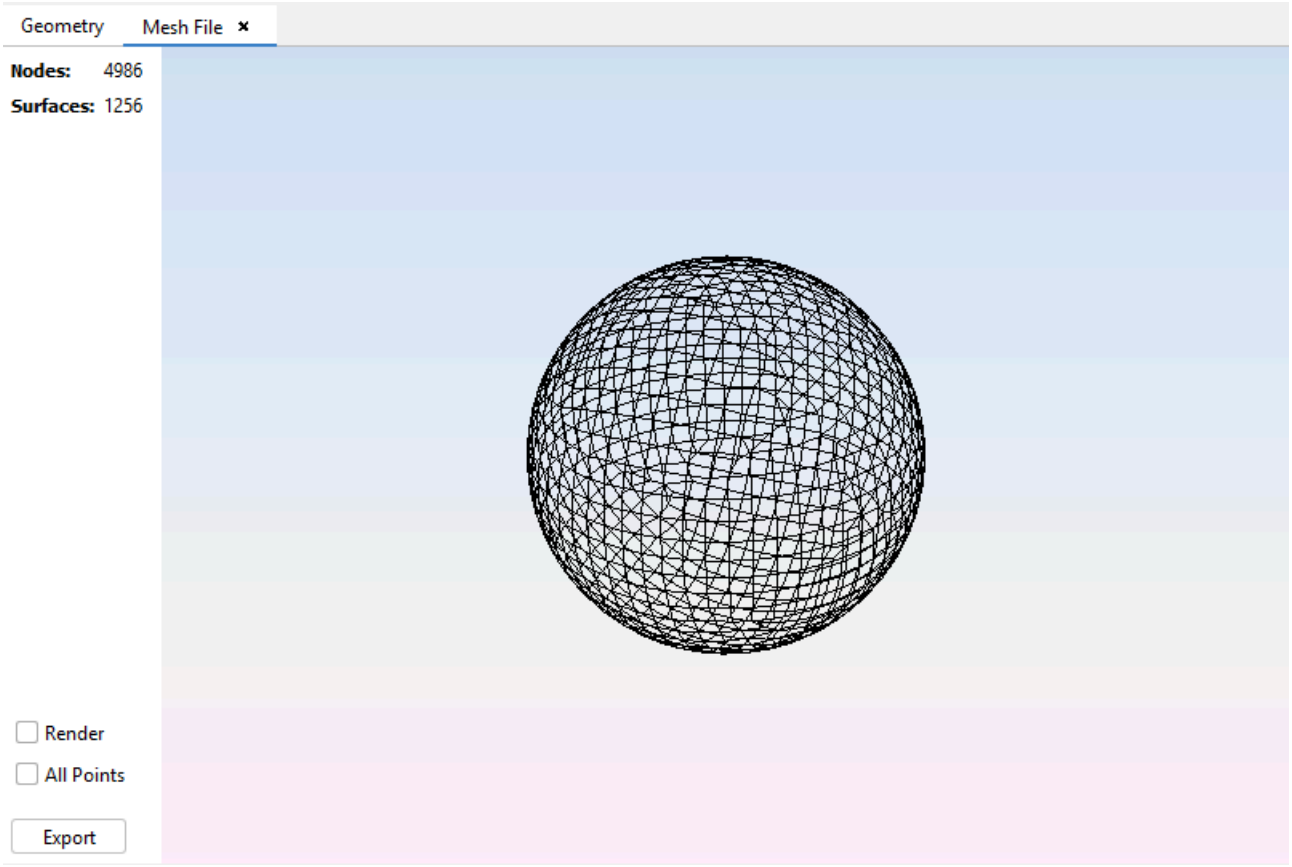


Figure 1048: Visualizing the created mesh

Step 10

Select Calculate → Execute and set the number of processors that will be used for the simulation. Press the Execute button to start the simulation.

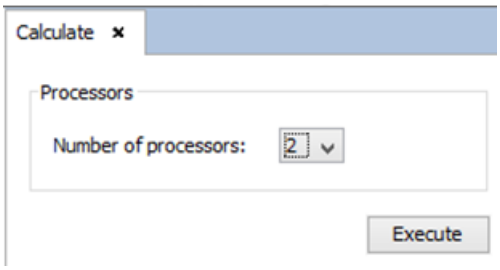


Figure 1049: Calculate options

Step 11

After the simulation finishes, select the option Show Results → Far Field → View Cuts.

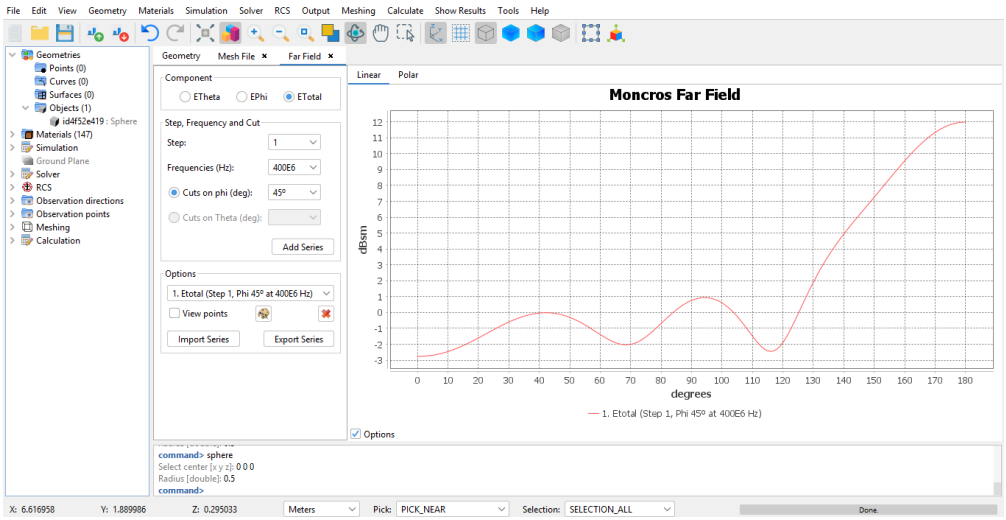


Figure 1050: RCS graphic

Initially, only the RCS for the first sampled frequency will be plotted. The user can add more series by selecting a different frequency in the Frequencies combo-box and left-clicking the Add Series button.

Step 12

Select Show Results → Far Field → View Text Files. Select the frequency to display the results for and left-click the OK button.

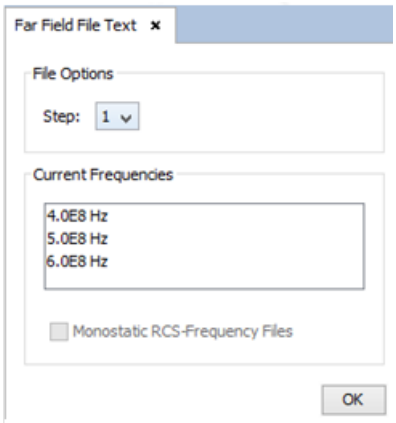


Figure 1051: Far Field Text File options

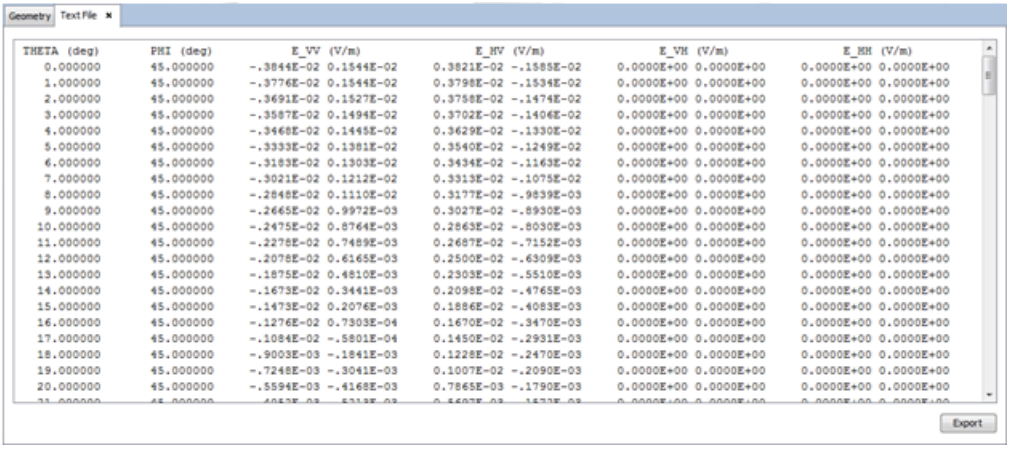


Figure 1052: Viewing RCS results in text format

Step 13

Select the Show Results → Near Field → View Near Field. Select the observation points to show (in our case, "line_0") and press the OK button.

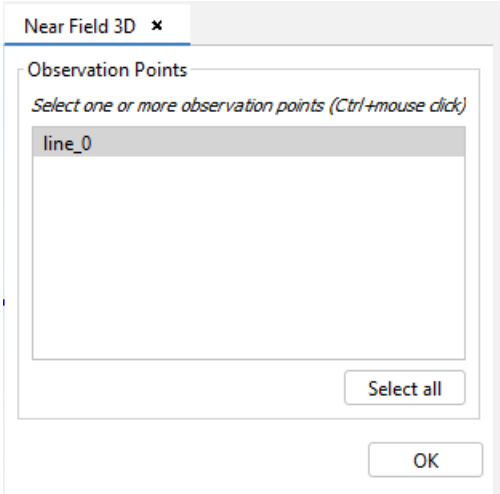


Figure 1053: View Near Field panel

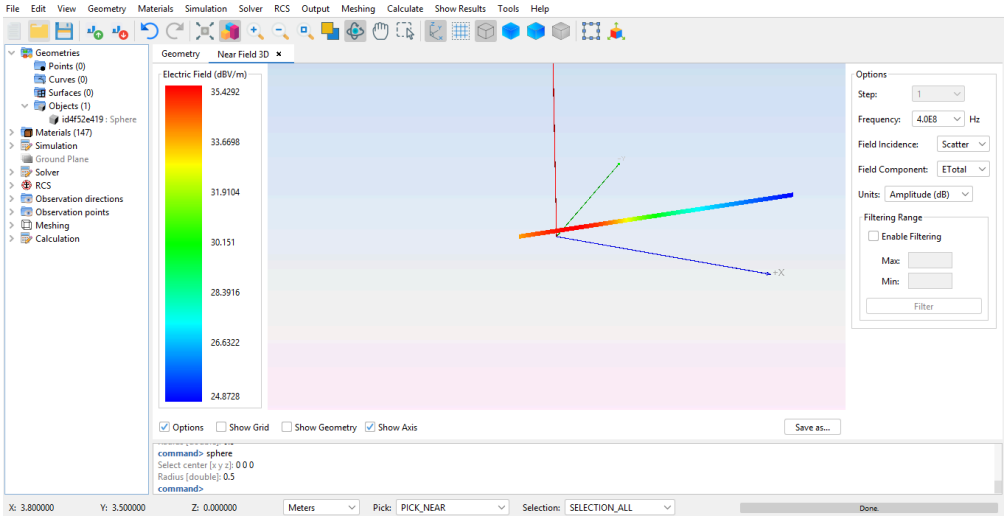


Figure 1054: Near Field representation

Step 14

Select the Show Results → View Currents option.

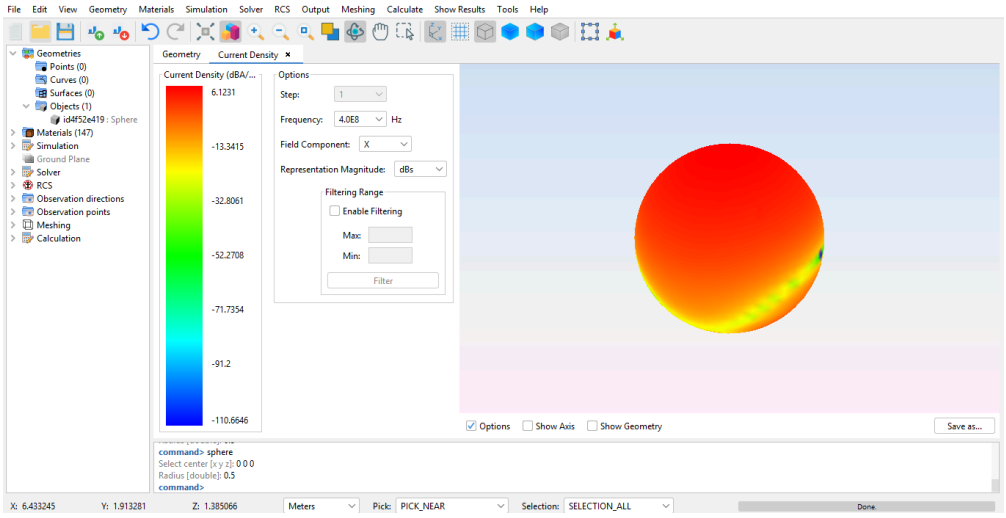


Figure 1055: Current Density representation

Step 15

Select the Show Results → View Charge option.

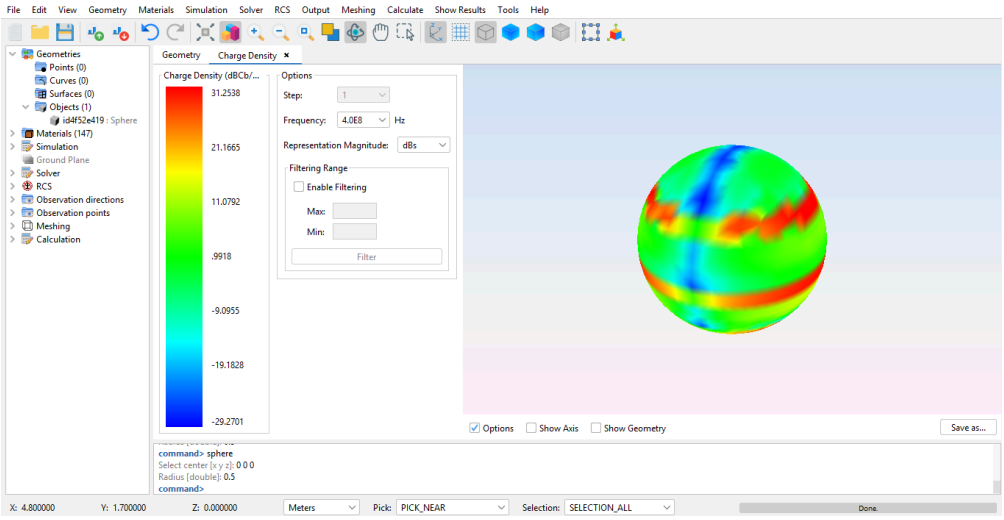


Figure 1056: Charge Density representation

Step 16

Select Materials → Add option.

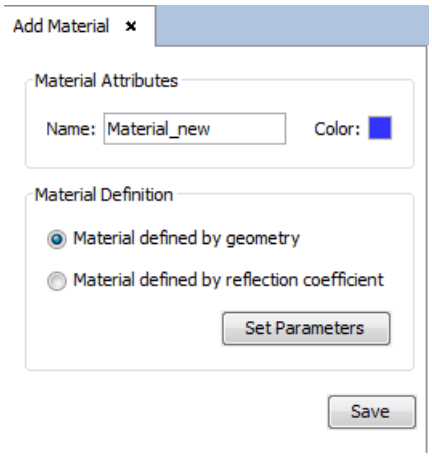


Figure 1057: Add material panel

Step 17

Select Material defined by Geometry option and press the Set Parameters button, to configure as shown the material properties.

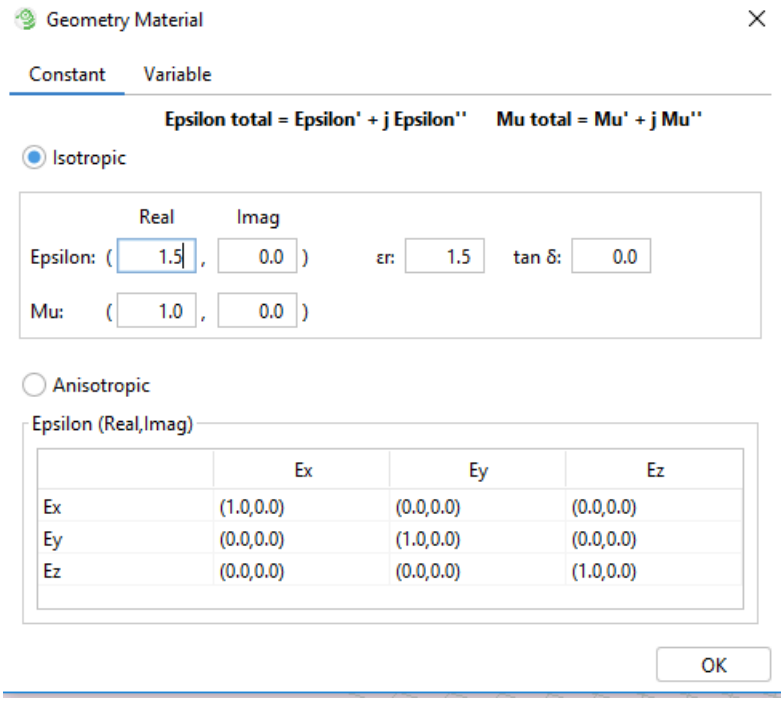


Figure 1058: Geometry material parameters

Step 18

Select the geometry on the screen and click on Material → Assign to assign the new material to those surfaces. Choose the material and specified the thickness.

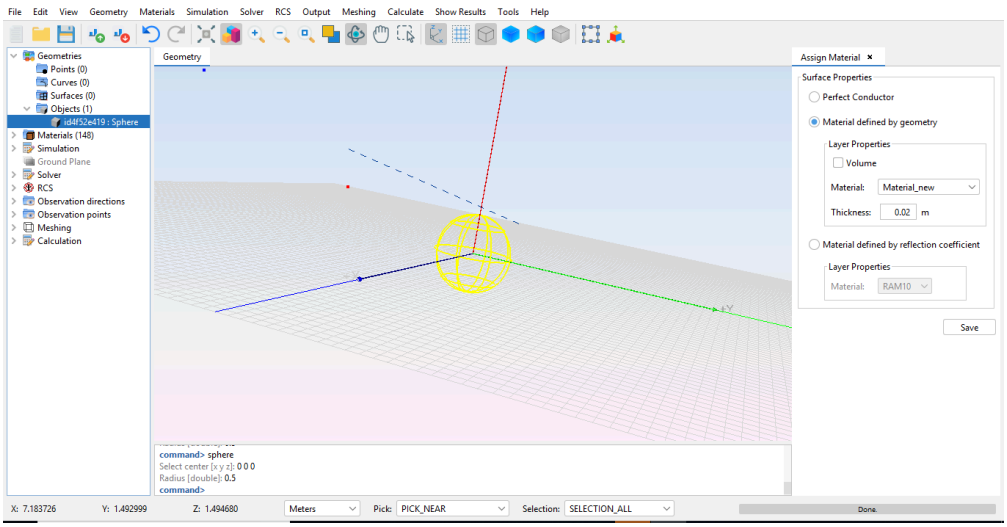


Figure 1059: Geometry selection

Step 19

Mesh the geometry and run the simulation again.

Step 20

Select the option Show Results → Far Field → View Cuts.

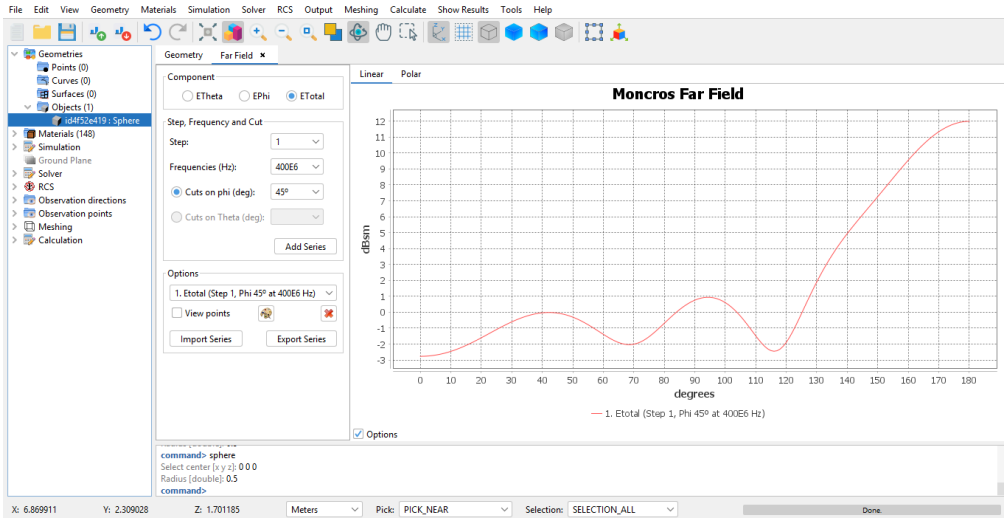


Figure 1060: RCS graphic

Step 21

Select the Show Results → View Currents option.

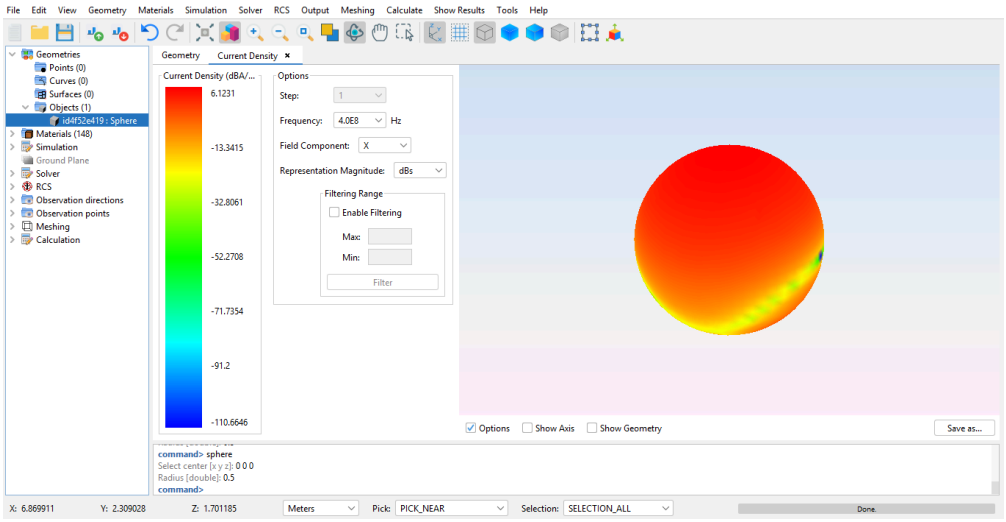


Figure 1061: Current Density representation

Step 22

Select the Show Results → View Charge option.

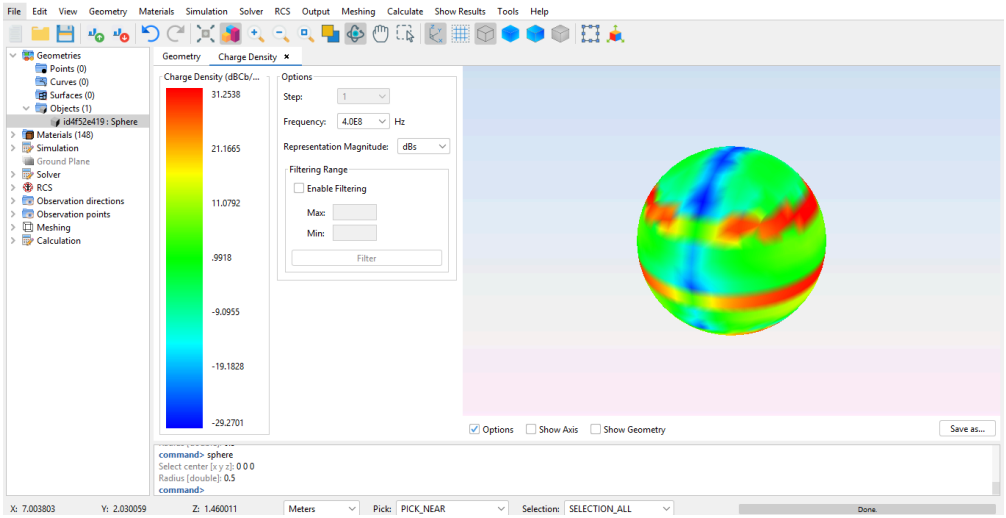


Figure 1062: Charge Density representation

10.15.3 Example 3

This example shows the dynamic monostatic RCS of two cubes.

Step 1

Create a new newFASANT project (using the MONCROS module) following the Step 1 to Step 3 in Example 1.

Step 2

Select Geometry → Parameters→ Define parameters and introduce the following parameters.

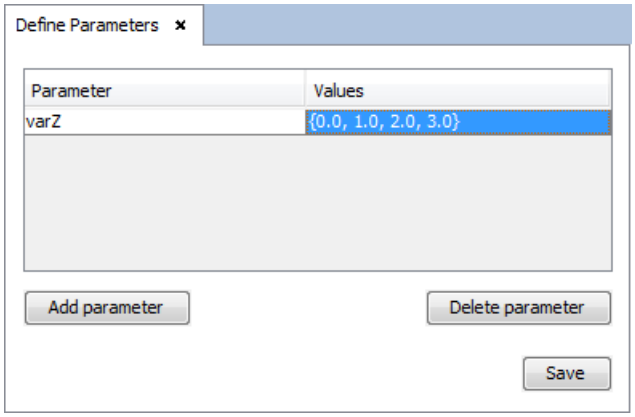


Figure 1063: Define parameters panel

Step 3

Select Geometry → Solid→ Box to define the box parameters as follows:

```
command> box
First corner of base [x y z]: -0.5 -0.5 varZ
Base size [width depth]: 1 1
Height [double]: 1
command>
```

Figure 1064: Box parameters

Step 4

Repeat Step 3 to define another box.

```
command> box
First corner of base [x y z]: 0.6 -0.5 0
Base size [width depth]: 1 1
Height [double]: 1
```

Figure 1065: Box parameters

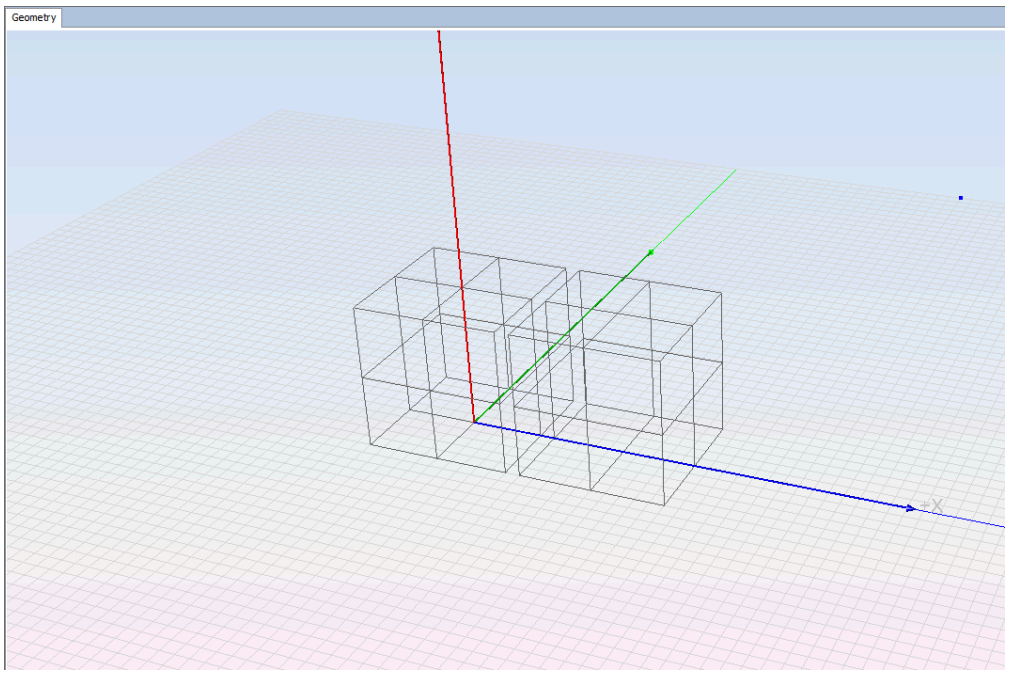


Figure 1066: Geometry composed of two boxes

Step 5

In the Simulation panel select a frequency of 0.3 GHz (default options) and press Save.

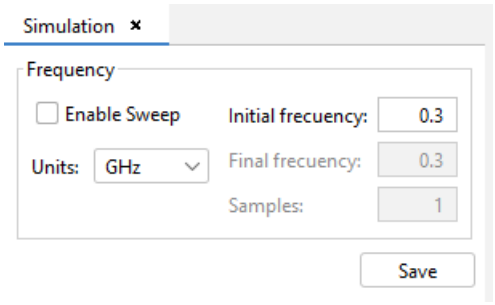


Figure 1067: Simulation panel

Step 6

Select the Solver → Parameters option and choose CFIE solution

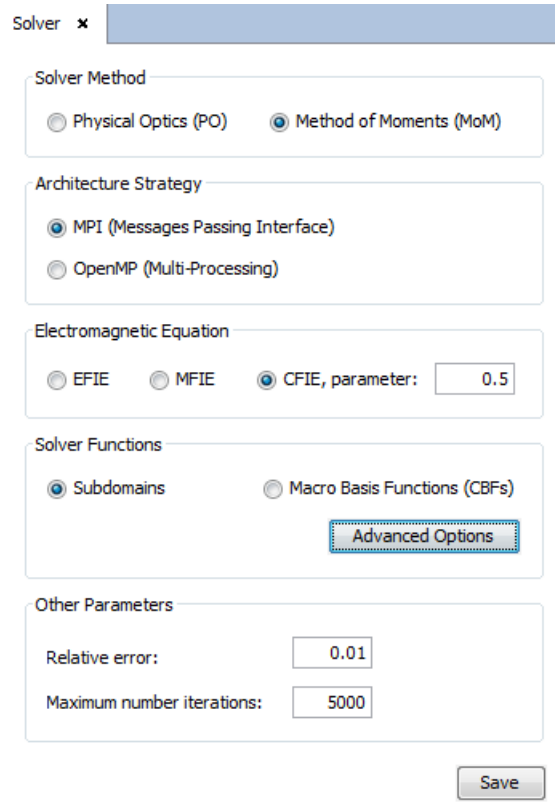
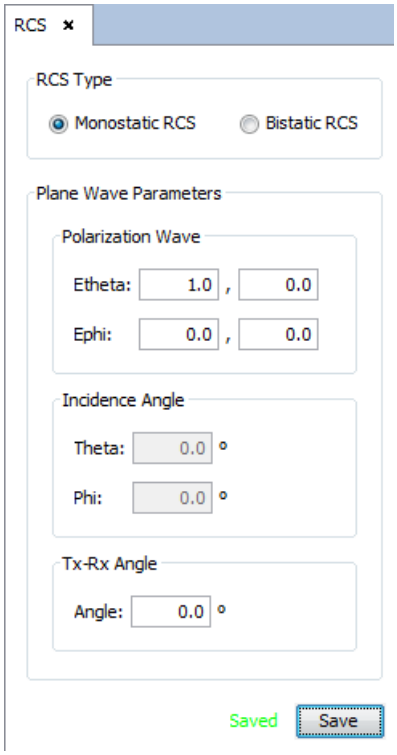


Figure 1068: Solver panel

Step 7

Select the RCS → Parameters option. Select Monostatic RCS and leave the default parameters. Press the Save button.



Step 8

Select Output → Observation directions. Leave the default observation directions (0.0 phi cut with theta angle varying between 0.0 and 180.0 using 181 samples).

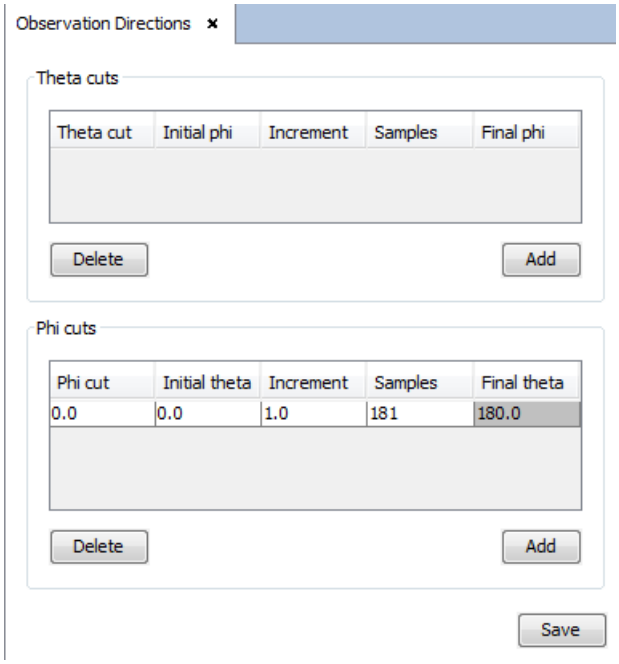


Figure 1069: Observation directions panel

Step 9

Before running the simulation, select the Meshing → Create Mesh option. Set 10 divisions on planar and curved surfaces, set Processors to 2 and the default number of bands per octave. Press the Mesh button.

Step 10

Select Calculate → Execute and set the number of desired processors to run the simulation. Press the Execute button to start the calculation.

Step 11

Select Show Results → Far Field→ View Cuts to visualize the far field values for the first step.

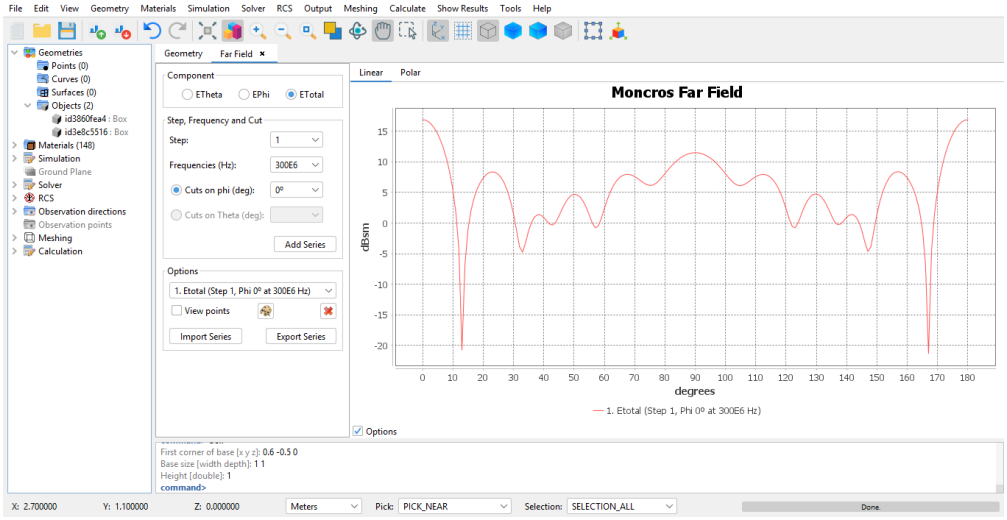


Figure 1070: Far Field results

Step 12

Add a new series selecting the step on the combo box and click on **Add Series** button.

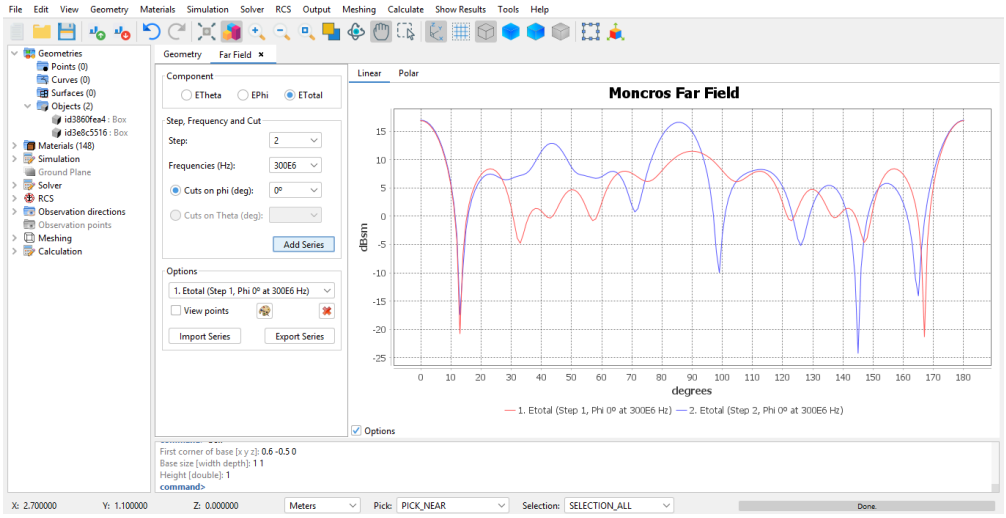
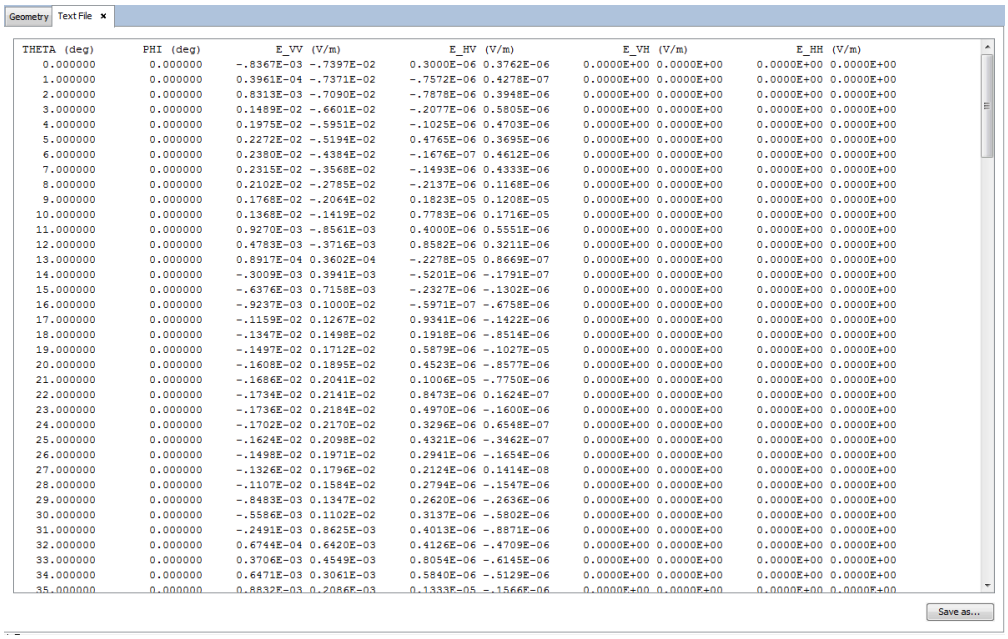


Figure 1071: Far Field results

Step 13

Select Show Results → Far Field→ View Text Files to visualize the text file with the results of the simulation.


Geometry | Text File ×

THETA (deg)	PHI (deg)	E_VV (V/m)	E_HV (V/m)	E_VH (V/m)	E_HH (V/m)
0.000000	0.000000	-.8367E-03 -.7397E-02	0.3000E-06 0.3762E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
1.000000	0.000000	0.3961E-04 -.7371E-02	-.7572E-06 0.4278E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
2.000000	0.000000	0.8313E-03 -.7090E-02	-.7878E-06 0.3948E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
3.000000	0.000000	0.1489E-02 -.6601E-02	-.2077E-06 0.5805E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
4.000000	0.000000	0.1975E-02 -.5951E-02	-.1025E-06 0.4703E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
5.000000	0.000000	0.2272E-02 -.5194E-02	0.4765E-06 0.3695E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
6.000000	0.000000	0.2380E-02 -.4384E-02	-.1676E-07 0.4612E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
7.000000	0.000000	0.2315E-02 -.3568E-02	-.1493E-06 0.4333E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
8.000000	0.000000	0.2102E-02 -.2785E-02	-.2137E-06 0.1168E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
9.000000	0.000000	0.1768E-02 -.2064E-02	0.1823E-05 0.1208E-05	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
10.000000	0.000000	0.1368E-02 -.1419E-02	0.7783E-06 0.1716E-05	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
11.000000	0.000000	0.9270E-03 -.8561E-03	0.4000E-06 0.5551E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
12.000000	0.000000	0.4783E-03 -.3716E-03	0.8582E-06 0.3211E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
13.000000	0.000000	0.8917E-04 0.3602E-04	-.2278E-05 0.8669E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
14.000000	0.000000	-.3009E-03 0.3941E-03	-.5201E-06 -.1791E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
15.000000	0.000000	-.6376E-03 0.7158E-03	-.2327E-06 -.1302E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
16.000000	0.000000	-.9237E-03 0.1000E-02	-.5971E-07 -.6758E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
17.000000	0.000000	-.1159E-02 0.1267E-02	0.9341E-06 -.1422E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
18.000000	0.000000	-.1347E-02 0.1498E-02	0.1918E-06 -.8514E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
19.000000	0.000000	-.1497E-02 0.1712E-02	0.5879E-06 -.1027E-05	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
20.000000	0.000000	-.1608E-02 0.1895E-02	0.4523E-06 -.8577E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
21.000000	0.000000	-.1686E-02 0.2041E-02	0.1006E-05 -.7750E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
22.000000	0.000000	-.1734E-02 0.2141E-02	0.8473E-06 0.1624E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
23.000000	0.000000	-.1736E-02 0.2184E-02	0.4970E-06 -.1600E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
24.000000	0.000000	-.1702E-02 0.2170E-02	0.3296E-06 0.6548E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
25.000000	0.000000	-.1624E-02 0.2098E-02	0.4321E-06 -.3462E-07	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
26.000000	0.000000	-.1498E-02 0.1971E-02	0.2941E-06 -.1654E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
27.000000	0.000000	-.1326E-02 0.1796E-02	0.2124E-06 0.1414E-08	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
28.000000	0.000000	-.1107E-02 0.1584E-02	0.2794E-06 -.1547E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
29.000000	0.000000	-.8483E-03 0.1347E-02	0.2620E-06 -.2636E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
30.000000	0.000000	-.5586E-03 0.1102E-02	0.3137E-06 -.5802E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
31.000000	0.000000	-.2491E-03 0.8625E-03	0.4013E-06 -.8871E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
32.000000	0.000000	0.6744E-04 0.6420E-03	0.4126E-06 -.4709E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
33.000000	0.000000	0.3706E-03 0.4549E-03	0.8054E-06 -.6145E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
34.000000	0.000000	0.6471E-03 0.3061E-03	0.5840E-06 -.5129E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00
35.000000	0.000000	0.8832E-03 0.2086E-03	0.1333E-05 -.1566E-06	0.0000E+00 0.0000E+00	0.0000E+00 0.0000E+00

Save as...

Figure 1072: Text file result

10.16 Annex 1: Post-Processing Currents

A new option has been added to the MONCROS module that allows to generate a set of currents file in binary files to be preprocessed; and with the new currents in binary format could be used to obtain the scattering field. To use this new option, the user needs to do the following steps:

Open newFASANT code and open MONCROS module.

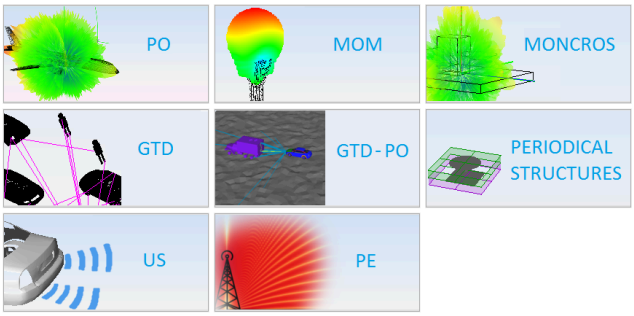


Figure 1073: Module selection screen

Create for example a flat face with 1 m x 1 m. Use the command plane from Command Window.

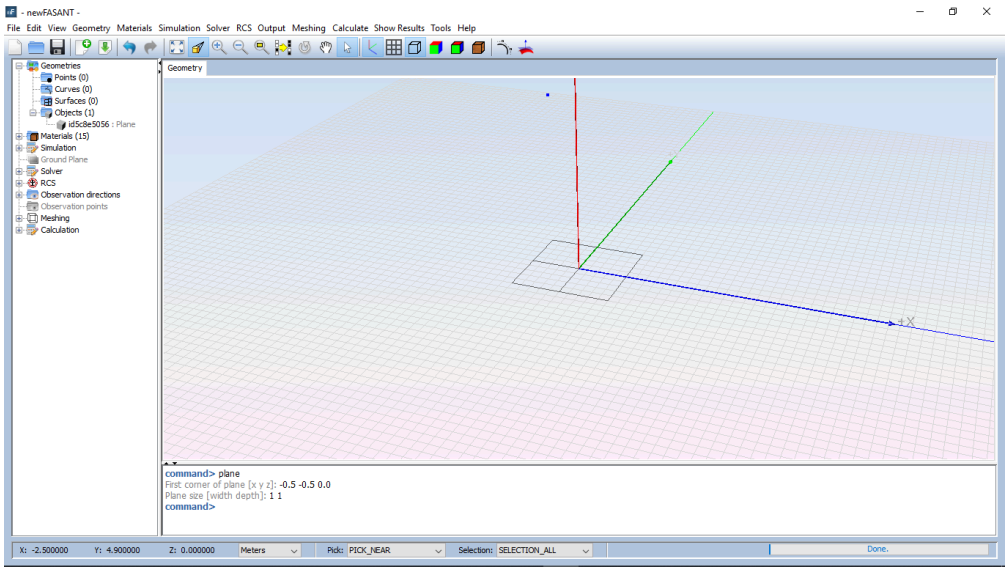


Figure 1074: Creating a 1x1 plane

Set the Simulation Parameters using Simulation > Parameters menu. In this case, use the default values.

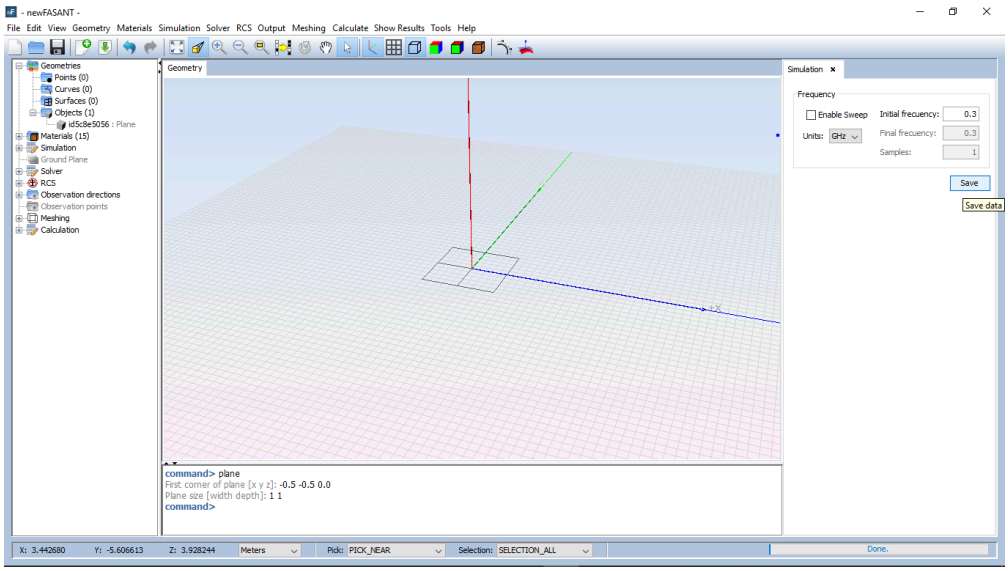


Figure 1075: Simulation options

From RCS menu, use Monostatic Option and select Output Currents option. This must be selected to activate the postprocess currents menu in the Output menu. Press Save button.

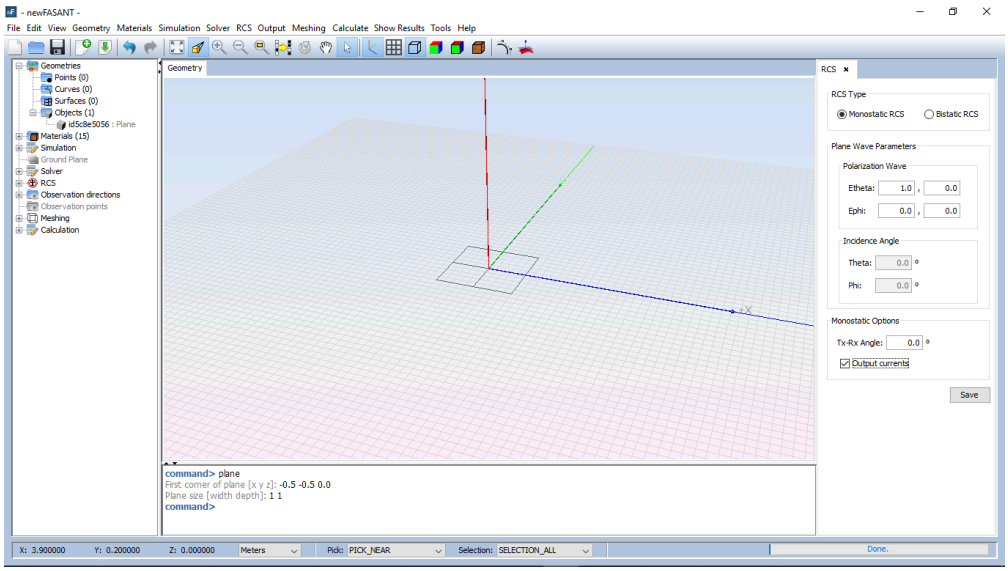


Figure 1076: RCS parameters

From Output > Observations Directions, the user has to define the observation directions that can be set using the cuts in theta or phi from the dialog, or use an external file to Import the samples of the observation directions where the currents will be calculated. If a file is used, the format has to be in ASCII file with this structure:

Theta_sample1 Phi_sample1

Theta_sample2 Phi_sample2

……..

Theta_smapleN Phi_sampleN

Example:

0.0 0.0

45.0 0.0

90.0 0.0

If this file is loaded, then the samples will be loaded in the theta cuts table. Use the Import File Button to select a .TXT file with the sample. Press Save to set.

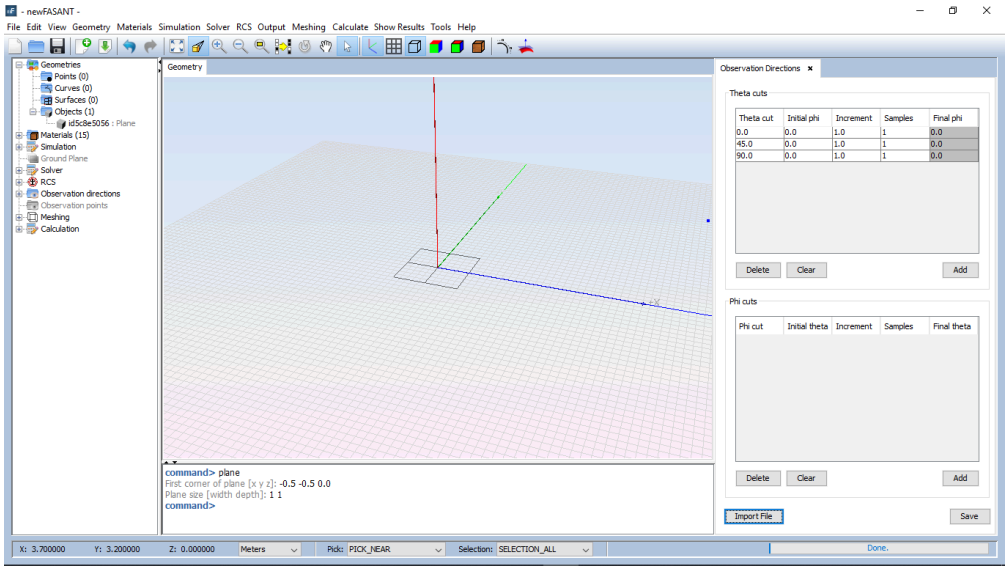


Figure 1077: Observation Directions options

The next step is to mesh the geometry. Please use Meshing > Create Mesh command menu. Use the default values.

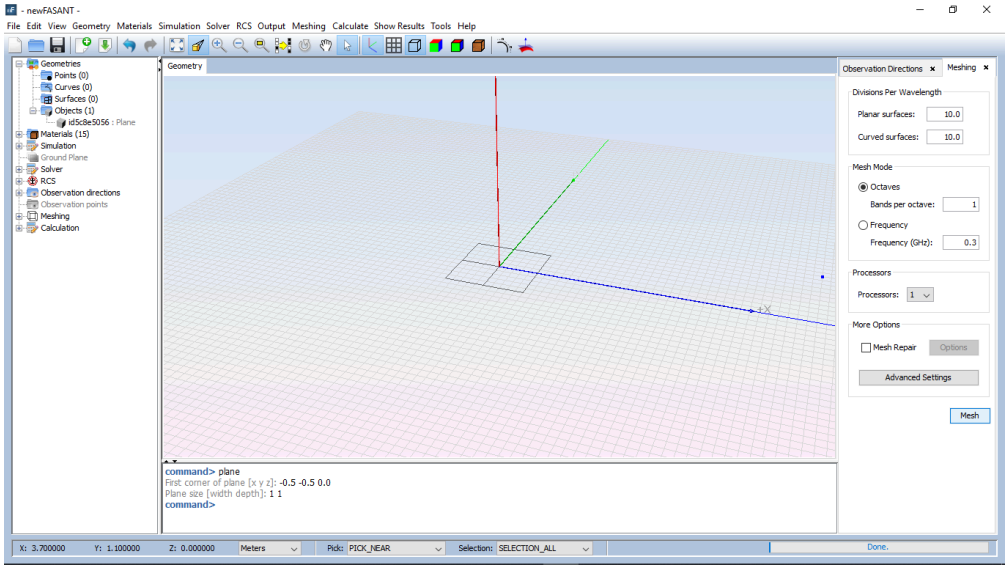


Figure 1078: Meshing options

To generate the currents of every observation direction, please use Calculate command menu.

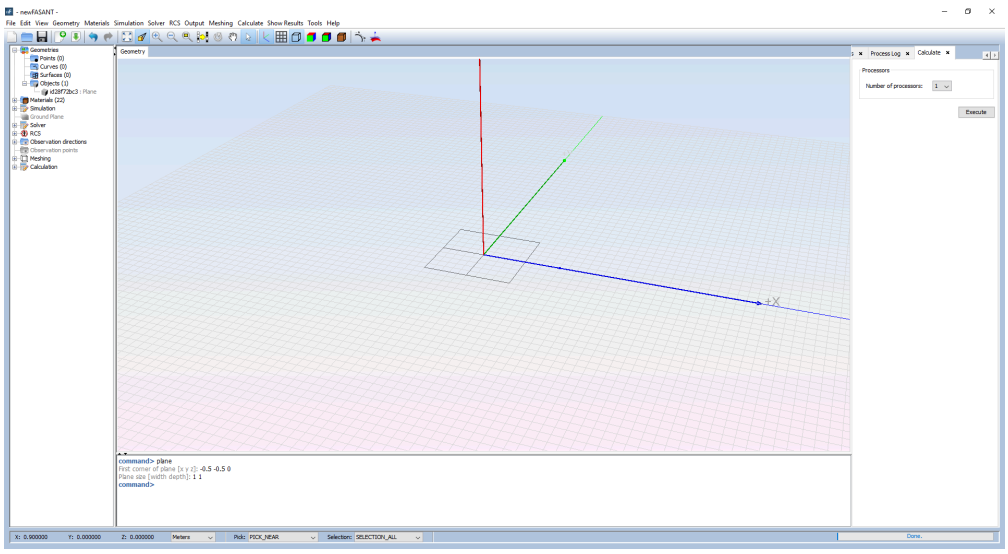


Figure 1079: Calculate options

When the simulation has finished, the binary currents file will be in the folder results from the project folder. So, from File > Project Folder command menu, a windows explorer from the project folder will be opened. Please log into results > step0 > f0 (first frequency) al the results will be there.

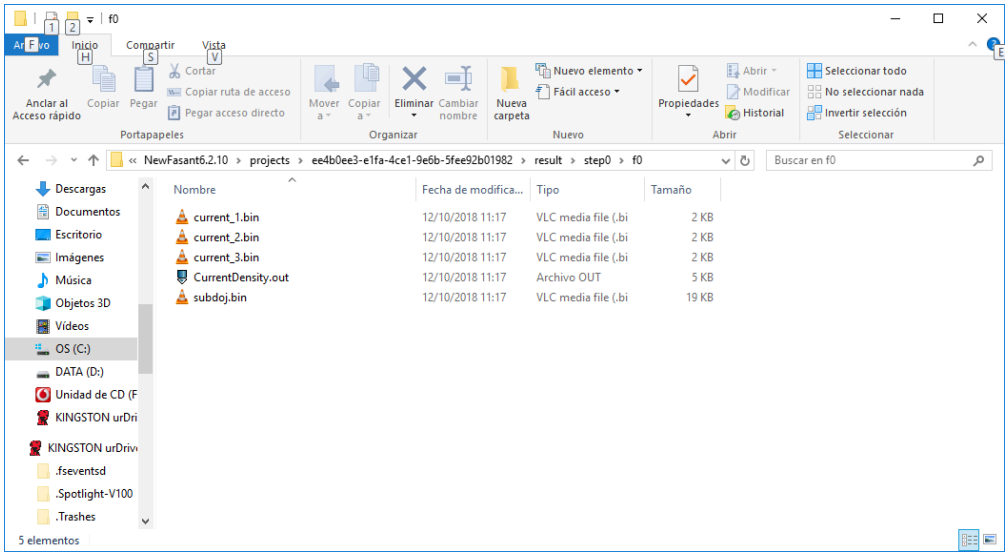


Figure 1080: Binary current files

There are three binary file because there are three observations directions samples, and also an important file called subdoj.bin file that it is necessary to obtain the scattering fields.

When the user has preprocessed the input currents binary files, then the user can put all the new currents in any folder, but the subdoj.bin file must be also included. For example, a new folder into mydatfiles has been created with the name currents. The new binary currents files will be placed in it and also the subdoj.bin file. If the new files are located into the folder project, the new files will be saved with the project.

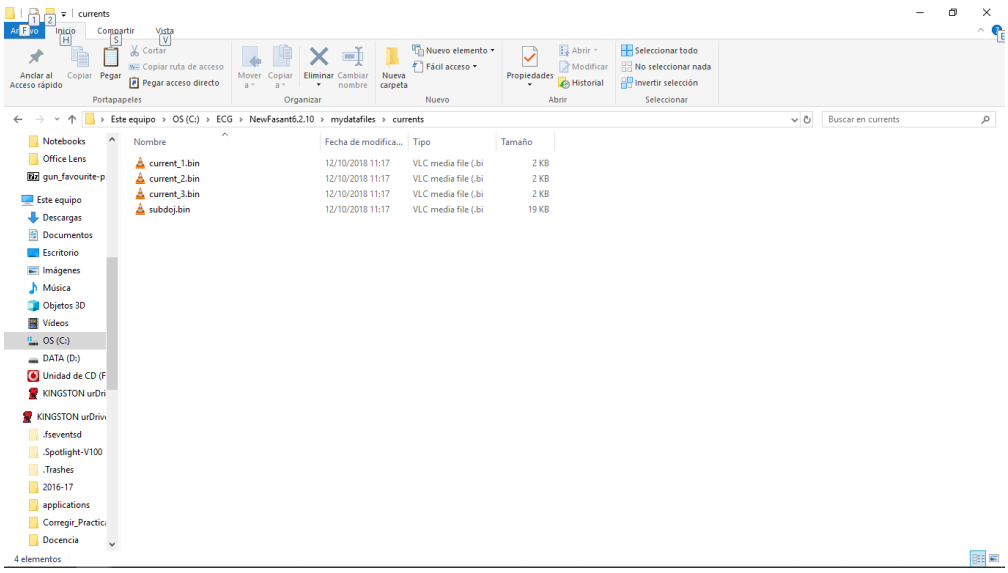


Figure 1081: Binary current files copied to mydatfiles

To postprocess the new currents and obtain the scattering field, the Show Results > Post-Process Currents command menu must be used.

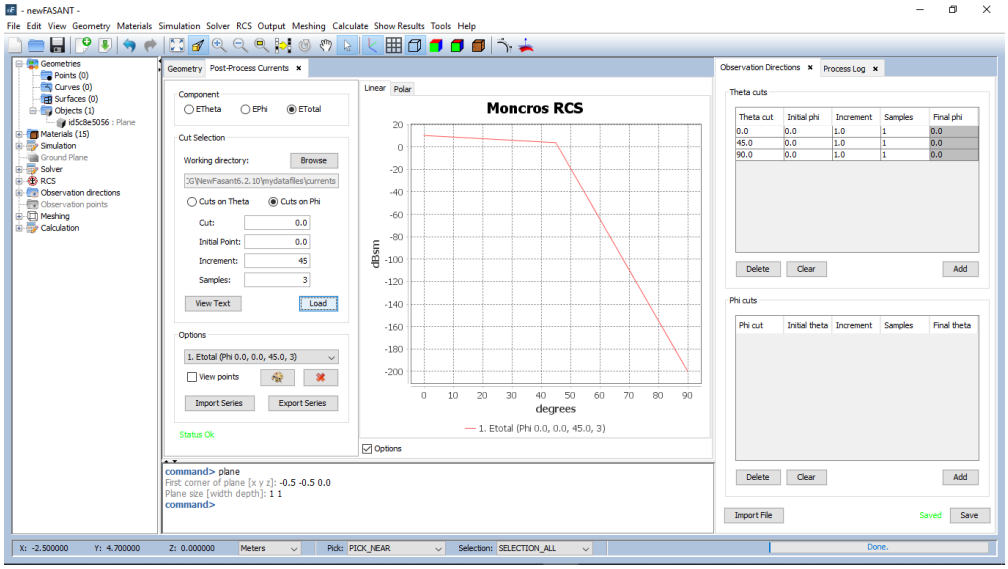


Figure 1082: RCS graphic

To obtain the results, please follow these steps:

- With the Browse button, select the folder where the new currents and subdoj.bin file is located.
- Select if the results that has to be displayed on the graph will be a cut on theta or cut on phi, and set the value of the cut (degree), the initial value (degree), increment (degree) and number of samples. In this case there is only three files with one cut in phi of 0º and a sweeping point in theta from 0º to 90º with a step of 45º and 3 samples.
- Press load to obtain the values. If everything is right, the Status will be OK, if there is anything wrong, the status will be Error.
- Also the results of the scattering field in text mode can be obtained if the View Text Button is pressed.

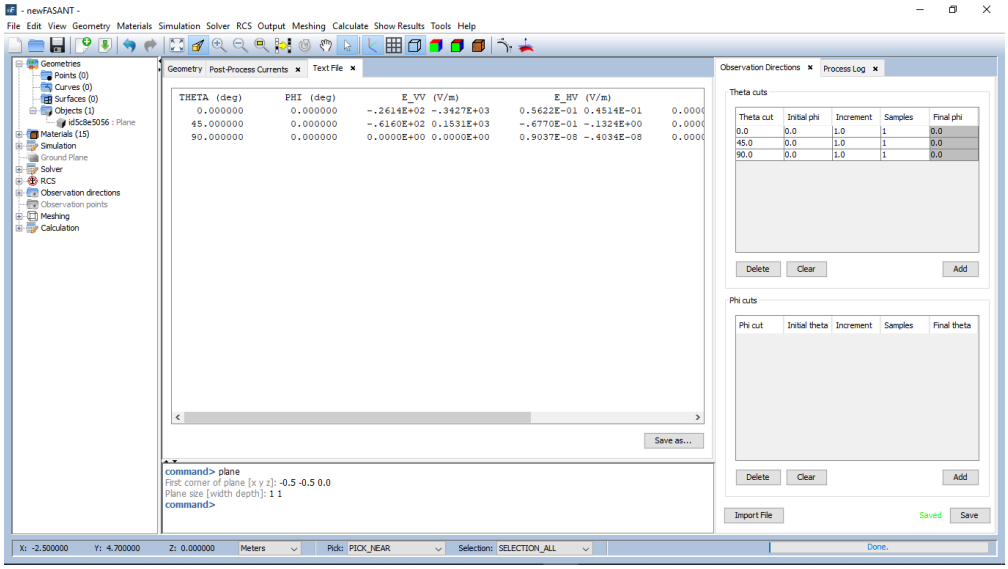


Figure 1083: Viewing results in text format

It is very important that the value of the theta,phi samples in the currents binary file must be exactly the value the user is setting in the Post-Process command, if not, there will be an error. Also, a binary file for every observation point is being processed must be present, if not, an error will be obtained in the post-processing stage.

MOM

11

MOM is used for the 3D analysis of complex antennas, on board antennas, electromagnetic compatibility and radar cross section (RCS).

This chapter covers the following:

- 11.1 File Menu (p. 1037)
- 11.2 Edit Menu (p. 1038)
- 11.3 View Menu (p. 1039)
- 11.4 Geometry Menu (p. 1040)
- 11.5 Materials Menu (p. 1041)
- 11.6 Simulation Menu (p. 1042)
- 11.7 Solver Parameters Menu (p. 1045)
- 11.8 Source (p. 1055)
- 11.9 Output Menu (p. 1056)
- 11.10 Meshing Menu (p. 1068)
- 11.11 Calculate Menu (p. 1076)
- 11.12 Show Results (p. 1078)
- 11.13 Tools Menu (p. 1132)
- 11.14 Help Menu (p. 1133)
- 11.15 Training Examples (p. 1134)
- 11.16 Test Cases (p. 1439)

11.1 File Menu

For information about the File menu, refer to File Menu.

11.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

11.3 View Menu

For information about the View menu, refer to View Menu.

11.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

11.5 Materials Menu

For information about the Materials menu, refer to Materials.

11.6 Simulation Menu

This menu contains the following options:

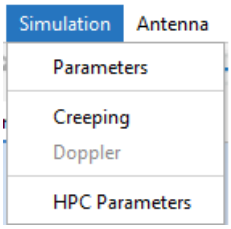


Figure 1084: Simulation Parameters menu

11.6.1 Parameters

Selecting this option opens the **Simulation** panel on the right of the window:

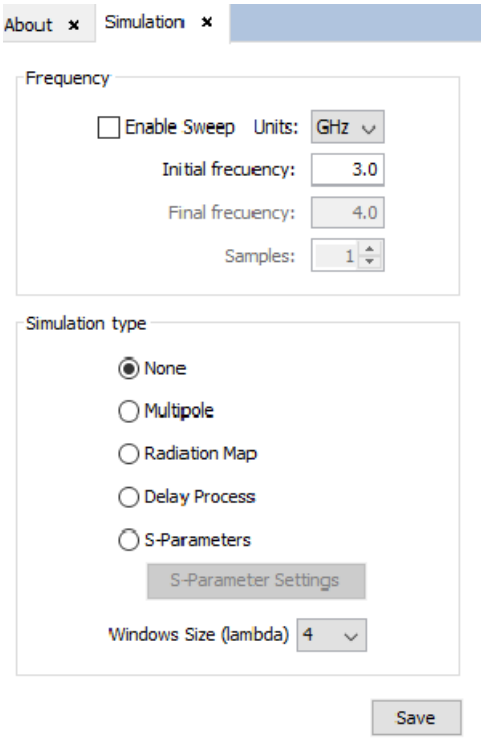


Figure 1085: Simulation panel

In this panel, the user can set the main parameters of the simulation:

Frequency

To set the simulation frequency. The following parameters may be modified:

- ***Enable Sweep***: select this option to simulate a range of frequencies.
- ***Units***: to set the magnitude of the frequency. Hz, kHz, MHz, and GHz are available.

- ***Initial frequency***: the first step in the frequency range, in the selected *Units*. If the frequency *Sweep* is disabled, this value coincides with the simulation frequency.
- ***Final frequency***: the last step in the frequency range, in the selected *Units*. This parameter may be only modified when the frequency *Sweep* is enabled.
- ***Samples***: number of frequency samples to be analyzed on the frequency *Sweep*. This parameter may be only modified when the frequency *Sweep* is enabled.

Simulation Type

- Selecting ***None*** type, the user will obtain default results as radiation pattern, current density, charge density, near-field observation, etc.
- Selecting ***Multipole*** type, the user will obtain default result and multipole files that can be used in other simulation as antenna input. A *multipole* consists in a division on parallelepipedal regions of the simulated case. Each region contains the whole information such as its geometry and its currents, so this representation is more accurate than the radiation pattern files. The multipole files only will provide accurate results in further Far Field simulations.
- Selecting ***Radiation Map*** type, the user will obtain default results and the radiation map results. This simulation enables the *Radiation Map* results, which show the scattered field by each region in the observation direction.
- Selecting ***Delay Process*** type, the user will obtain default results and the delay process results. This simulation enables the *Delay Process* results, which show the interferences introduced in a given input *Signal* in the time domain by each region in the observation direction.
- Selecting ***S-Parameters***, the user will obtain S-Parameters results of coaxial feed antennas and waveguides. Note that with simulation type, the only provided results after the simulation process are the S-Parameters. When this simulation type is selected, the *S-Parameters Settings* button is enabled to modify advanced parameters that affect to the waveguides analysis. The following parameters may be configured:
  - ***Lambda guide factor***: it is the value for dividing the wavelength and then obtaining the distance to extend the main port in the analysis process. The higher parameters result in shorter extensions.
  - ***Lambda guide factor for short-circuiting***: it is the value for dividing the wavelength and then obtaining the distance to extend the ports that are not fed in the analysis process. The higher parameters result in lower extensions.

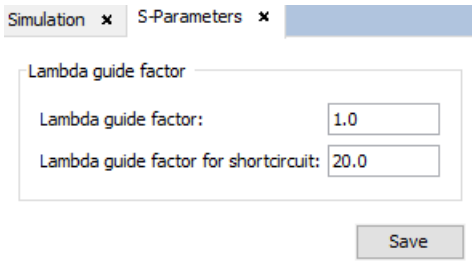


Figure 1086: Advanced S-Parameters setting

The **Window Size (lambda)** parameter specifies the number of lambdas which defines the size edge of the generated regions in multipoles output files. This option only may be modified when a **Multipole** or **Radiation Map** simulation types are selected.

11.6.2 HPC Parameters

For more information on how to set up a queueing system, see High Performance Computing (HPC)/ Queueing.

11.7 Solver Parameters Menu

This section contains the options available to set the set-up the solver parameters.

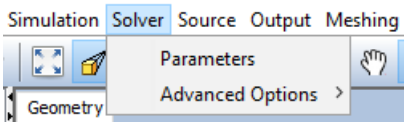


Figure 1087: Solver menu

11.7.1 Parameters

Clicking on the **Solver** → **Parameters** option, the next panel is opened.

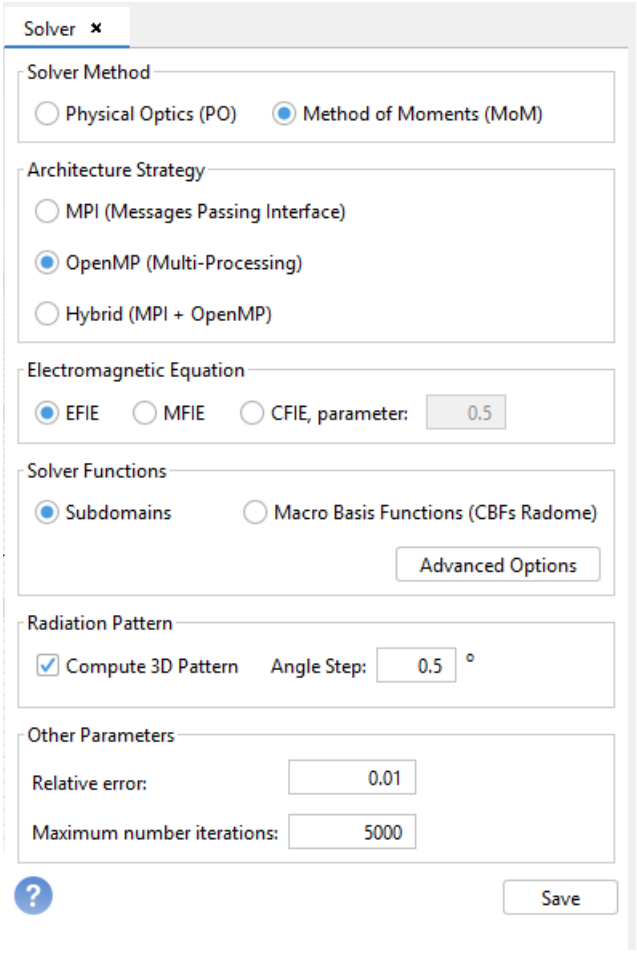


Figure 1088: Solver Panel

- **Solver Method** the method used to solve the solution. Two options are available:
  - **PO (Physical Optics)**. It is a high-frequency approximation that computes the currents on the surfaces from the **Physical Optics** technique, so the iterative process of the resolution

method is not carried out. The normal vectors of the geometry must be pointing to the proper direction.

  - **MoM (Method of Moments)**. It is an accurate full wave method that may be used to solve all types of situations. Selecting the **Method of Moments** solver, the **Subdomains** option with the **MLFMA-MoM (Multi-Level Fast Multipole Algorithm – Method of Moments)** technique is applied. This is the most conventional technique.

- **Architecture Strategy** method to divide the problem based on hardware properties. Two different options are available:
  - **MPI (Message Passing Interface)**. It is valid for all types of computer architectures, but more memory may be required for shared memory machines.
  - **OpenMP**. It is only valid for shared memory machines, where it may require fewer memory resources than the **MPI** strategy.
  - **Hybrid (MPI + OpenMP)**. This architecture uses both MPI + OpenMP. It is commonly used with computational clusters.
- **Electromagnetic Equation** equation used to solve the solution. Further information about this options is explained in Method of Moments.
  - **EFIE** (Electric Field Integral Equation)
  - **MFIE** (Magnetic Field Integral Equation)
  - **CFIE** (Combined Field Integral Equation)
- **Solver Function**: The selected solver function is used to set the electromagnetic technique being used through the simulation process:
  - If the subdomains option is chosen, the MLFMA-MoM (Multi-Level Fast Multipole Algorithm) will be used. This is the most conventional technique.
  - If the macro basis functions (CBFs Radome) option is selected, then the CBFM-MLFMA (Characteristic Basis Function Method - Multi-Level Fast Multipole Algorithm) is used instead. This option is only available for the analysis of antennas composed by radomes.
- **Relative Error** it is the maximum value error allowed in the iterative process. When the relative error of any iteration is lower than the value specified, the current computation stay is considered as a valid solution and the iterative process is finished. The smaller is the **Relative Error** the more accurate is the provided solution, but the larger is the computation time.
- **Maximum number of iterations** maximum number of iterative steps used to search an iteration that satisfies the specified Relative Error. If the **Maximum number of iterations** is reached without getting a valid solution, the last iteration solution is saved.

Click on **Save** button to confirm the updated configuration.

Method of Moments

In the **Method of Moments** option, different parameters can be selected to configure the analysis.

Firstly, the user needs to choose the **Electromagnetic Equation** to solve:

- **EFIE** the **Electric Field Integral Equation** is considered. This options solves the most of problems and is the most accurate one, but the convergence may be slow or even it may not be achieved in some cases.

- **MFIE** the **Magnetic Field Integral Equation** is considered. This option provides valid results only for closed bodies, whenever the normal vectors of the geometry is pointing to the proper direction (outside the objects).
- **CFIE** the **Combined Field Integral Equation** is considered. It is also recommended only for closed bodies, whenever the normal vectors of the geometry is pointing to the proper direction (outside the objects), and it uses a weighted combination of the two previous methods. The **CFIE parameter** must be defined between 0 and 1, and it weigths the rate in which the **EFIE** is considered. A value close to 1 is recommended for this parameter to ensure accurate results provided by the **EFIE** solution, but with the benefits of speeding-up the iterative process with the **MFIE** formulation (0.8 or 0.9, for example). The **CFIE** solution combines the two previous methods according to the next formula: **CFIE = EFIE · CFIEparameter + (1 - CFIEparameter) · MFIE**

Advanced Solver Options

Solver functions have custom settings that can be changed by selecting the **Advanced Options** button in the Solver Functions. Pressing this button shows a dialog with two visible tabs (three if MacroBasis Function method is selected):

- **Main Properties**: This tab has settings for changing how the solver works, including which kind of algorithm will the solver use.
- **Preconditioner**: This tab contains available preconditioners for the selected solver.
- **CBFs Properties**: This tab is only enabled when the solver functions are set to CBFs.

The Main Properties tab looks like this:

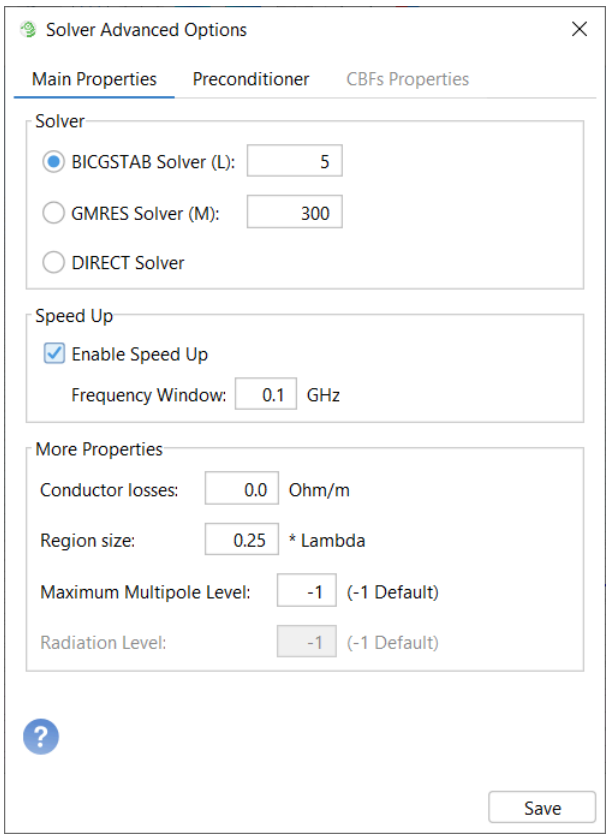


Figure 1089: Solver Advanced Options. Main Properties tab

- **Solver**: Three algorithms to solve the problem are available. We can choose between two iterative methods, like BICGSTAB (BiConjugate Gradient STAbilized method) and GMRES (Generalized Minimal Residual method), and the DIRECT solver. If no convergence is achieved by using any of the iterative methods, it is recommended to try to use the other one.

  > **Note:** The direct solution method may require huge memory and time resources when a large number of unknowns is considered.

- **Speed Up**: If the user enables the speed-up, this option reduces the analysis time for problems that have several frequencies.
- **More Properties**: The user can specify the next parameters
  - **Conductor losses**: the metallic structures may induce the conduction losses defined by its parameter, specified in Ohm/m. By default, no conductor losses are considered.
  - **Region Size**: this parameter defines the size edge (in terms of **wavelengths**) of the regions generated in the **MLFMA-MoM (Multi-Level Fast Multipole Algorithm – Method of Moments)** algorithm.
  - **Maximum Multipole Level**: it is an advanced parameter that defines the maximum number of levels considered in the **MLFMA-MoM** solver to consider the coupling effect. The default value (-1) consider the coupling in every levels, whereas an integer positive value specifies that the coupling is only considered between the regions up to this level. The more levels consider the coupling effect, the more accurate is the provided solution but also the slower

is the solution process. **Radiation Level** this parameter sets the maximum radiation level in the multipole generation to obtain the radiated fields. The default value (-1) let the program to adjust automatically this configuration. For very large simulations, it may be used to save memory and time resources by avoiding the computation of radiated far field in the largest regions.

- **Compute 3D Pattern**: The **3D Pattern** is a spherical diagram that shows the field distribution of the analyzed problem. The resolution of the spherical diagram may be modified by the user with the **Angle Step** parameter (in degrees), that specifies the angular step taken into account in the diagram computation. Disable the **Compute 3D Pattern** to avoid the **3D Pattern** generation whenever it is not required, the simulation time may be reduced.

Preconditioner tab:

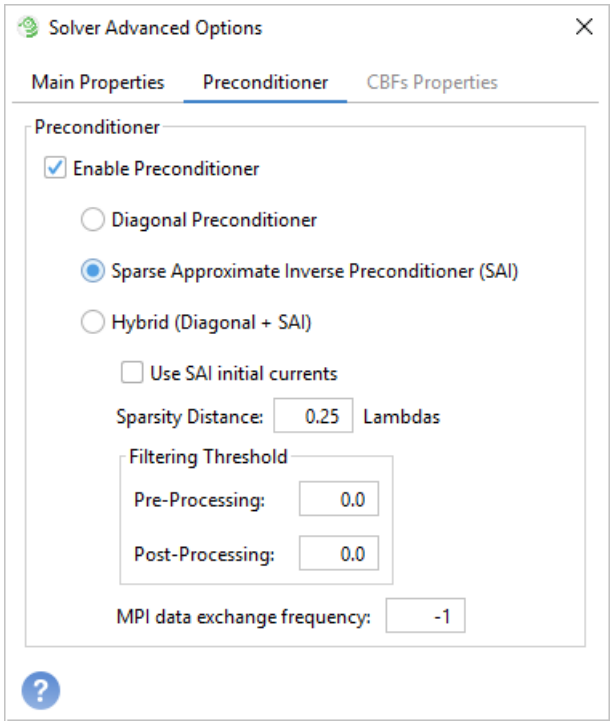


Figure 1090: Solver Advanced Options. Preconditioner tab

In the subdomains analysis, the user can enable the preconditioner to speed up the resolution of the problem with the **Enable Preconditioner** option. The user can choose between two different preconditioners:

- **Diagonal Preconditioner** The diagonal preconditioner is fast to compute and requires a reduced amount of memory, although the improvement in the convergence rate it produces is normally moderate. This preconditioner it is only recommended when more than 8 divisions per wavelength is set in the meshing process, as a shorter number of divisions slows down the convergence instead not using this preconditioner.
- **Sparse Approximate Inverse Preconditioner (SAI)** This preconditioner will generally result in a faster convergence than the diagonal preconditioner.

  - **Use SAI initial currents**: This option set the currents computed using the SAI preconditioner as the initial vector of the iterative method. It may be useful if not convergence is achieved in the solution of the problem.
  - **Sparsity Distance**: This parameter is expressed in wavelengths (0.25 as the default value) and indicates how accurately this preconditioner will resemble the inverse of the rigorous MoM matrix. Higher values will normally involve a faster convergence, but the memory required to store the preconditioner data will grow fast, non-linearly. We advice to keep the default value or increase it slightly in case of specially ill-conditioned systems.
  - **Filtering Threshold**: These parameters should contain a value between 0.0 and 1.0. The default values should be adequate in most cases.
  - **Pre-Processing**: This parameter controls the amount of data considered to generate the preconditioner. Lower values entail a more accurate generation, while higher values entail a faster computation.
  - **Post-Processing**: This parameter controls the amount of data to be stored after the generation of the preconditioner. Lower values entail better convergence, while higher values entail less RAM required to store the preconditioner.
  - **MPI Data Exchange Frequency**: This parameter sets up how often the MPI nodes request more coupling terms to generate the preconditioner. Larger values require less interactions speeding up the simulation, although more memory will be needed to store these terms. A negative or 0 value indicates that the coupling terms are only exchanged once.
- **Hybrid (Diagonal + SAI)** : This preconditioner is suited for memory-shared machines, in which case will use both Diagonal and SAI preconditioners.

Due to its numerical nature, the **SAI preconditioner** is better suited for the case of shared memory parallelization (**OpenMP**), while the conventional diagonal preconditioner can be used either for **OpenMP** or for the **MPI** paradigm.

CBFs Propierties tab (available only for antennas with radome structures):

When **Macro Basis Functions (CBFs Radome)** option is enabled in the panel **Solver Functions** in the **Solver** menu, the following options are available as well:

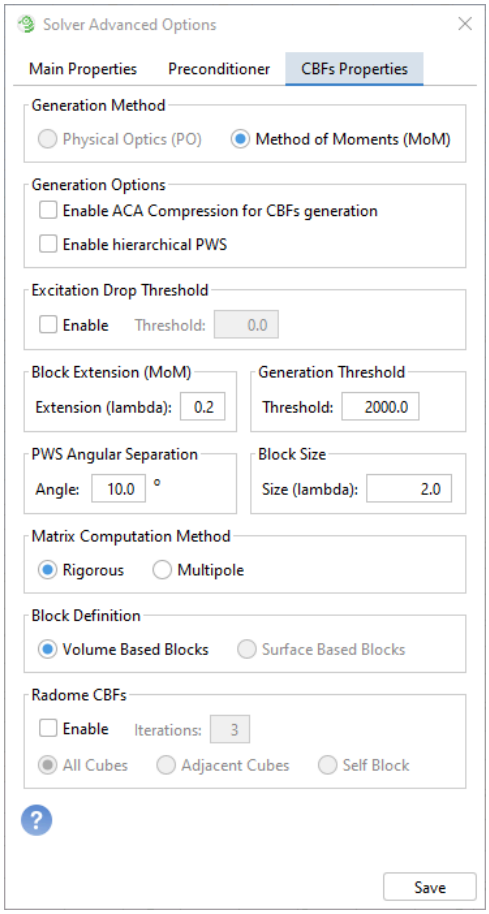


Figure 1091: Solver Advanced Options. CBFs Properties tab

These settings let the user configure different parameters of the CBFM method. For most of the analysis, default parameters are suitable.

- **Generation method**: CBFs can be generated using PO currents or MoM currents (default MoM). In this last option, it is desirable to extend the size of the block to avoid the edge effect. The extension is selected in the **Block Extension (MoM)** panel (by default, 0.2 lambda).
- **Generation Options**: There are several techniques that could speed up the generation of CBF. In this panel, you can select the compression of the PWS using the ACA technique, the hierarchical computation of the CBFs by dividing the PWS into groups.
  - **Enable ACA Compression for CBFs generation**: This check box enables a compression algorithm using ACA in order to reduce the number of excitations used to obtain the CBFs. This will decrease the total simulation time.
- **Excitation Drop Threshold**: With this method, a second stage is implemented to discard cbfs. In this case, only CBFs that get a significant impressed field due to external sources are preserved. If used, a threshold of 0.01 is recommended. Lower values give more accuracy.
- **Generation Threshold**: It is used to set how many cbfs should be retained to solve the problem (Default 2000).
- **PWS angular separation**: Defines the angular separation between two plane waves in the PWS, which determines how many plane waves will be used in generating the PWS. (By default, 10º).

- **Block size**: This parameter defines the size of the block of the CBFM method in lambdas.
- **Matrix calculation method**: You can define the method to calculate the reduced matrix of CBFM, using rigorous calculation or multiple approximation.
- **Block definition**: This option allow to choose the definition of the block. "Surface-based blocks" define a block that includes only subdomains inside of a MLFMM region that belongs to the same surface. "Volume based blocks" define a block as the set of all subdomains inside of a MLFMM region.
- **Radome CBFs**: This parameter defines the CBFs terms considered when solving the radome problems.
  - **Enable**: Activates the Domain Decomposition approach for the analysis of Radomes using CBFs and separating the computation of the antenna and the radome. Three options are available:
    - **All Cubes**: A MLFMA-CBFM full-wave analysis is performed to solve the radome problem.
    - **Adjacent Cubes**: Only the coupling terms associated to MLFMM-cubes that are adjacent are considered to solve the radome problem.
    - **Self Block**: Only the coupling terms associated to CBFs that belongs to the same MLFMM-cube are considered.
  - **Iterations**: It defines the number of interactions between antenna and radome that should be considered to obtain the solution.

11.7.2 Advanced Options

In this submenu, the user can access the options explained below:

- **Specify EFIE**: this option is only taken into account when the **Electromagnetic Equation** of **CFIE** is selected on the **Solver Parameters** menu.

  It is recommended when the geometry contains open and closed entities, because only the closed objects may be analyzed with the **CFIE** formulation, and the open ones are analyzed with the **EFIE** formulation.

  This option opens a table list with all the names of the objects that are going to be analyzed with **EFIE**. Selecting the objects on the geometry panel and clicking on **Add** button, the object names are added to the **Objects with EFIE** list. Use the **Remove** button to delete objects from the list.

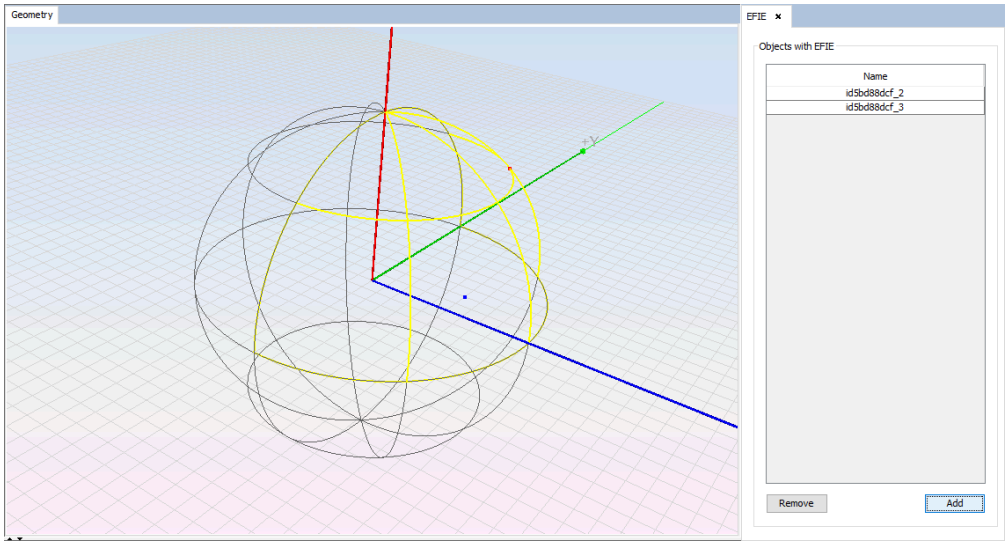


Figure 1092: Solver - Specify objects with EFIE

Specify Divisions

using this option, the user can force the sampling density to be applied to any object, which will be used in the meshing process. It is recommended when the most of the geometry may be meshed with the same density - which will be set in the meshing configuration -, but some regions of the geometry require different meshing resolutions.

This option opens a table list with all the names of the objects that are going to be meshed with a different number of divisions than the meshing configuration together with the value of its divisions. Selecting the objects on the geometry panel and clicking on **Add** button, the object names will be added to the **Objects with divisions** list and then the number of divisions for it may be edited. To edit the number of divisions of a given object, select its row, introduce the desired number of divisions in the field left to the **Fill value to selection** button, and then click on the button. Use the **Remove** button to delete objects from the divisions' table.

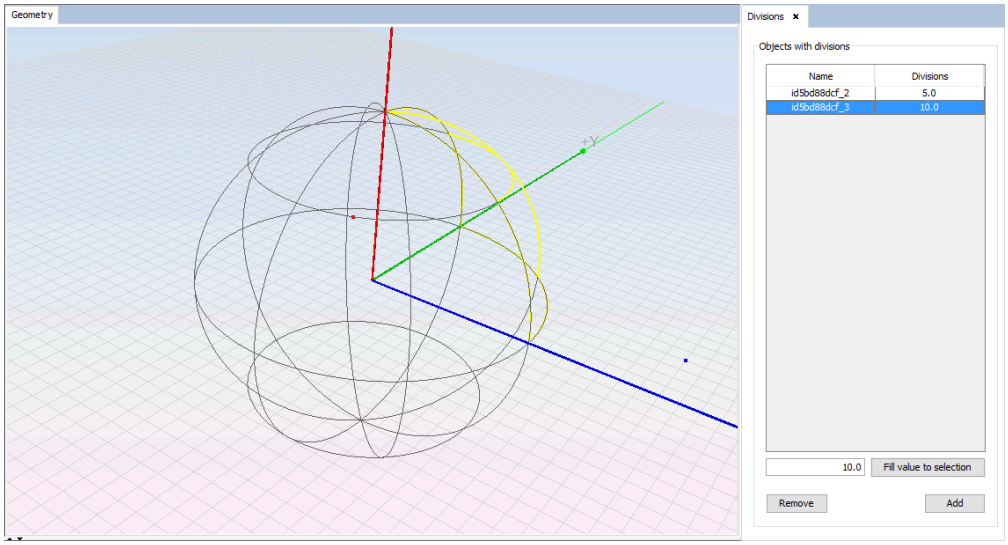


Figure 1093: Solver - Specify divisions

Surfaces to not duplicate

this option is only taken into account when the **Replicate Open Surfaces** of **Surfaces Replication** is selected on the **Meshing Advanced Parameters** menu (see Meshing Menu). It may be useful to analyze open objects as volumes that are automatically approximated by two parallel thin layers, and then apply the **CFIE** formulation on them. Using this option, the user can select which objects are analyzed only with their normal vectors orientation, so they are not going to be duplicated.

Selecting the objects on the geometry panel and clicking on **Add** button, the object names will be added to the list of object which will be not duplicated. Use the **Remove** button to delete objects from the table.

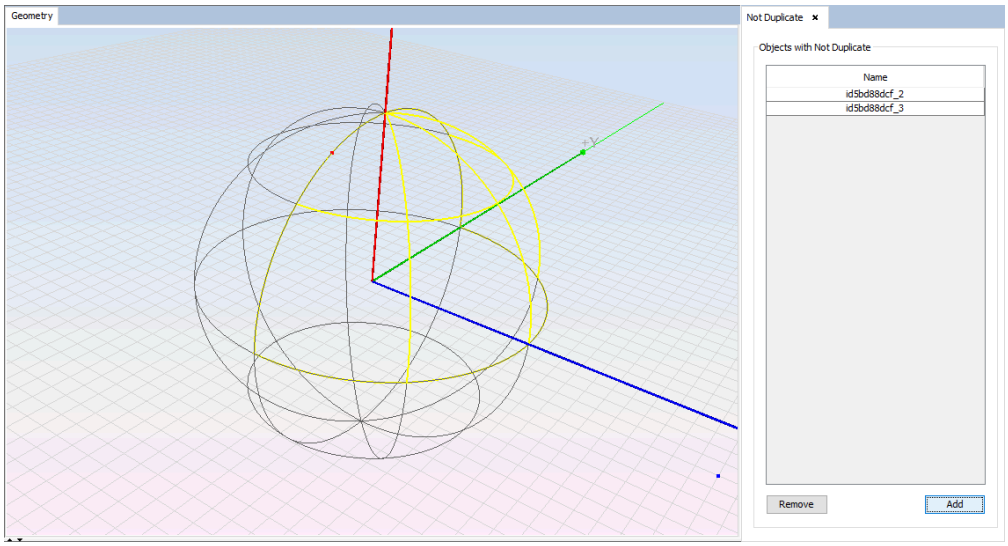


Figure 1094: Solver - Specify objects with no duplication

11.8 Source

For information about the Source menu, refer to Antennas.

11.9 Output Menu

The output results to obtain may be specified within this menu, such as the coupling between the available antennas, observation directions in spherical coordinates and observation points.

11.9.1 Passive Antenna

With this option, the user can enable the coupling effect between antennas to obtain results of this kind. Selecting **Output - Passive Antenna** and the following panel will be displayed to configure the passive antenna parameters.

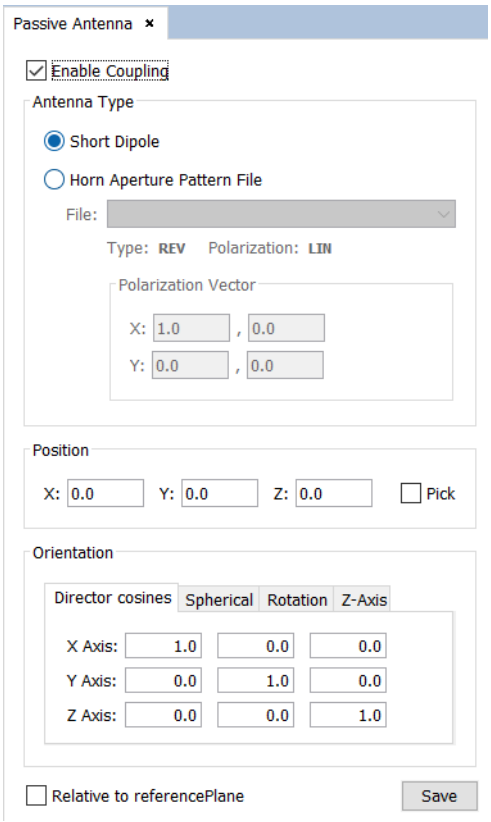


Figure 1095: Passive Antenna Panel

The following parameters may be edited:

- **Enable / Disable Coupling** the check box on the top of the window can be enabled or disabled. This option allows computing the mutual coupling between active and passive antennas.
- **Antenna Type** the passive antenna can be:
  - **Short dipole** creates a logical dipole with default length.
  - **Radiation pattern file**. The format of the radiation pattern file can be **REV**, **RV2** or **3DE**, and the polarization can be linear (**LIN**) or circular (**CIR**). The name of the pattern file has to be indicated. See DIA Files to read further information regarding the **DIA Files**.

- **Position-Coords** location of the passive antenna (from the keyboard or using the mouse with ' **Pick Point Edition**' option).
- **Polarization Vector** polarization given by its real and imaginary parts for each coordinate.
- **Orientation Director Cosines** orientation of the passive antenna coordinate system related to the absolute coordinate system.

If all the information is correct, the antenna will be shown on the screen. A red parabola represents the active antenna and a magenta parabola the passive antenna. The axis of the antenna is also visualized. If the antenna is not visualized on the screen after pressing **Save** button, the most common reason is that the radiation pattern has not been found in the newFASANT installation folder.

11.9.2 Observation Directions

Clicking on the **Output** -> **Observation directions** menu will show the following panel:

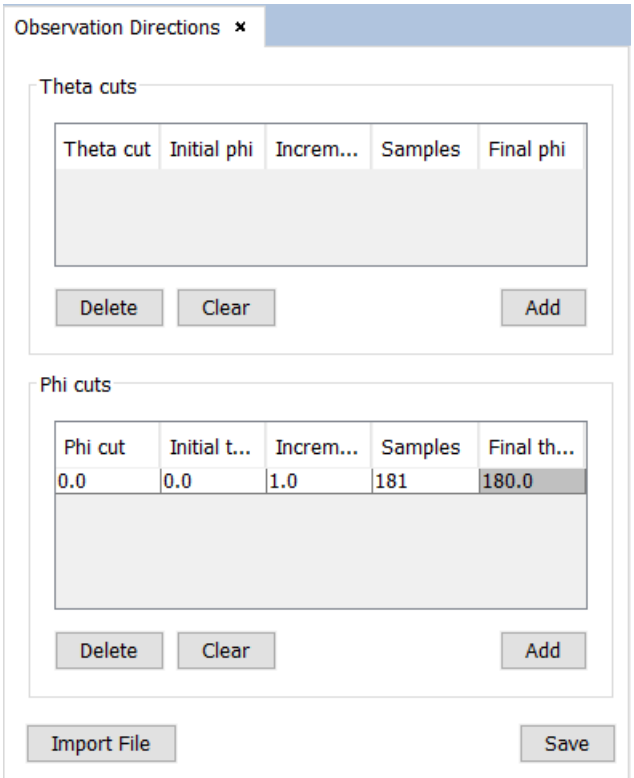


Figure 1096: Observation Directions panel

The user needs to configure the directions of observation where the electrical far field is going to be calculated. Given a **Theta** or **Phi** table, the following parameters may be modified:

- **ThetaPhi cut** defines the value of the angular cut (in degrees) in the current table.
- **Initial PhiTheta** defines the angular initial value (in degrees) of the other component in the current table.
- **Increment PhiTheta** defines the angular step (in degrees) between adjacent samples in the current cut.

- **Samples** column describes the number of steps for the row cut. It is an integer value.

The **Final Phi/Theta** value is automatically computed according to the previous configuration.

Insert **Theta** or **Phi** cuts by clicking on the **Add** buttons of the corresponding section, and remove undesired cuts by using the **Delete** buttons. **Clear** buttons removes all cuts from table.

Press **Save** button before closing this window to confirm the changes.

The user can import a file of directions using the **Import File** button. The file to be imported must be saved with `.txt` extension and the following format is required:

- Each direction is defined by Theta Phi angles.
- Every direction to be imported must be defined in separated lines.

Considerer an example file (`cuts.txt`) to plot with the following content:

```
10.0 10.0
10.0 20.0
20.5 35.0
45.0 0.0
```

After importing the file, a new cut is added to Theta cuts table for each direction.

11.9.3 Observation Points

This panel allows setting observation points in near field. When this option is selected, the following panel is shown:

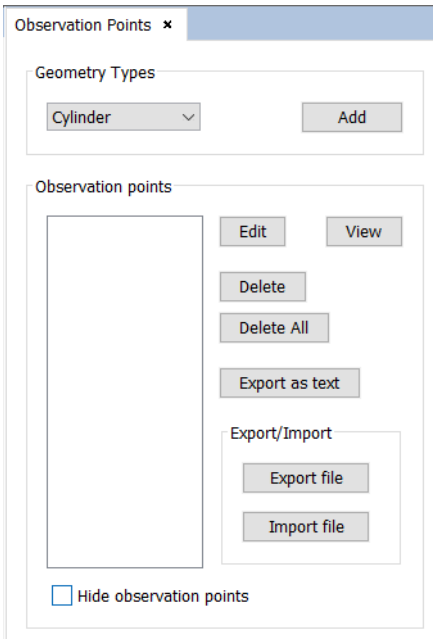


Figure 1097: Observation Points panel

The user can select near field observation points in different ways. The next options are available:

- **Geometry Types** observation points may be inserted as group that define different shapes. Select the desired type entity and click on **Add** button to set its parameters. The available **Geometry Types** are detailed in next subsections. Before adding new elements, check that the **Name** of the new entity to be added is different of the already existing ones; otherwise an error will be shown.
- **Observation points** this section contains several options to work with observation points.
-
  - The left list contains all the observation shapes that has been added as observation points.
  - The **Edit** button allows editing an observation points shape selected on the left list.
  - The **Delete** button allows removing the observation entities selected on the left list.
  - Use the **Delete All** button to remove all the observation points shapes on the left list.
  - The **Export as Text** button allows saving all the generated observation points. Every point is defined in a different line by its X, Y and Z coordinates.
  - The **Import File** allows to add a previously saved observation points file. Only `.obs` files are supported, which are the internal format generated by the application.
  - The **Export File** allows to save the current observation points configuration in a new file. It must be saved as `.obs` file, which are the internal format generated by the application.
  - Check the **Hide observation points** option to hide the observation points from the main geometry window but without deleting them. Disable this option to show the observation points.

Cylinder

The observation cylinder may be defined as a closed cylinder (circular section) or only a cylindrical sector (arc section). The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Base Center** coordinates where the base of the cylinder will be placed (in the XY plane).
- **Radius** radius of the cylinder
- **Height Coordinates** to define the hight sampling rate of the cylinder.
  - **Number of points** define the number of samples to be generated along the height of the cylinder (in the Z axis).
  - **Height** defines the total height of the cylinder.
- **Angle Coordinates** to define the circular sampling rate of the cylinder.
  - **Number of points** define the number of samples to be generated along the transversal section of the cylinder (in the XY plane).
  - **Initial angle** angle where the first circular sample is generated in the transversal section, in degrees.
  - **Final angle** angle where the last circular sample is generated in the transversal section, in degrees.

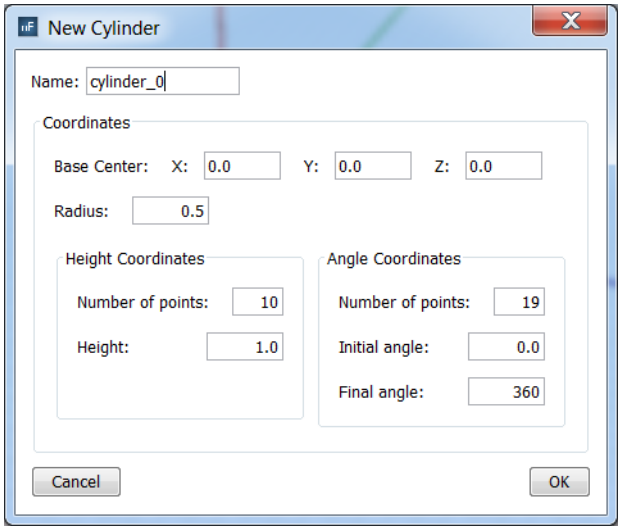


Figure 1098: New Cylinder parameters

Click on **OK** button to insert the observation points from the cylinder, or abort the insertion by clicking on **Cancel** button.

External File

The user can also import the near field points from a text file. This file can then be imported by selecting **External File** into the **Geometry Types** combo-box. Note that this option imports a file of points, but does not preserve the near field type (i.e. every point is imported as a separate type). In order to preserve the near field type, the **Export file** and **Import file** must be used instead.

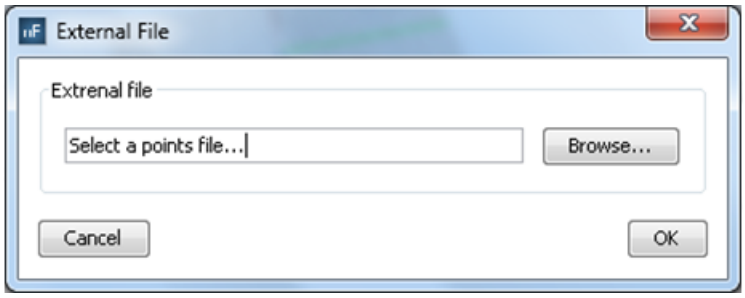


Figure 1099: Open External File dialog

The external file must contain a list of points given by its **X**, **Y** and **Z** coordinates, with every point specified in separated lines.

Click on **OK** button to insert the observation points from the file, or abort the insertion by clicking on **Cancel** button.

From Curve

This option lets you select a line or curve of the main panel and sampling it as observation points. Before clicking on **Add** button having select the **From Curve** type, the curve has to be selected in the **Geometry** panel. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Number of points** to specify the number of samples generated along the curve.

Click on **OK** button to insert the observation points from the curve, or abort the insertion by clicking on **Cancel** button.

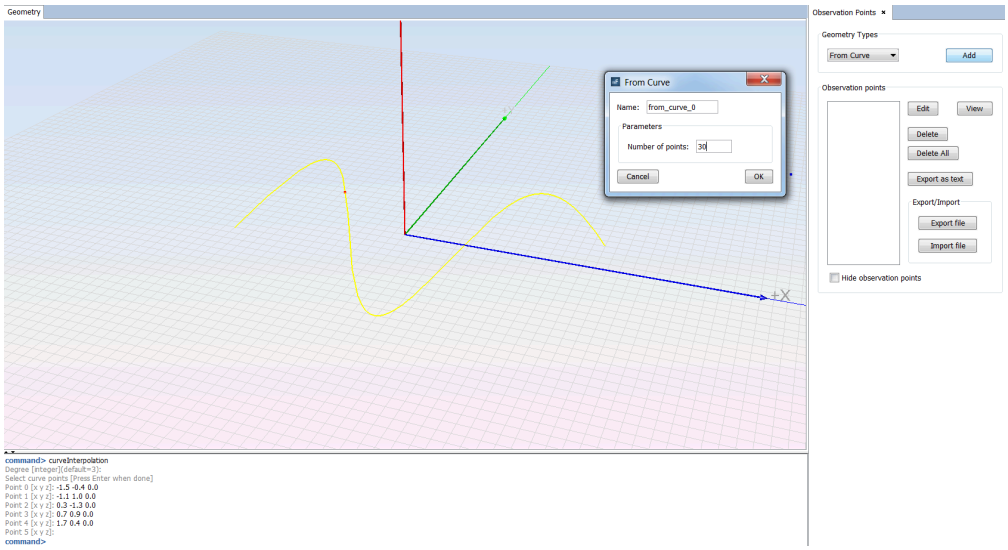


Figure 1100: Adding a curve of observation points

The next figure shows that the grid points are added to the Observation points list, and represented together the geometry as a dotted grid.

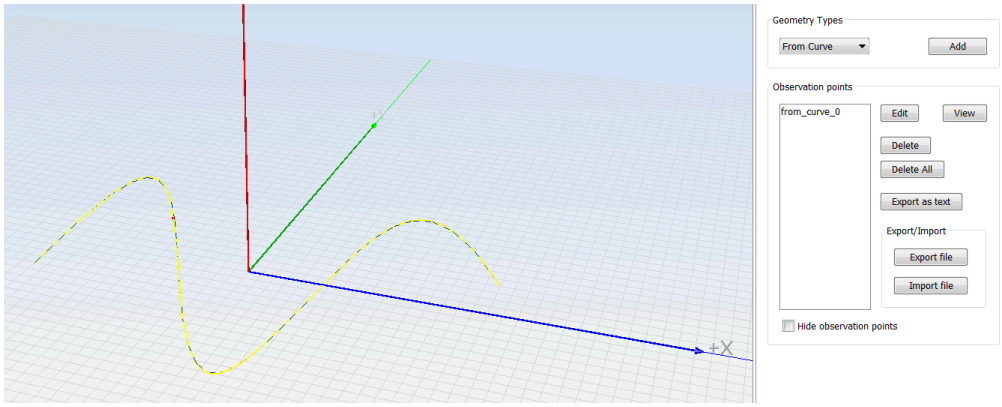


Figure 1101: Observation points generated on the curve

From Surface

This option lets you select a surface of the main panel and sample it as observation points. Before clicking on **Add** button having select the **From Surface** type, the **surface** has to be selected in the **Geometry** panel. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Parameters** the generated observation entity will be a matrix of U x V points obtained by sampling the surface.
  - **U-Dimension** to specify the number of samples generated along the U-Dimension of the surface.
  - **V-Dimension** to specify the number of samples generated along the V-Dimension of the surface.

Click on **OK** button to insert the observation points from the surface, or abort the insertion by clicking on **Cancel** button.

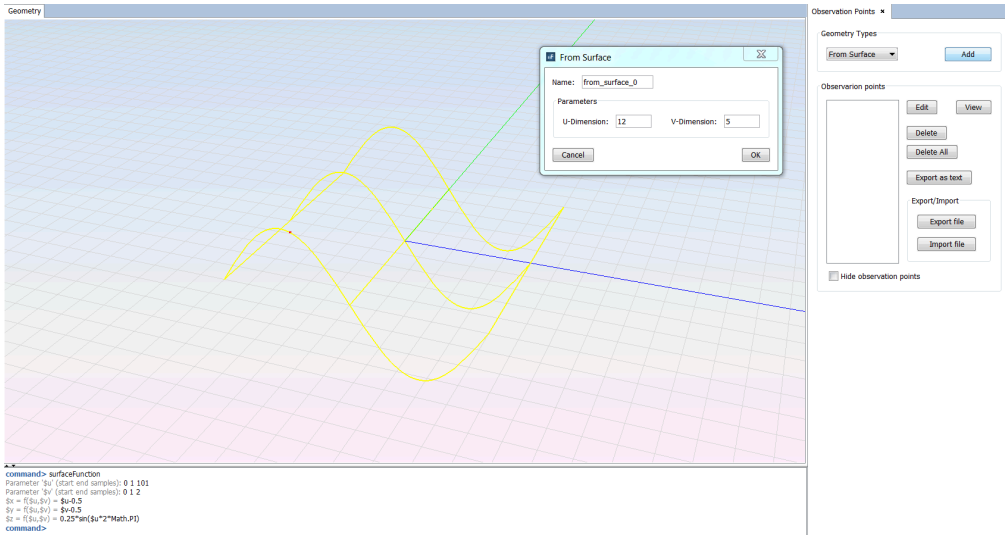


Figure 1102: Adding a surface of observation points

The next figure shows that the grid points are added to the **Observation points** list, and represented together the geometry as a dotted grid.

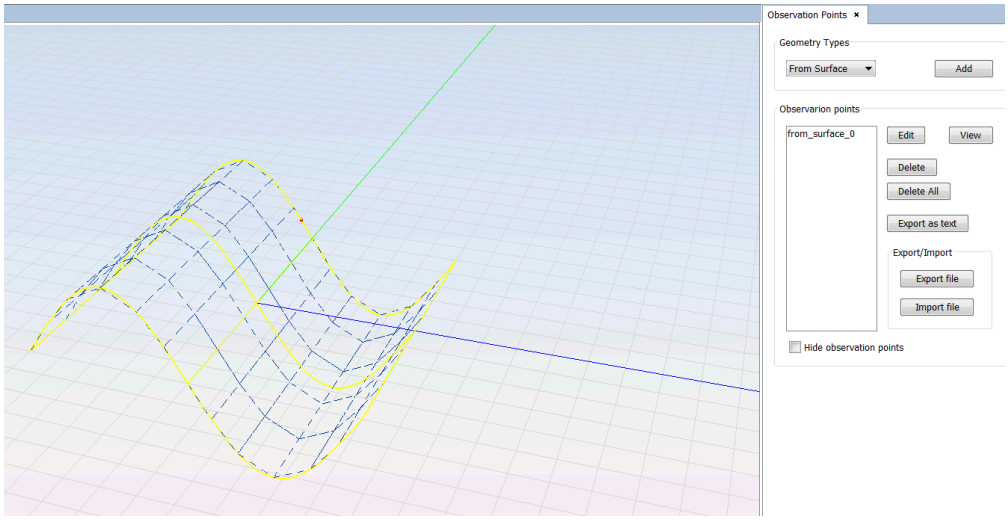


Figure 1103: Observation points generated on the surface

Line

Observation points also may be generated along a virtual straight line sampled between its extremes. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Coordinates** the line is defined by its side points.
  - **Initial Point** first extreme of the straight line.
  - **Final Point** opposite extreme of the straight line.

Click on **OK** button to insert the observation points from the line, or abort the insertion by clicking on **Cancel** button.

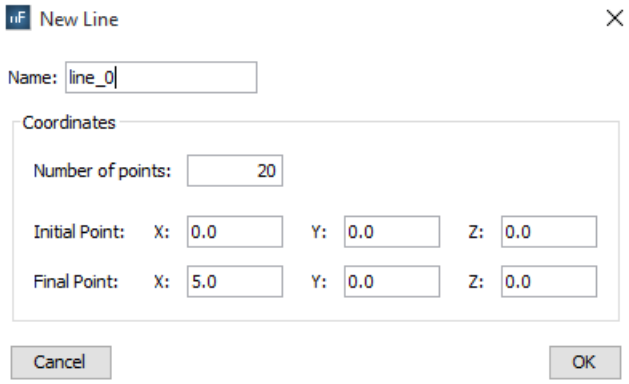


Figure 1104: New Line parameters

Parallelogram

Use this option to generate a parallelogram of coplanar observation points. The parameters required to its shape are:

- Name to define the name of this type of observation points. The name must be different of the already defined ones.
- Coordinates the parallelogram is defined by three points that form its two main axis, so the generated observation entity will be a matrix of Side x Bottom points obtained by sampling the parallelogram.
  - Side left axis is defined from **Point 1** to **Point 2**.
  - Bottom axis is defined from **Point 2** to **Point 3**.

Click on **OK** button to insert the observation points from the parallelogram, or abort the insertion by clicking on **Cancel** button.

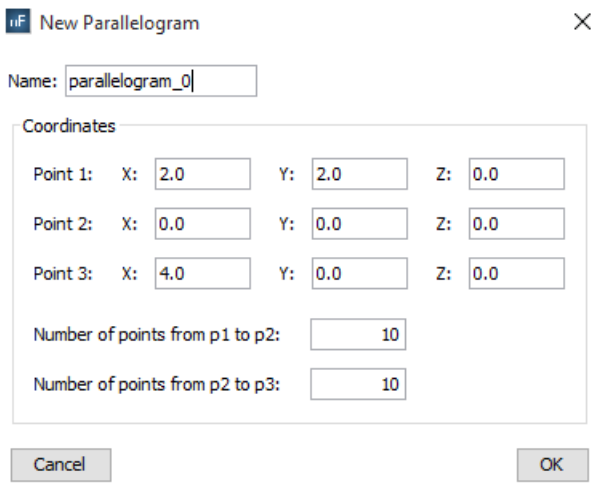


Figure 1105: New Parallelogram parameters

Plane

In this window, the user can create a set of observation points laid out in a grid pattern. To do so, the user needs to specify several parameters:

- **Name** A short name that will be used by the user to identify the created set of observation points.
- **Constant axis** The normal axis of the plane that will contain the grid of observation points.
- **Coordinate value** Value of the coordinate of the axis specified in the previous option.

For each of the axes different than the one selected in the **Constant axis** drop-down list, the user needs to specify the following parameters:

- **Initial value** Initial value of the coordinate along each axis.
- **Total size** The total length of the grid of observation points along the axis.
- **Number of points** The number of observation points along the axis.

Note The spacing between each pair of consecutive observation points along an axis will be given by the formula **spacing = (total size) / (number of points - 1)**. This value will be shown next to the label "Distance btw. points".

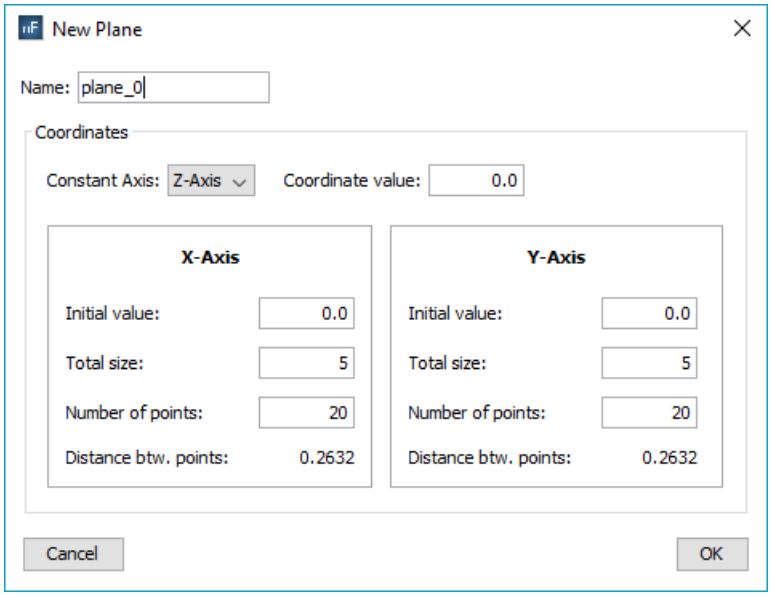


Figure 1106: New Plane parameters

Point

Isolated observation points may be generated by using this option. The parameters to specify are:

- **Name** to define the name of the observation point. The name must be different of the already defined ones.
- **Coordinates** position of the observation point, specified by its X, Y and Z coordinates.

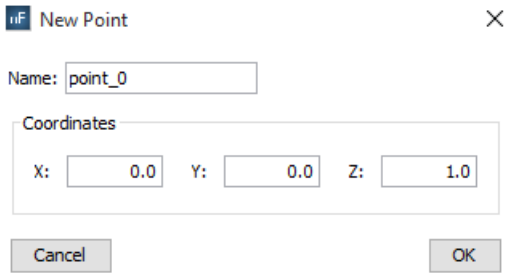


Figure 1107: New Point parameters

Click on **OK** button to insert the observation point, or abort the insertion by clicking on **Cancel** button.

Sphere

The observation sphere may be defined as a full closed sphere or only a spherical sector specified. Multiple concentric spheres (or spherical sectors) may be defined within the same entity. The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different from the already defined ones.
- **Parameters**:
- ◦ **Center Point** X, Y and Z coordinates where the sphere will be centered on.
  - ◦ **Radius** radius of the sphere. A radius sweep may be defined to generate multiple concentrical spheres.
    - · **Initial Radius** radius of the first generated sphere, which is the only one if the sweep is not configured.
    - · **Increment Radius** step between two adjacent spheres. It is only considered when the sweep is configured.
    - · **Radius Samples** number of concentric spheres to be generated, it must be an integer number. The radius sweep is enabled when this parameter is higher than 1.
  - ◦ **Theta Coordinates** to set the spherical range where te observation points are generated, in theta dimension.
    - · **Number of points** defines the number of samples to be taken in theta dimension along the specified angular range.
    - · **Initial angle** defines the angular angle (in degrees) of the first theta point to be generated.
    - · **Final angle** defines the angular angle (in degrees) of the last theta point to be generated.
  - ◦ **Phi Coordinates** to set the spherical range where te observation points are generated, in phi dimension.
    - · **Number of points** defines the number of samples to be taken in phi dimension along the specified angular range.
    - · **Initial angle** defines the angular angle (in degrees) of the first phi point to be generated.
    - · **Final angle** defines the angular angle (in degrees) of the last phi point to be generated.

The total number of observation points to be generated in the sphere is (**Radius Samples**) x (**Number of Points**) $_{\text{Theta}}$ x (**Number of Points**) $_{\text{Phi}}$.

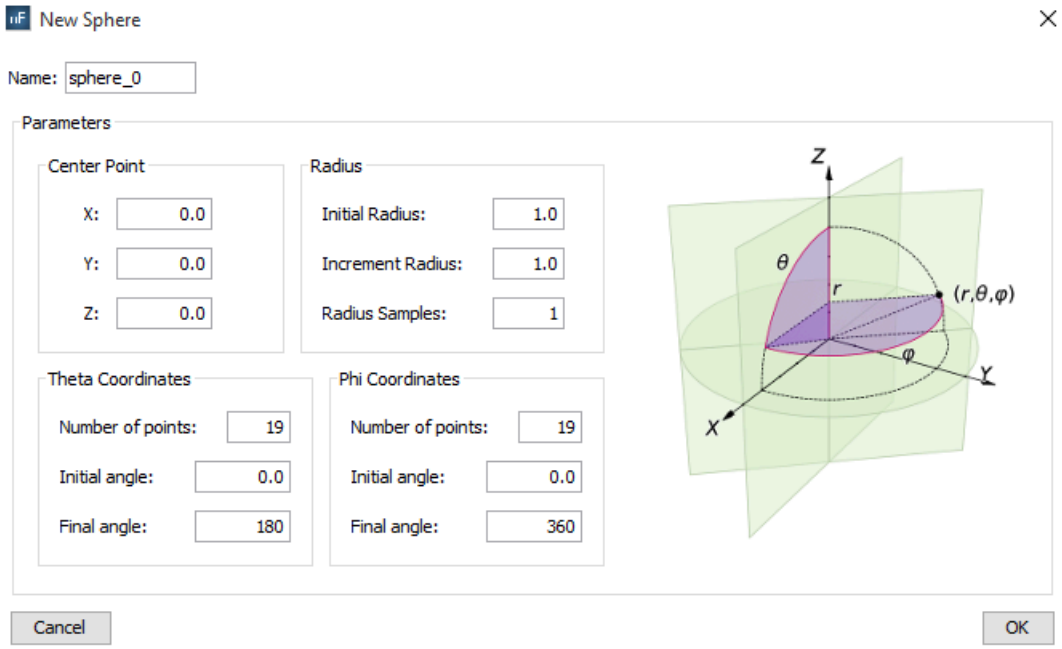


Figure 1108: Spherical observation points.

Click on **OK**button to insert the observation points from the sphere, or abort the insertion by clicking on **Cancel**button. The View Radial Points will be only enabled after the simulation process if an observation sphere has been added.

11.10 Meshing Menu

In this menu, the features of the meshing menu of the MOM module are explained. The options available in this menu are shown in the figure below:

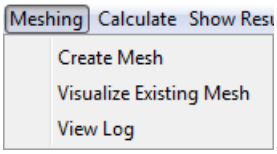


Figure 1109: Meshing menu

11.10.1 Create Mesh

The most common parameters for the meshing process may be edited in this window.

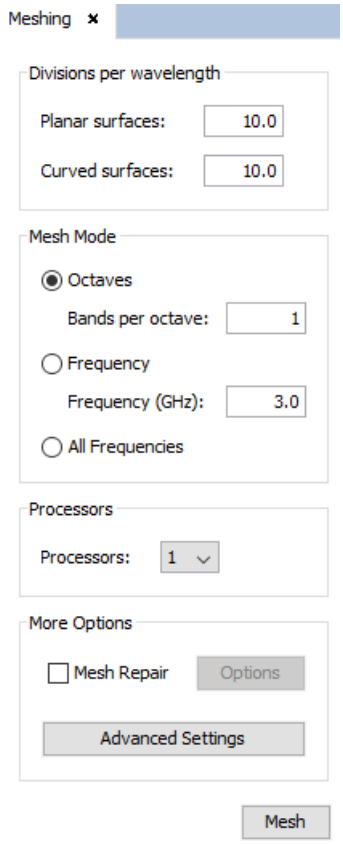


Figure 1110: Meshing panel

The following parameters may be edited:

- **Divisions per wavelength** This section specifies the size of the elements to be generated in the meshing process. Let **c** be the speed of light in the vacuum (meters per second), given a number of divisions **D** and a frequency **f**, the size of the generated elements **L** is given by: **L = lambda /**

D ; lambda = c / f The larger is the number of divisions specified the denser are the generated meshes. Different meshing resolution is considered for planar and curved surfaces. Also, a specific number of divisions may be assigned to different used by using the **Specify Divisions** option within the Advanced Options section.

  - **Planar surfaces** To set the number of divisions to be applied on planar surfaces which are contained in a single plane.
  - **Curved surfaces** To set the number of divisions to be applied on curved surfaces which are not contained in a single plane.

- **Mesh Mode** This section specifies how the meshing frequencies are assigned. It allows considering different frequencies in the meshing process than the specified in **Simulation parameters** section.
  - **Octaves** When a frequency swept is enabled, the meshing frequencies are different to the simulation ones. An automatic frequency range per octaves is performed, that depends on the **Initial frequency**, the **Final frequency** set in the **Simulation parameters** panel, and the number of **Bands per octave** specified. The higher is this parameter, the more frequencies are considered for the meshing process.
  - **Frequency** Select this option to set the desired **Frequency** for the meshing process, in GHz. If a **Frequency Sweep** has been enabled within the **Simulation parameters** panel, all the steps in the sweep will consider the mesh generated at the **Frequency** value.
  - **All frequencies:** a meshing by each frequency is built, so it is the more accurate option. **This option will only be chosen when the user needs a very accurate mesh.**
- **Processors** To set the number of processors used in the meshing process.
- **More Options**:
  - **Mesh Repair** Select this option to evaluate and repair the generated mesh. It is recommended for analyzing complex geometries, especially whenever a good convergence is not achieved. Several problems will be studied and solved isolated and spurious elements are removed, and overlapped regions are repaired. The detection of this defects on the mesh depends on two parameters that are explained below and can be edited by clicking on **Options** button. Elements with smaller area than the minimum allowed, isolated elements or parallel elements closer than a minimum distance are deleted from the output mesh. The minimum area and distance depend on the **wavelength**, the number of **divisions per wavelength** selected, and the **Area** and **Border factors**, and they are determined as:

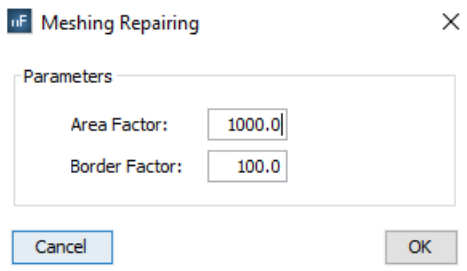


Figure 1111: Meshing Repairing options

$$MinimumArea = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)^2}{AreaFactor_{MeshingRepairParameter}} \quad (6)$$

$$MinimumDistance = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)}{BorderFactor_{MeshingRepairParameter}} \quad (7)$$

- Click on **Advanced Settings** button to edit more complex meshing parameters. The available options are detailed in next subsection.
- **Mesh** click on this button to save the parameters and start the meshing process.

Advanced Settings

The advanced meshing parameters are detailed within this section, separated in so many sections as available tabs.

Click on **OK** button to save the changes performed on every tab. Otherwise, the last configuration will not be taken into account.

General

The most common advanced parameters are resumed in the **General** section.

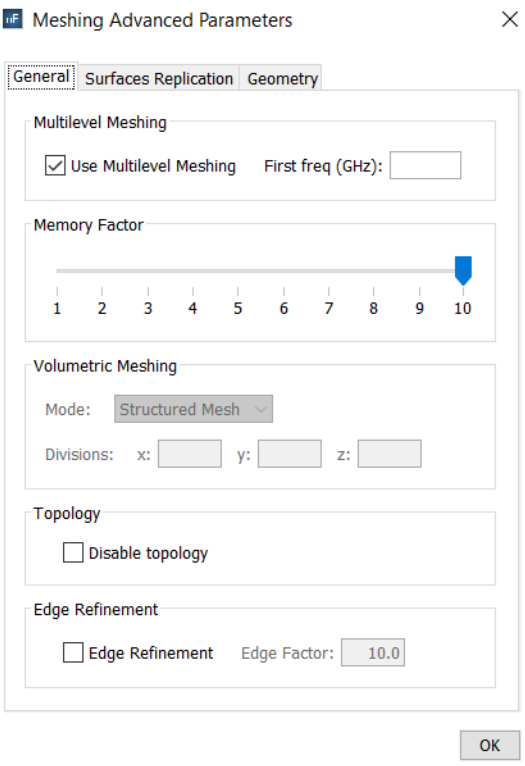


Figure 1112: General advanced parameters.

The following options are available within this panel:

- **Multilevel Meshing** This option generates the mesh automatically in several steps, and it is usually more efficient (in terms of runtime) than common mesh generation, so it is selected by default. However, minor differences may appear between the meshes obtained with and without multilevel mode.
  - **Use Multilevel Meshing** enable this option to use the multilevel mode.
  - **First frequency** The frequency of the first step may be specified by the user with this parameter. If this field is empty, the first frequency is automatically computed. It is only considered when the **Multilevel Meshing** is enabled.
- **Memory Factor** This option allows a reduction of the memory resources required in the mesh generation process. The memory allocation is an automatic feature in this process, so the highest one is selected by default to ensure that the mesh will be successfully obtained. If the memory resources exceed the available memory, the message "**ERROR Not enough memory. Please, try decreasing Memory Factor**" will appear to suggest decreasing this factor. Sometimes, a short factor may lead in not enough allocation for the required mesh resolution.
- **Volumetric Meshing** When a dielectric material is assigned to any geometric object, it may be defined as a volume, so it will be meshed with hexaedrons instead of quads. If there are any volume in the geometry, this section is enabled.
  - **Mode** This option allows to change the algorithm used in the mesh generation process of dielectric objects (defined as Volumes). Two different modes are available:

    - · **Structured Mesh** All elements of the volumetric mesh are perfect parallelepipeds. This method is an approximation of the real shape of the dielectric objects by simple cubes. Some parts of the mesh can be out of the volume and other regions can have incomplete regions, but the average volume of the mesh is very similar to the volume of the original dielectric objects. This algorithm is the fastest one.
    - · **Conformed Mesh** Most of elements of the volumetric mesh are perfect parallelepipeds, but only in the completely inner regions of them. The inner structured mesh is joined with the boundaries of the volume by using hexahedrons of variable shapes. This method is more accurate than the **Structured Mesh**, but is also slower and irregular elements may appear in specific cases.
  - **Divisions** Different divisions per wavelength for the X, Y and Z dimensions may be considered for meshing the volumetric objects. If some of this field is empty, the divisions specified in the main **Meshing Parameters** window are assigned.
- **Topology** If the electrical continuity between the surfaces have to be broken, the option **Disable topology** must be selected. Note that the accuracy may be reduced when the electrical continuities are not correctly analyzed as virtual fissures are introduced.
- **Edge Refinement** Within this section, the border effect in the mesh generation may be configured. The border effect consist on generating thinner elements in the boundary of the surfaces than the meshing size, so currents analysis is more accurate. This function is recommended for analyzing planar structures.
  - **Edge Refinement** This option allows enabling the modelling of border effect in the mesh generation.
  - **Edge Factor** It represents the portion of size of the final elements to model the border effect the higher is this value the are the boundary elements. This parameter may be higher than 3. It is only enabled when **Edge Refinement** is selected.

Surfaces Replication

During the meshing process, open objects may be replicated in the meshing process to approximate them as two thin parallel layers. It may be useful to analyze open structures by using the **CFIE** formulation, as the thin layer replication allows to consider its objects at approximate volumes.

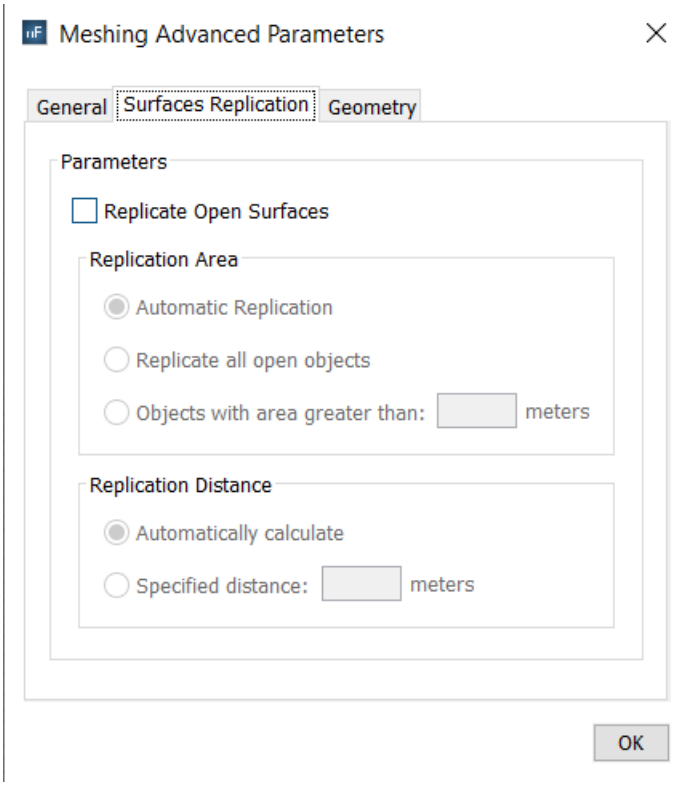


Figure 1113: Meshing advanced parameters - Surfaces Replication

The following options may be edited within this section:

- **Replicate Open Surfaces** Select this option to activate the replication and the to enable rest of parameters in the window. If replication is enabled, the elements of the open objects are replicated to an automatic frequency-dependent distance from the original ones.
- **Replication Area** This section defines the criterion to select the open objects that would be replicated or not, according to its dimensions. To force any open objects to not be replicated, use the **Surfaces to not duplicate** option included in Advanced Options. The criterion may be edited:
  - **Automatic replication** Any object with a total area greater than

$$a \geq 10 \cdot \left(\frac{lambda}{maximum(div_{plane}, div_{curve})} \right)^2 \quad (8)$$

    square meters is replicated, where **div_plane** and **div_curve** are the number of **divisions per wavelength** for **planar** and **curved** surfaces.. It is the default option.
  - **Replicate all open objects** every open objects are replicated.
  - **Objects with area greater than** To specify manually the threshold of the minimum total area of the open objects to be replicated, in square meters.
- **Replication Distance** The distance between the original objects and their parallel replicas can also be edited. Two different methods are available:
  - **Automatically calculate** By default, the distance (in meters) is given by the following expression:

$$d = 0.03 \cdot \left(\frac{lambda}{maximum(div_{planer}, div_{curve})} \right) \quad (9)$$

- **Specified distance** To specify manually the desired distance, in meters.

Geometry

Within section, several parameters may be modified to set how the input geometry is preprocessed before the meshing process. It may result on getting or not electrical continuity between surfaces, or discarding spurious shapes.

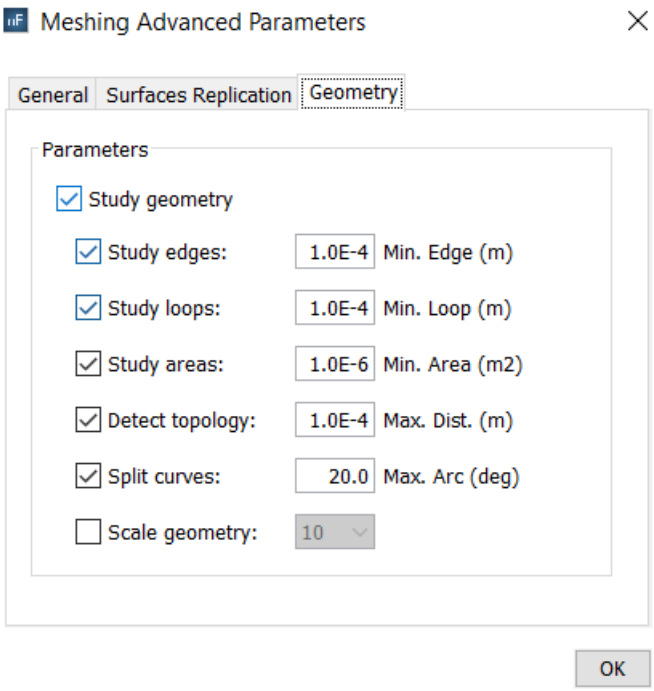


Figure 1114: Meshing advanced parameters geometry tab

Note that every option that may be enabled has at the right side its corresponding parameter to set the maximum threshold allowed. The following parameters are included in this panel:

- **Study geometry** enable this option to carry out the input geometry preprocessing. Some features such as the surface sizes or electrical continuities may be evaluated.
  - **Study edges** enable this option to delete the edges of the input surfaces that are shorter than the **Minimum Edge** parameter (in meters), extending its adjacent edges. It is only enabled when the **Study geometry** option is enabled.
  - **Study loops** enable this option to delete the loops of the input surfaces that are shorter than the **Minimum Loop** parameter (in meters). It is only enabled when the **Study geometry** option is enabled.
  - **Study areas** enable this option to delete the input surfaces that are smaller than the **Minimum Area** parameter (in square meters). It is only enabled when the **Study geometry** option is enabled.

- **Detect topology** enable this option to detect automatically electrical continuity between neighboring surfaces that are very close but have not been modeled with precise continuity. The maximum separation allowed to set the electrical continuity is the **Maximum Distance** parameter (in meters). It is only enabled when the **Study geometry** option is enabled.
- **Split curves** enable this option to divide the curved borders of the input surfaces that describe curvatures greater than the **Maximum Arc** parameter (in degrees). It is only enabled when the **Study geometry** option is enabled.
- **Scale geometry** enable this option to scale internally the geometry during the meshing process by the given factor. When the mesh generation finishes, the output mesh has the same sizes as the original geometry. This option is useful for meshing very small structures. It is only enabled when the Study **geometry** option is enabled.

11.10.2 Visualize Existing Mesh

This option can be chosen by the user to load a mesh file (`.msh` or `.mcs` file extension). When this option is selected the following window appears:

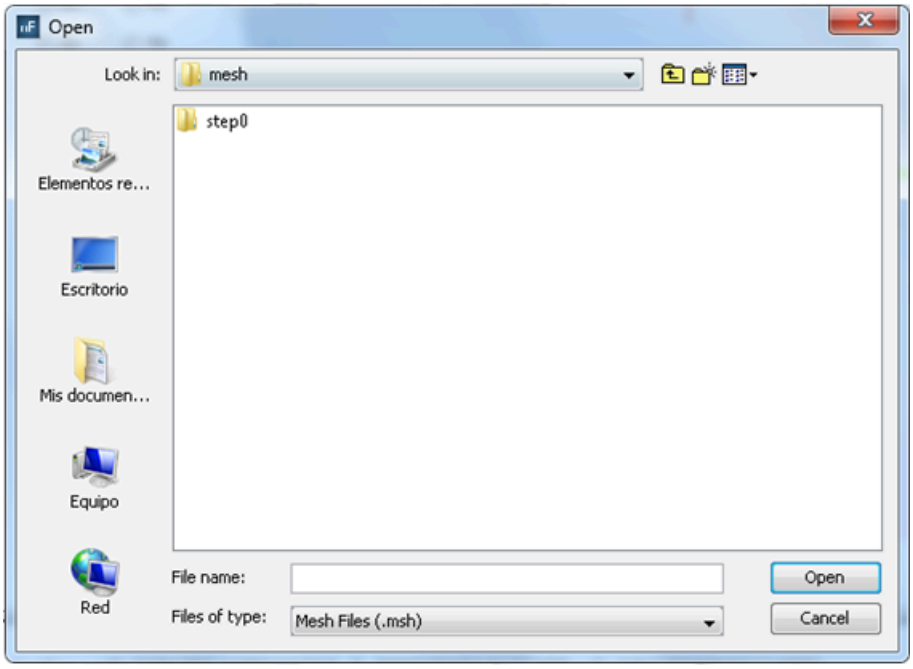


Figure 1115: Open a Visualize Existing Mesh

11.10.3 View Log

This option loads a file named `mesh_log.txt` that contains information about the meshing process. This information is printed in a new tab on the screen.

11.11 Calculate Menu

Before executing the simulation, the user needs to make sure that every simulation parameter is enabled and well-defined. If something is wrong (e.g. the geometry is not meshed) a warning message will be displayed to remember that it is necessary to properly configure the simulation data.

11.11.1 Execute

This option runs the simulation. The user can set the **Number of processors** the simulation will run on.

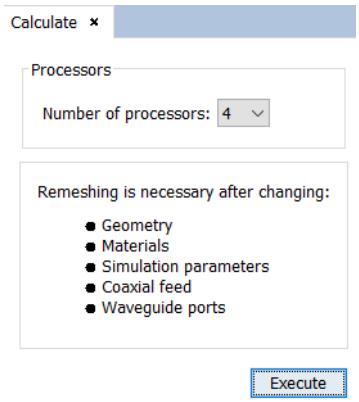


Figure 1116: Execute Panel

Click on **Execute** button to launch the simulation. Once the simulation starts, the following panel is shown:

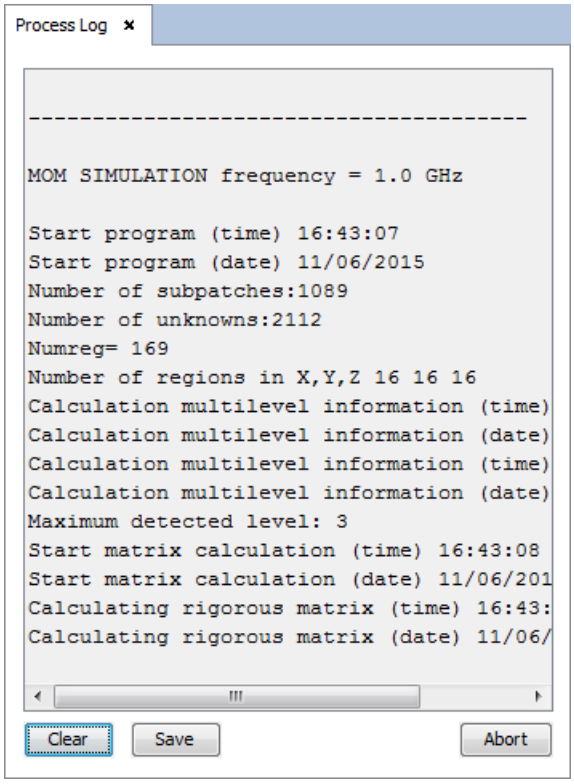


Figure 1117: Calculate, process log

In this panel, the program shows the process log of the running simulation. Several actions may be performed within this panel:

- Click on **Clear** button to remove all the text shown in the process log.
- Click on **Save** button to export the text shown in the process log to a text file.
- Click on **Abort** to cancel the current simulation.

11.12 Show Results

This menu allows the user to visualize the results of the simulation. The results are stored in external files that the user can access to obtain the corresponding information. Depending on the options selected in the **Simulation Parameters Menu** and the **Antennas Menu**, the enabled options within the results section may be different.

Most of the plots show 2-D curves of a result field versus an input parameter. This type of graphics shares additional features of zoom, axis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

In the next sections, a description of the results for the different simulation types is detailed.

11.12.1 Far Field

This menu is only enabled if at least one cut has been added within the **Output** - **Observation Directions** section. It allows to visualize the field or gain values in every far field cut defined on **Observation Directions**.

The following ways of results representation are included within this menu:

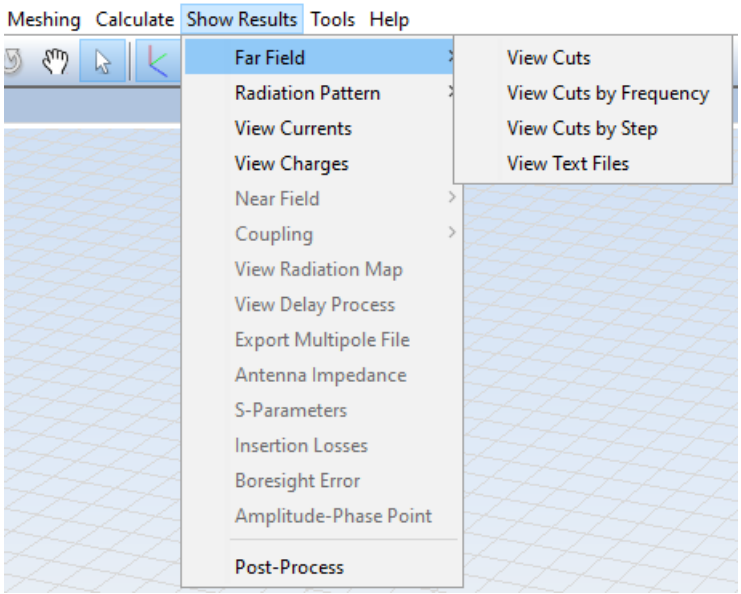


Figure 1118: Far Field options

View Cuts

Use this option to plot the results on the desired **Far Field cut**, that must be specified in the **Output - Observation Directions** menu.

When the user click on the **View Cuts** option, the following panel will appear:

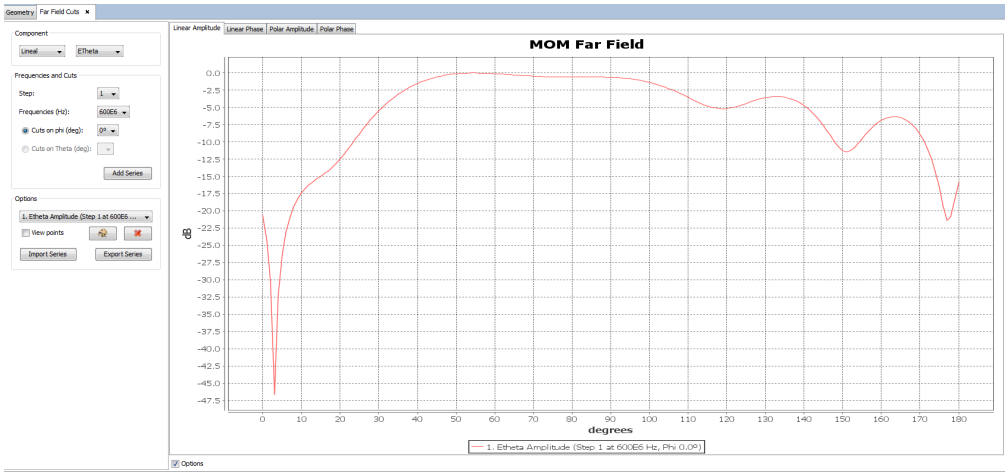


Figure 1119: View Cuts panel. Linear Amplitude

This window plots both the amplitude and the phase of the **FIELD** or **GAIN** computed for every angular cut. To plot a graph, select the desired parameters:

- **Component** The results may be plotted by using different polarizations and its components.
  - **Lineal ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.
- **Frequencies and Cuts** the far field cuts may be represented for each frequency and direction.
  - **Step** To select the geometric step in a parametric simulation. If the geometry does not contain any parameter, only one Step is available.
  - **Frequencies** To select the desired frequency in a sweep. If just a frequency has been analyzed, only one **Frequency** is available.
  - **Cuts on...Phi/Theta** so many cuts as specified in the Observation Directions may be plot by using this section.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options** section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.
- **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the X and Y values to be plotted.

- **Export Series**:To export the selected curve in the above Plot List in a `.txt` file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

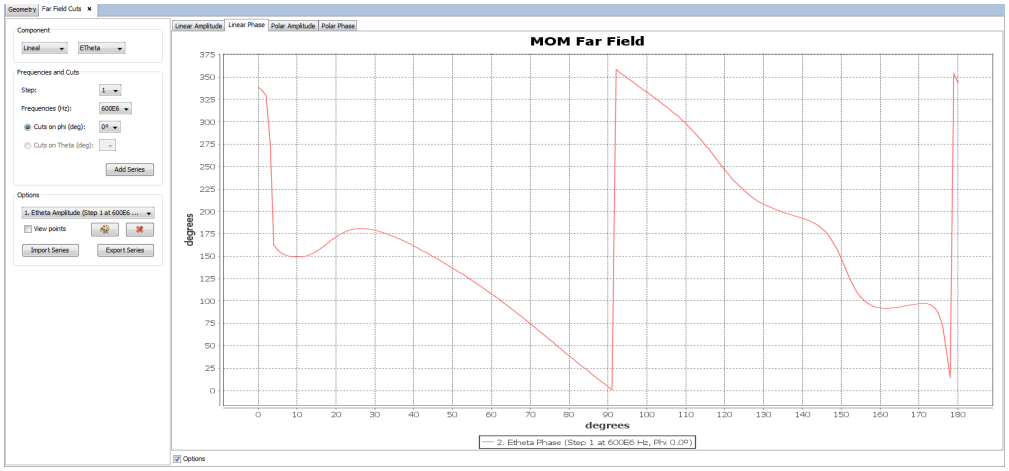


Figure 1120: View Cuts panel. Linear Phase

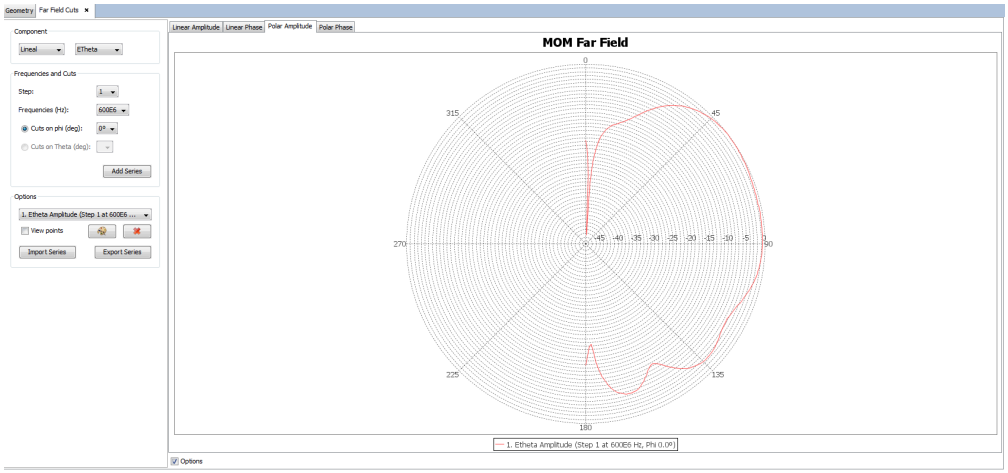


Figure 1121: View Cuts panel. Polar Amplitude

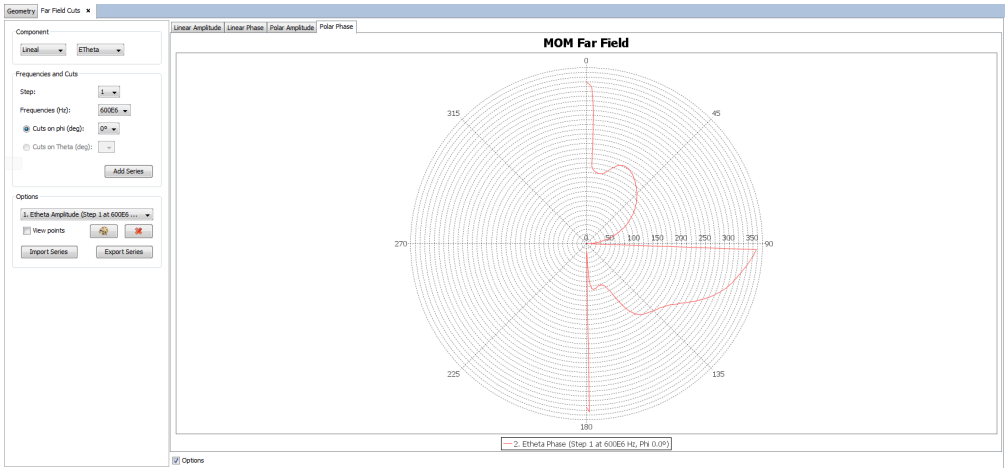


Figure 1122: View Cuts panel. Polar Phase

View Cuts By Frequency

This option is only enabled when a **Frequency Sweep** is chosen within the **Simulation Parameters**. Use it to compare the field or gain values on a **Far Field** observation direction according to the frequency.

When the user click on the **View Cuts By Frequency** option, the following panel will appear:

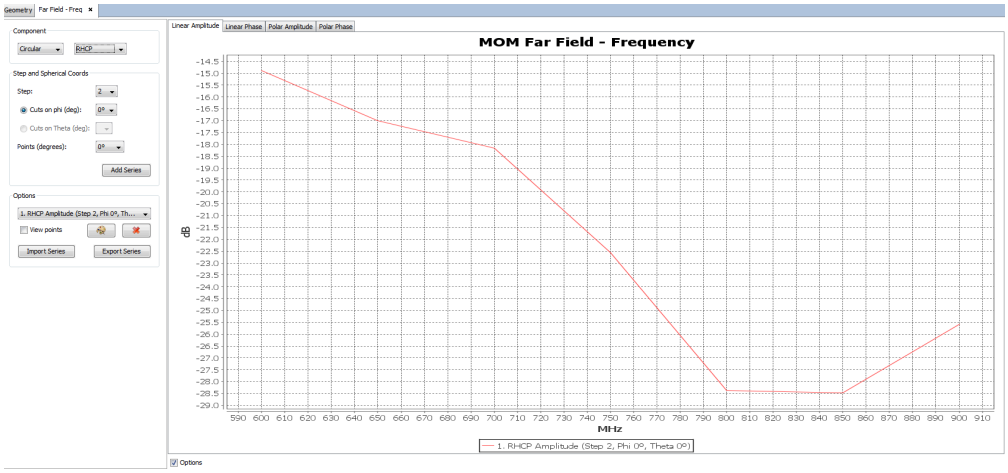


Figure 1123: View Cuts by Frequency panel. Linear Amplitude

To plot a graph, select the desired observation points:

- **Component** The results may be plotted by using different polarizations and its components.
  - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.

- **Step and Spherical Coords** the far field cuts are represented for each selected direction.
  - **Step** To select the geometric step in a parametric simulation. If the geometry does not contain any parameter, only one **Step** is available.
  - **Cuts on...Phi/Theta** so many cuts as specified in the Observation Directions may be plot by using this section.
  - **Points** Given a selected cut on **Phi** or **Theta** directions, the angular steps in the other spherical dimension must be chosen to be plot along the frequency.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options**section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.
- **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series**:To export the selected curve in the above Plot List in a `.txt` file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

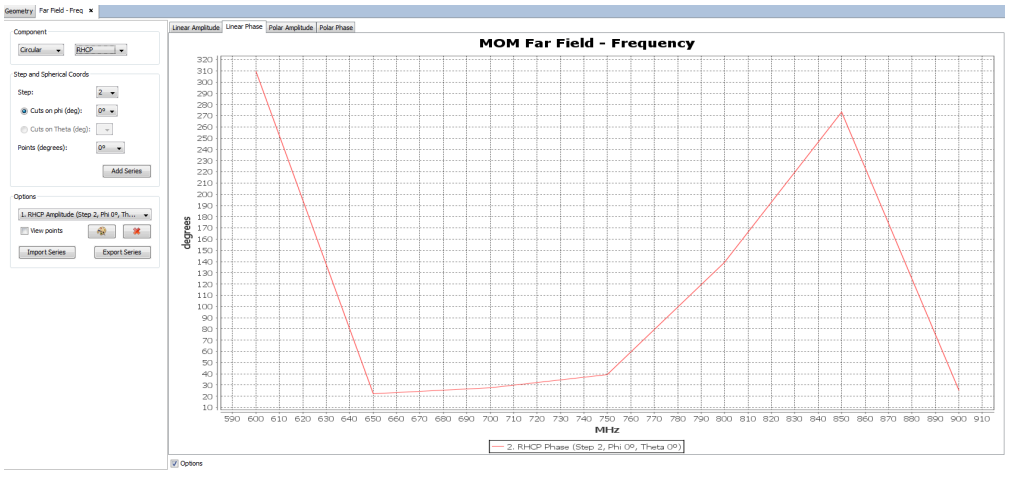


Figure 1124: View Cuts by Frequency panel. Linear Phase

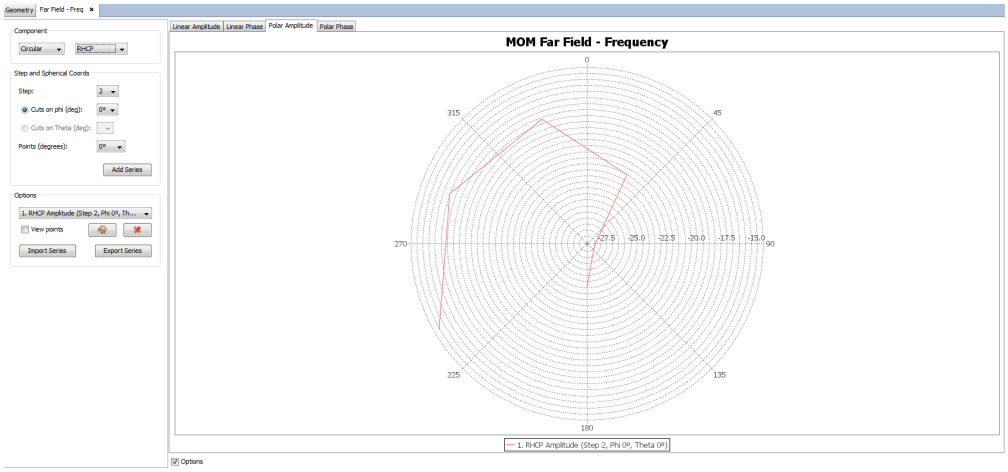


Figure 1125: View Cuts by Frequency panel. Polar Amplitude

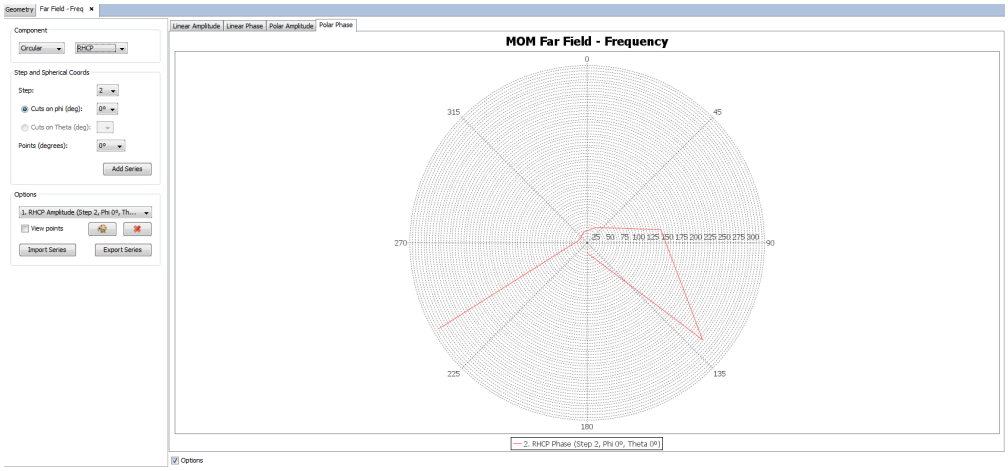


Figure 1126: View Cuts by Frequency panel. Polar Phase

View Cuts By Step

This option is useful when a **Parametric Simulation** has been performed, as the results at each **Far Field** observation direction may be compared for every parametric step. We can compare the field values in each direction at every frequency.

When the user click on the **View Cuts By Step** option, the following panel will appear:

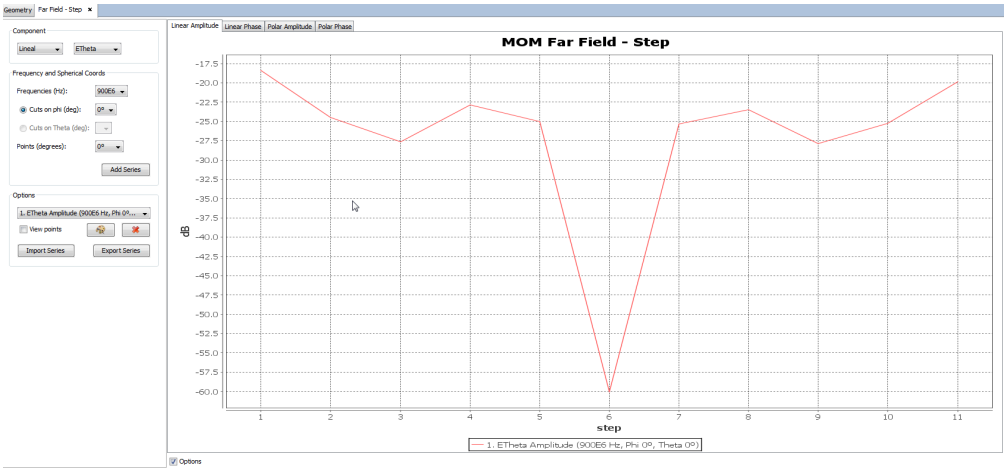


Figure 1127: View Cuts by Step panel. Linear Amplitude

Select the results to plot according to next options:

- **Component** The results may be plotted by using different polarizations and its components.
- - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.
- **Frequency and Spherical Coods** the far field cuts are represented for each frequency and direction.
  - **Frequencies** To select the desired frequency in a sweep. If just a frequency has been analyzed, only one Frequency is available.
  - **Cuts on...Phi/Theta** so many cuts as specified in the Observation Directions may be plot by using this section.**Points** Given a selected cut on **Phi** or **Theta** directions, the angular steps in the other spherical dimension must be chosen to be plot along the frequency.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options**section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.
- **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series** To export the selected curve in the above Plot List in a `.txt` file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

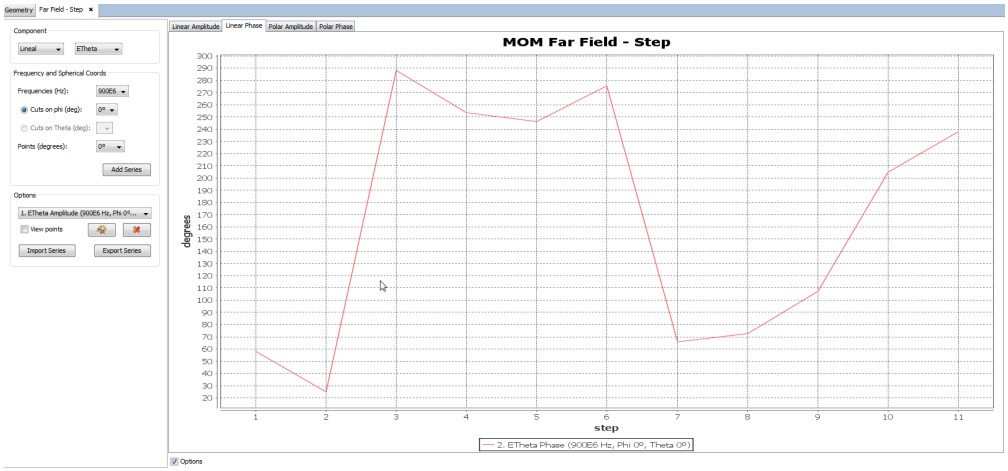


Figure 1128: View Cuts by Step panel. Linear Phase

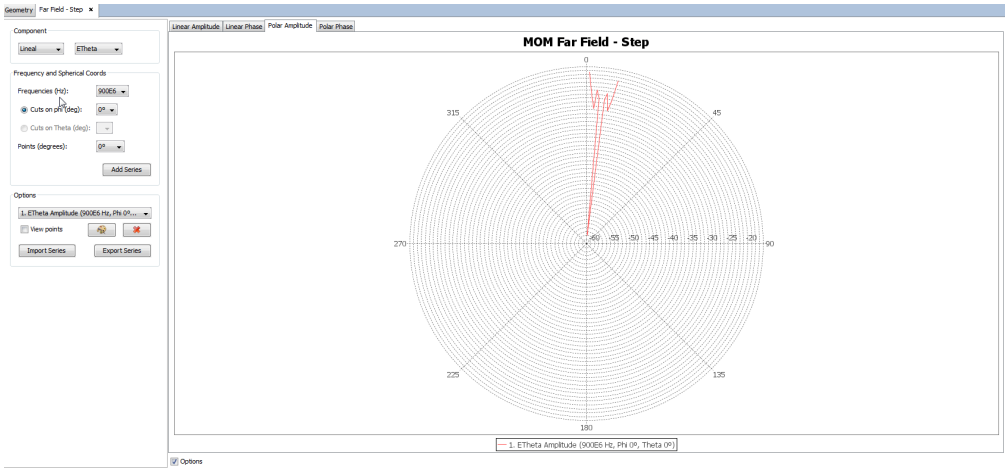


Figure 1129: View Cuts by Step panel. Polar Amplitude

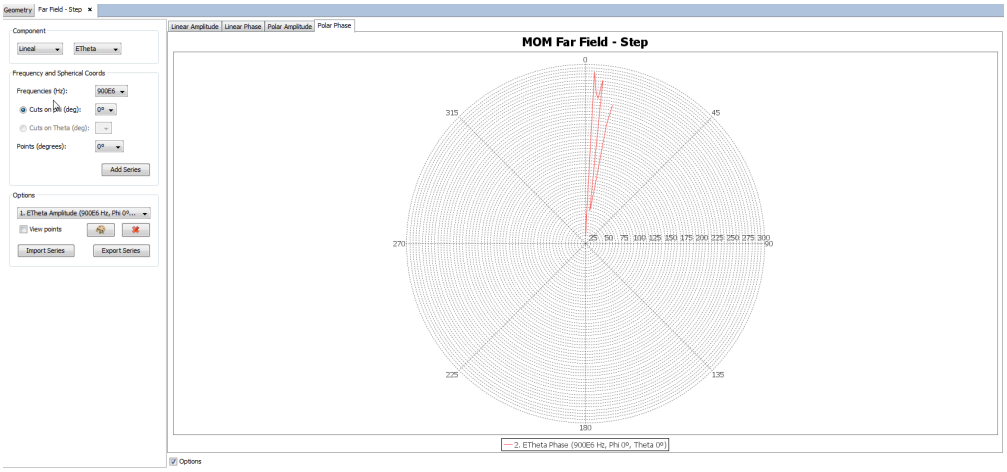


Figure 1130: View Cuts by Step panel. Polar Phase

View Text Files

The values of the field components at every far field directions may be visualized within this menu. When click on this option, a new tab-panel is opened on right side with some options to select the results to visualize. Click on Ok button to confirm and visualize the text files of the selected options.

- **Step** to select the results corresponding to a given status of the parametric simulation.
- **Current Frequencies** for the selected **Step**, a list of all the simulated frequencies is available.

When a simulation of an unique step and frequency is performed, the corresponding file text is directly opened to the left side of the **File Text** panel.

Click on **Save as** button to save the text file in the desired path.

11.12.2 Radiation Pattern

This menu is only enabled when the **Method of Moments** is selected on **Solver** section, and the option **Compute 3D Pattern** is selected in the Method of Moments. The angular step specified in the solver parameters is used for the whole spherical far field sweep, so this section allows to visualize results in these spherical coordinates.

The following results are available within this menu:

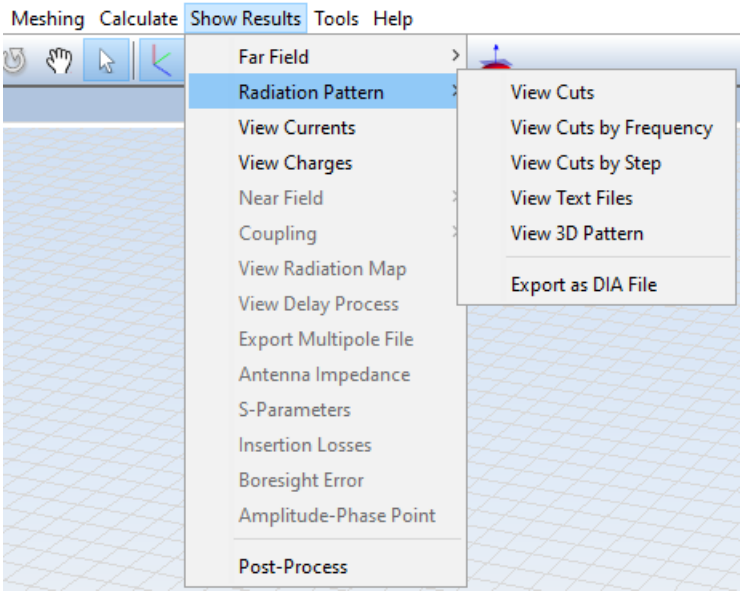


Figure 1131: Radiation Pattern results menu

View Cuts

When the user click on the **View Cuts** option, the following panel will appear:

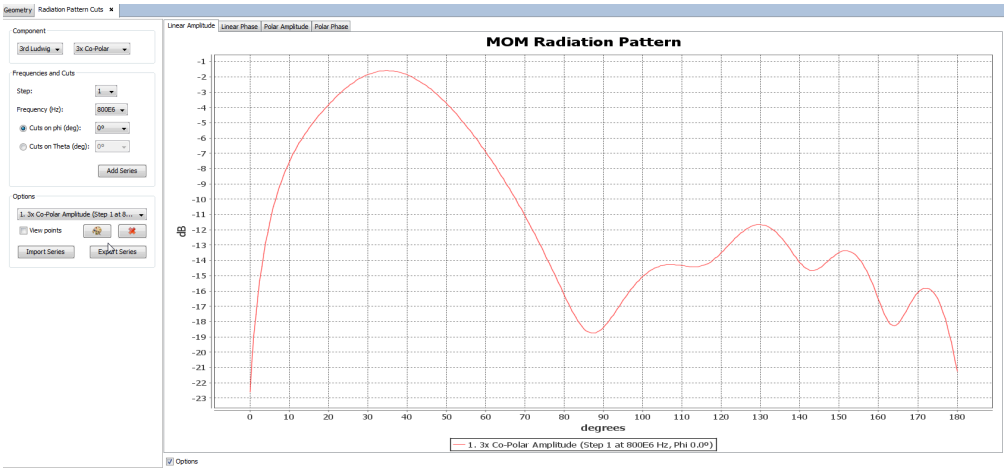


Figure 1132: Radiation Pattern View Cuts panel. Linear Amplitude

This window plots both the amplitude and the phase of the **FIELD** or **GAIN** computed in the radiation patter for every angular cut in the full spherical range. To plot a graph, select the desired parameters:

- **Component** The results may be plotted by using different polarizations and its components.
  - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.

  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.
- **Frecuencies and Cuts** the far field cuts may be represented for each frequency and direction.
-
  - **Step** To select the geometric step in a parametric simulation. If the geometry does not contain any parameter, only one Step is available.
  - **Frequencies** To select the desired frequency in a sweep. If just a frequency has been analyzed, only one **Frequency** is available.
  - **Cuts on...Phi/Theta** so many cuts as specified in the Observation Directions may be plot by using this section.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options**section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.
- **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series**:To export the selected curve in the above Plot List in a `.txt` file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

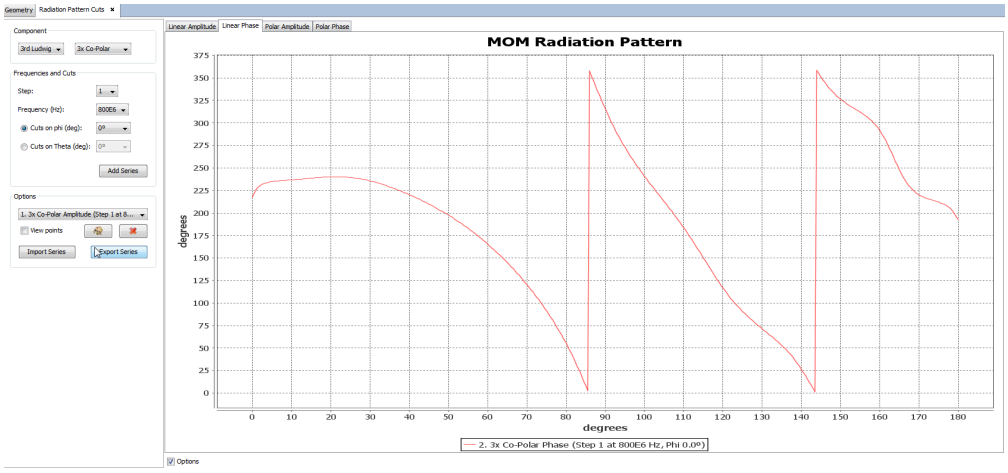


Figure 1133: Radiation Pattern View Cuts panel. Linear Phase

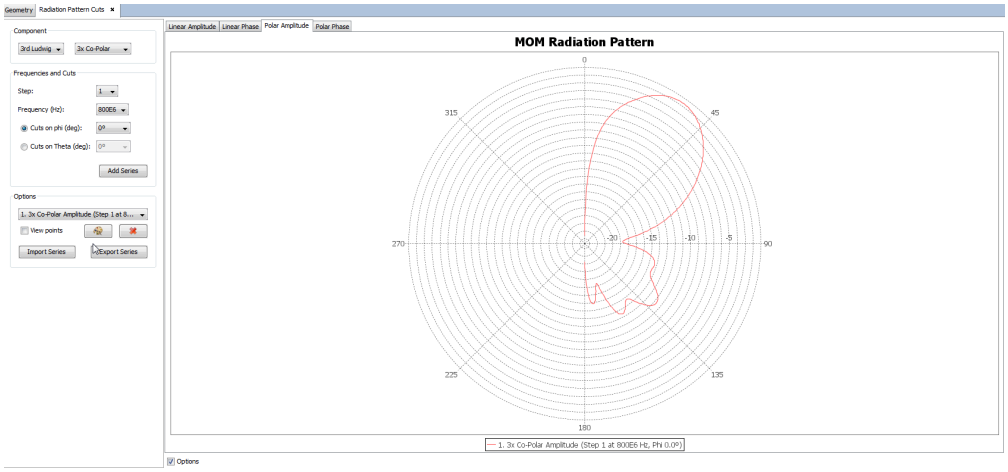


Figure 1134: Radiation Pattern View Cuts panel. Polar Amplitude

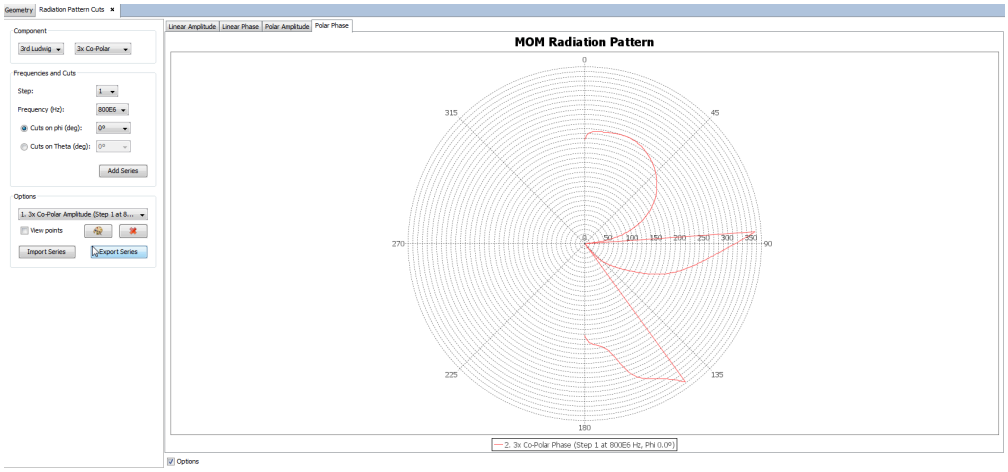


Figure 1135: Radiation Pattern View Cuts panel. Polar Phase

View Cuts by Frequency

This option is only enabled when a **Frequency Sweep** is chosen within the **Simulation Parameters**. It allows to compare the results in a given direction according to the frequency. When the user click on the **View Cuts By Frequency** option, the following panel will appear:

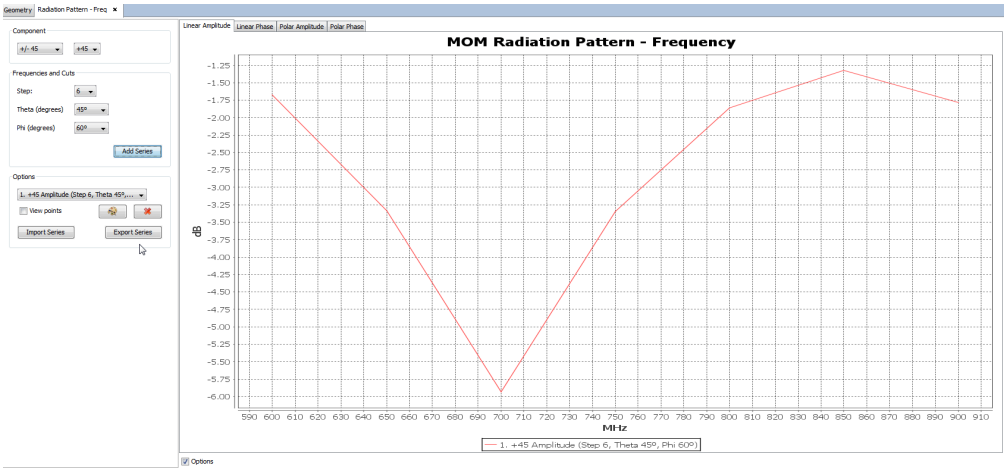


Figure 1136: Radiation Pattern View Cuts panel by frequency. Linear Amplitude

This window plots both the amplitude and the phase of the FIELD or GAIN computed for every simulated frequency. It is specially useful when a Frequency Sweep is enabled, as we can compare the field values in each observation direction/point at every frequency. To plot a graph, select the desired observation points:

- **Component** The results may be plotted by using different polarizations and its components.
- - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.
- **Step and Spherical Coords** the far field cuts are represented for each direction in the **3D Pattern**.
- - **Step** To select the geometric step in a parametric simulation. If the geometry does not contain any parameter, only one **Step** is available.
  - **Theta** select the theta direction in the **3D pattern** to compare its results according to the frequency.
  - **Phi** select the phi direction in the **3D pattern** to compare its results according to the frequency.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options**section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.

- **Import Series** To add a new plot from an external file. The file must have ". **txt**" extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series**:To export the selected curve in the above Plot List in a "**.txt**" file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

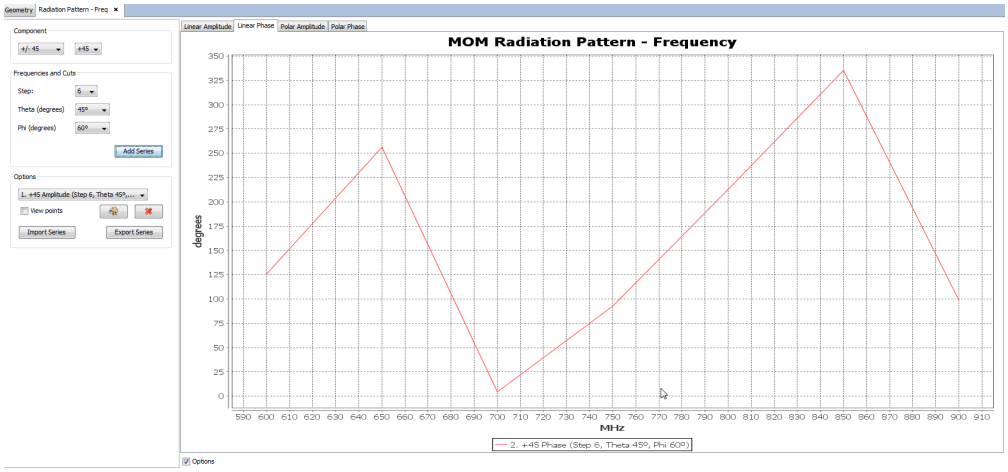


Figure 1137: Radiation Pattern View Cuts panel by frequency. Linear Phase

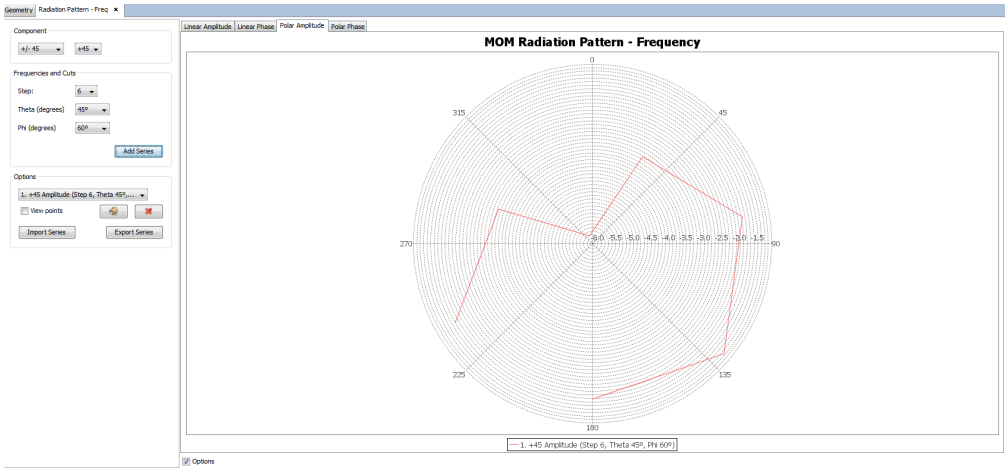


Figure 1138: Radiation Pattern View Cuts panel by frequency. PolarAmplitude

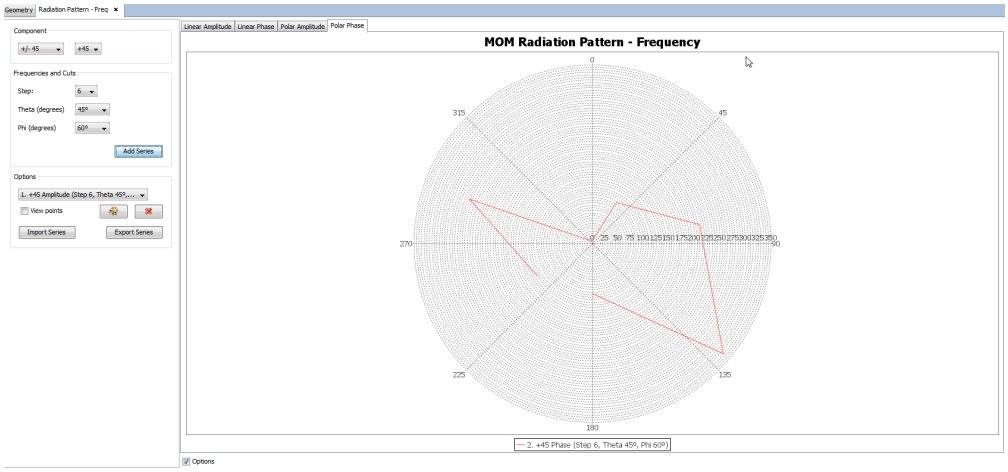


Figure 1139: Radiation Pattern View Cuts panel by frequency. Polar Phase

View Cuts by Step

This option is useful when a **Parametric Simulation** has been performed, as the results at each 3D pattern direction may be compared for every parametric step. We can compare the field values in each direction at every frequency.

When the user click on the **View Cuts By Step** option, the following panel will appear:

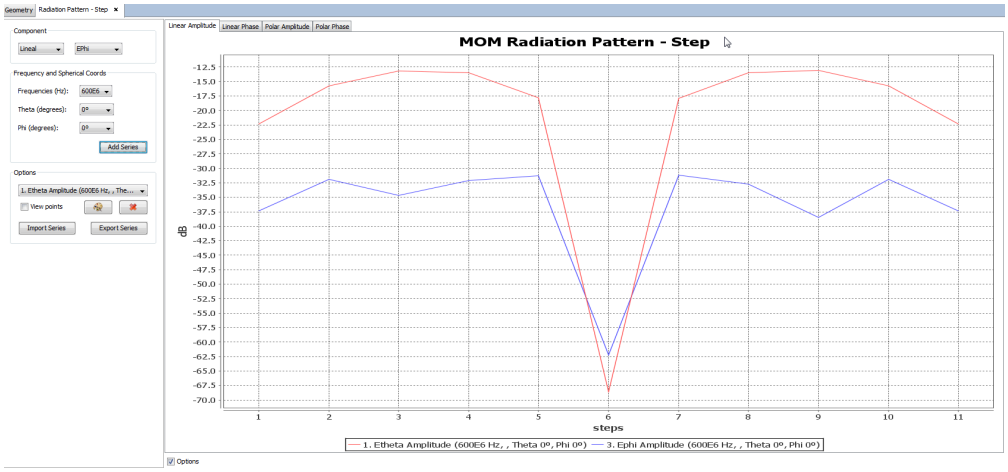


Figure 1140: View Cuts by Step panel. Linear Amplitude

Select the results to plot according to next options:

- **Component** The results may be plotted by using different polarizations and its components.
  - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.

- **Frequency and Spherical Coords** the 3D pattern cuts are represented for each frequency and direction.
  - **Frequencies** To select the desired frequency in a sweep. If just a frequency has been analyzed, only one Frequency is available.
  - **Theta** select the theta direction in the 3D pattern to compare its results according to the parametric steps.
  - **Phi** select the phi direction in the 3D pattern to compare its results according to the parametric steps.

Then, click on **Add Series** button to plot the selected curve. Having inserted several plots, the **Options**section allow to edit the result curves:

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
- **Remove** To delete the selected curve in the above **List of Series**.
- **Import Series** To add a new plot from an external file. The file must have ". **txt**" extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series**:To export the selected curve in the above Plot List in a "**.txt**" file, with the X and Y values of the plot in a new line.

The user can also switch between the linear graph and the polar graph to visualize far field results. It is possible to visualize the **Linear Phase**, **Polar Amplitude** and **Polar Phase** by clicking on their respective tabs.

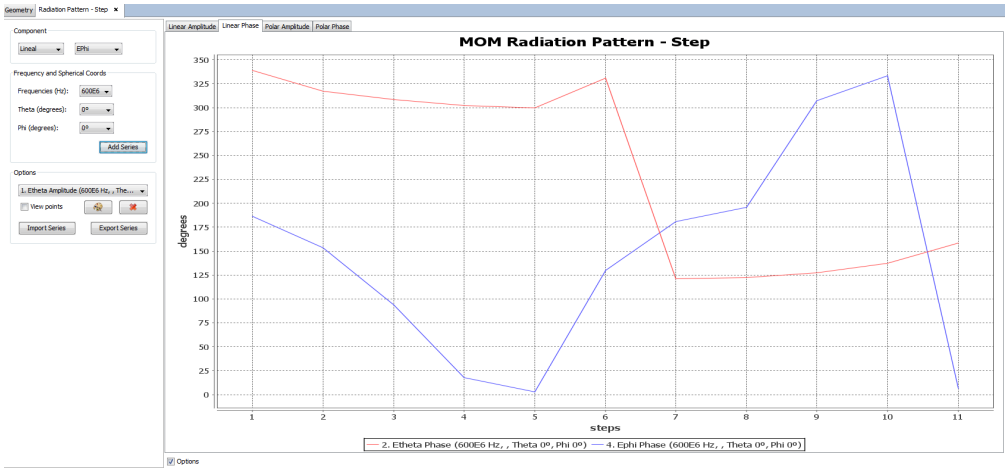


Figure 1141: View Cuts by Step panel. Linear Phase

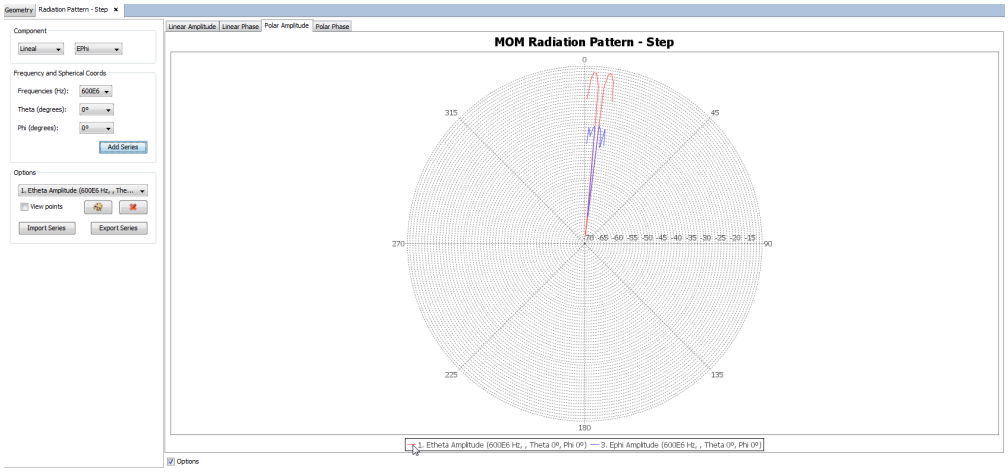


Figure 1142: View Cuts by Step panel. Polar Amplitude

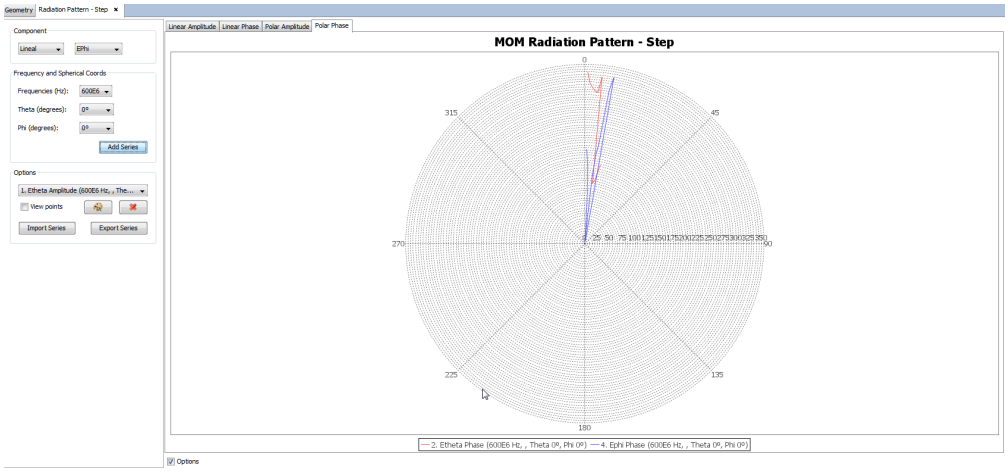


Figure 1143: View Cuts by Step panel. Polar Phase

View Text Files

The values of the field components of the full radiation pattern may be visualized within this menu. When click on this option, a new tab-panel is opened on right side with some options to select the results to visualize:

- **Step** to select the results corresponding to a given status of the parametric simulation.
- **Current Frequencies** for the selected **Step**, a list of all the simulated frequencies is available.

When a simulation of an unique step and frequency is performed, the corresponding file text is directly opened to the left side of the **Far Field File Text** panel.

Click on **Save as** button to save the text file in the desired path.

View 3D Pattern

This option allows to visualize the full 3D pattern, classified by parametric and frequency steps. The next figure shows and example of 3D pattern:

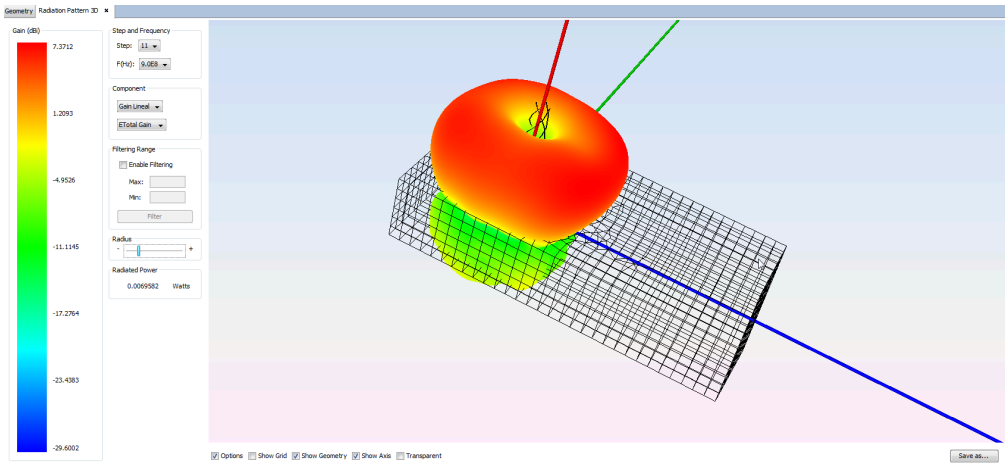


Figure 1144: 3D Radiation Pattern diagram

The following elements are available within this panel:

- **Left color legend** it shows the magnitude and range of values of the represented results.
- **Step and Frequency** to select the results from the available files to plot the diagram.
  - **Step** specify the parametric step when a parametric simulation has been run.
  - **Frequency** specify the desired frequency.
- **Component** The 3D diagram may be plotted by using different polarizations and its components.
  - **Lineal**: **ETheta, EPhi, ETotal, Ex, Ey** and **Ex/Ey Total** components.
  - **Circular RHCP, LHCP, Circular Total, and AxialRatio** components.
  - **+/- 45 +45** and **-45** components.
  - **3rd Ludwig 3x Co-Polar, 3x Cross-Polar, 3y Co-Polar**, and **3y Cross-Polar** components.
  - **Gain (dBi)**: **Theta Gain, Phi Gain, RHCP Gain, LHCP Gain** and **Total Gain** components.
- **Filtering Range** Use this section to skip results that are outside the desired range.
  - **Enable Filtering** The filter only is applied when this check box is selected.
  - **Max** To specify the maximum value of the range.
  - **Min** To specify the minimum value of the range.
  - **Filter** Click on this button to update the results filtering.
- **Radius** Use this slider to increase or decrease the size of the 3D pattern representation.
- **Radiated Power** Shows information about the effective radiated power (ERP), in watts.
- **Bottom Options** Below the 3D pattern diagram, the following options are included:
  - **Options** Disable this option to hide the left menus and visualize just the diagram.
  - **Show Grid** Enable this option to show a line grid in the 3D pattern.
  - **Show Geometry** Enable this option to visualize the geometry together the 3D pattern. Note that the 3D pattern is always represented in the origin of coordinates.

- **Transparent** Enable this option to add a transparency property to the visualized diagram.
- **Save as** To export the current representation as an image file.

Export as DIA File

This menu allows the user to export the radiation diagram into a DIA file to use it later in another project. After clicking on this option, a new panel is opened on right side with some options to export the desired DIA file:

- **Step** to select the results corresponding to a given status of the parametric simulation.
- **Frequency** for the selected **Step**, a list of all the simulated frequencies is available.

Click on **OK** button to confirm and export the selected file, and then select the desired **Path** to save the file.

11.12.3 View Currents

This option allows to visualize the induced currents on the geometry, classified by parametric and frequency steps. Click on **Show Results** - **View Currents** to open the window shown in next figure:

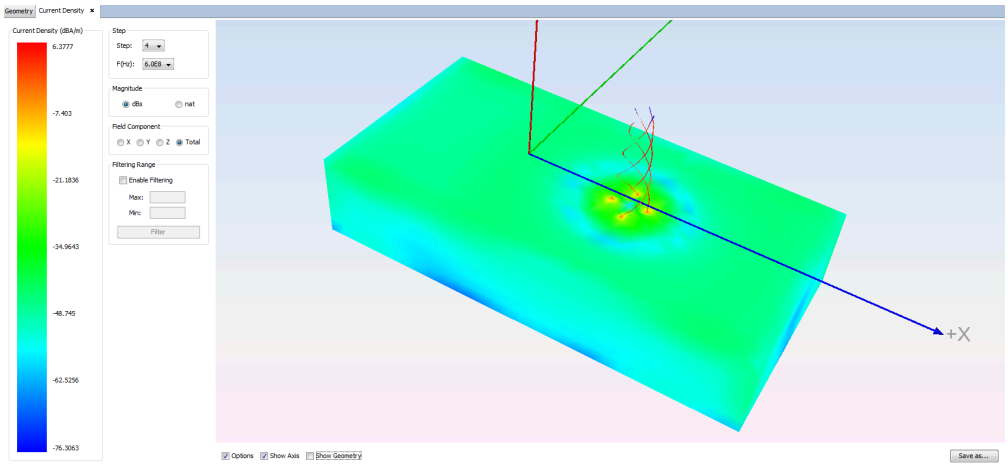


Figure 1145: Currents density diagram

The following elements are available within this panel:

- **Left color legend** it shows the magnitude and range of values of the represented **Current Density**.
- **Step and Frequency** to select the results from the available files to plot the currents.
  - **Step** specify the parametric step when a parametric simulation has been run.
  - **Frequency** specify the desired frequency.
- **Magnitude** The **Current Density** may be represented in natural or dBs magnitudes.
  - **dB** The current density is represented in logarithmic scale, given in dB-Amperes per meter.
  - **nat** The current density is represented in natural scale, given in Amperes per meter.

- **Field Component** To specify the component of the current density to plot. Only one component may be selected:
  - **X** Plot the X component of current density.
  - **Y** Plot the Y component of current density.
  - **Z** Plot the Z component of current density.
  - **Total** Plot the total current density, which is the combination of the X, Y and Z components.
- **Filtering Range** Use this section to skip results that are outside the desired range.
  - **Enable Filtering** The filter only is applied when this check box is selected.
  - **Max** To specify the maximum value of the range.
  - **Min** To specify the minimum value of the range.
  - **Filter** Click on this button to update the results filtering.
- **Bottom Options** Below the currents diagram, the following options are included:
  - **Options** Disable this option to hide the left menu and visualize just the currents distribution.
  - **Show Axis** Enable this option to show the coordinate axes.
  - **Show Geometry** Enable this option to visualize the meshed geometry together the current density.
  - **Antenna Results** Enable this option to visualize the antenna currents or disable it to hide them. It is only available when a radome has been analyzed.
  - **Save as** To export the current representation as an image file.

11.12.4 View Charges

This option allows to visualize the charge density on the geometry, classified by parametric and frequency steps. Click on **Show Results** - **View Charges** to open the window shown in next figure:

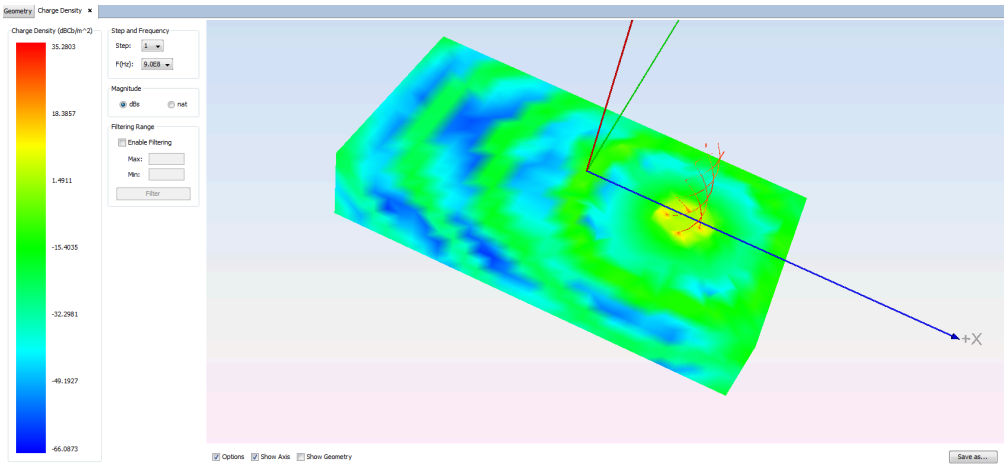


Figure 1146: The results represented in the figure above are corresponding to the MOM - Example 10.

The following elements are available within this panel:

- **Left color legend** it shows the magnitude and range of values of the represented **Charge Density**.
- **Step and Frequency** to select the results from the available files to plot the charges density.
  - **Step** specify the parametric step when a parametric simulation has been run.
  - **Frequency** specify the desired frequency.
- **Magnitude** The **Charge Density** may be represented in natural or dBs magnitudes.
  - **dB** The charges density is represented in logarithmic scale, given in dB-Coulomb per square meter.
  - **nat** The charges density is represented in natural scale, given in Coulomb per square meter.
- **Field Component** To specify the component of the current density to plot. Only one component may be selected:
  - **X** Plot the X component of current density.
  - **Y** Plot the Y component of current density.
  - **Z** Plot the Z component of current density.
  - **Total** Plot the total current density, which is the combination of the X, Y and Z components.
- **Filtering Range** Use this section to skip results that are outside the desired range.
  - **Enable Filtering** The filter only is applied when this check box is selected.
  - **Max** To specify the maximum value of the range.
  - **Min** To specify the minimum value of the range.
  - **Filter** Click on this button to update the results filtering.
- **Bottom Options** Below the charges diagram, the following options are included:
  - **Options** Disable this option to hide the left menu and visualize just the charges distribution.
  - **Show Axis** Enable this option to show the coordinate axes.
  - **Show Geometry** Enable this option to visualize the meshed geometry together the charges density.
  - **Antenna Results** Enable this option to visualize the antenna charges or disable it to hide them. It is only available when a radome has been analyzed.
  - **Save as** To export the charges representation as an image file.

11.12.5 Near Field

This menu is only enabled with any observation point has been added in **Output** - **Observation points** section. With in this menu, the following options are available to visualize different results on near field.

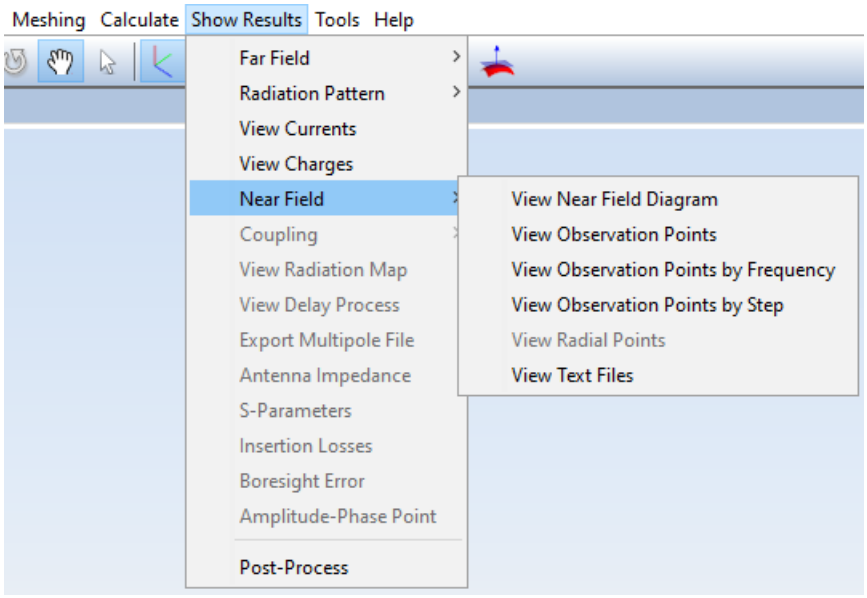


Figure 1147: Near Field menu

View Near Field Diagram

After clicking on **Show Results - Near Field - View Near Field Diagram**, a new panel is opened on right side to select which observation points are considered in the diagram. In this panel, the following options are available:

- **Step** To specify the step results files when a parametric simulation has been run.
- **Frequency** To specify the desired frequency results files when a frequency sweep has been run.
- **Observation points** List with all the observation points entities defined. Select the desired ones to plot the near field results.
- **Select all** Click on this button to select all the observation points entities.
- **Ok** Confirm the observation points to plot and open the **Near Field 3D** diagram.

Then, the options available in the **Near Field 3D** results panel are listed:

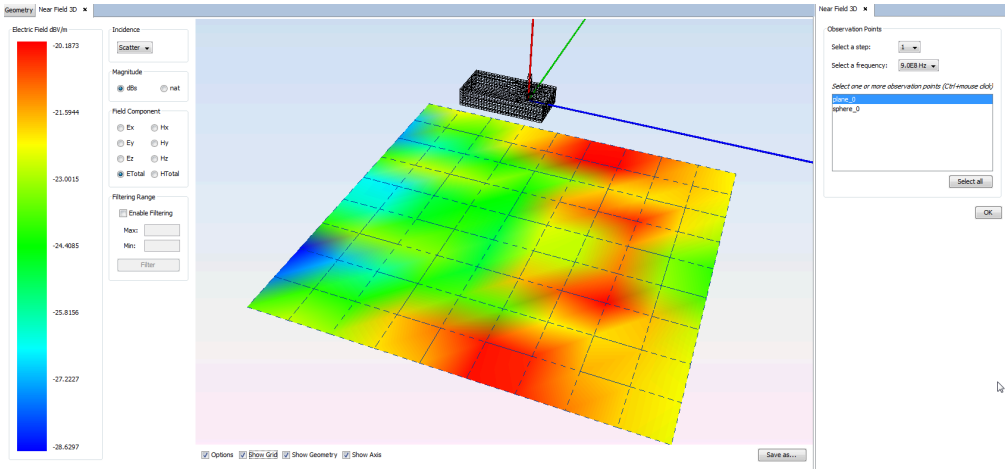


Figure 1148: Near Field diagram

- **Left color legend** it shows the magnitude and range of values of the represented results.
- **Incidence** The near field may be plotted by considering different contributions on the observation points.
  - **Scatter** Scattered field components.
  - **Direct** Direct field components.
  - **Total** Combination of the scattered field plus the direct field components.
- **Magnitude** The near field may be represented in natural or dBs magnitudes.
  - **dBs** The near field is represented in logarithmic scale, given in dB-Amperes per meter for magnetic field, or in dB-Volts per meter for electric field.
  - **nat** The near field is represented in natural scale, given in Amperes per meter for magnetic field, or in Volts per meter for electric field.
- **Field Component** To specify the field component to plot. Only one component may be selected:
  - **Ex** Plot the X component of electric field.
  - **Ey** Plot the Y component of electric field.
  - **Ez** Plot the Z component of electric field.
  - **ETotal** Plot the total electric field, which is the combination of the X, Y and Z components.
  - **Hx** Plot the X component of magnetic field.
  - **Hy** Plot the Y component of magnetic field.
  - **Hz** Plot the Z component of magnetic field.
  - **HTotal** Plot the total magnetic field, which is the combination of the X, Y and Z components.
- **Filtering Range** Use this section to skip results that are outside the desired range.
-
  - **Enable Filtering** The filter only is applied when this check box is selected.
  - **Max** To specify the maximum value of the range.
  - **Min** To specify the minimum value of the range.
  - **Filter** Click on this button to update the results filtering.
- **Bottom Options** Below the near field diagram, the following options are included:

- ◦ **Options** Disable this option to hide the left menu and visualize just the near field diagram.
  - ◦ **Show Grid** Enable this option to visualize the grid of the near field observation entity.
  - ◦ **Show Geometry** Enable this option to visualize the meshed geometry together the near field diagram.
  - ◦ **Show Axis** Enable this option to show the coordinate axes.
  - ◦ **Save as** To export the current representation as an image file.

View Observation Points

Use this option to plot curves of near field components along the observation points. Note that the observation points are listed in the X axis, so its order may vary the curve shape. Click on **Show Results - Near Field - View Observation Points**, and the below window will appear:

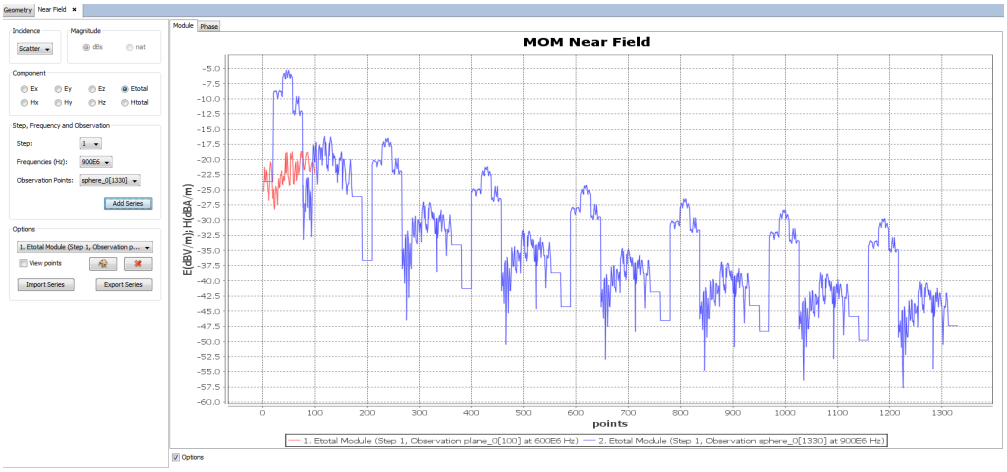


Figure 1149: View Observation Points panel

Within this window, the following options are available:

- **Incidence** The near field may be plotted by considering different contributions on the observation points.
  - ◦ **Scatter** Scattered field components.
  - ◦ **Direct** Direct field components.
  - ◦ **Total** Combination of the scattered field plus the direct field components.
- **Magnitude** The near field may be represented in natural or dBs magnitudes.
  - ◦ **dBs** The near field is represented in logarithmic scale, given in dB-Amperes per meter for magnetic field, or in dB-Volts per meter for electric field.
  - ◦ **nat** The near field is represented in natural scale, given in Amperes per meter for magnetic field, or in Volts per meter for electric field.
- **Component** To specify the field component to plot. Only one component may be selected:
  - ◦ **Ex** Plot the X component of electric field.
  - ◦ **Ey** Plot the Y component of electric field.
  - ◦ **Ez** Plot the Z component of electric field.

  - **ETotal** Plot the total electric field, which is the combination of the X, Y and Z components.
  - **Hx** Plot the X component of magnetic field.
  - **Hy** Plot the Y component of magnetic field.
  - **Hz** Plot the Z component of magnetic field.
  - **HTotal** Plot the total magnetic field, which is the combination of the X, Y and Z components.
- **Step, Frequency and Observation** Select the observation points to plot.
  - **Step** To specify the step results files when a parametric simulation has been run.
  - **Frequency** To specify the desired frequency results files when a frequency sweep has been run.
  - **Observation points** List with all the observation points entities defined. Select the desired ones to plot the near field results.
  - **Add series** Confirm the observation points to plot and add its curve.
- **Options** Having inserted several plots, the **Options** section allow to edit the result curves:
  - **List of Series** All the plotted field components are listed. Note that two options are included for every plot for most of components both the amplitude and the phase. However, **ETotal** and **HTotal** only represent the module.
  - **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
  - **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
  - **Remove** To delete the selected curve in the above **List of Series**.
  - **Import Series** To add a new plot from an external file. The file must have ". **txt**" extension, and it must contain in every new line the point index and the values to be plotted.
  - **Export Series**:To export the selected curve in the above Plot List in a "**.txt**" file, with the point index and the value of the plot in a new line.
- **Options** Disable this option to hide the left menu and visualize just the near field curves.

View Observation Points By Frequency

This option allow to compare the near field values in a given observation point according to the frequency. It is only useful if a **Frequency Sweep** has been run. Click on **Show Results - Near Field - View Observation Points By Frequency**, and the next window will appear:

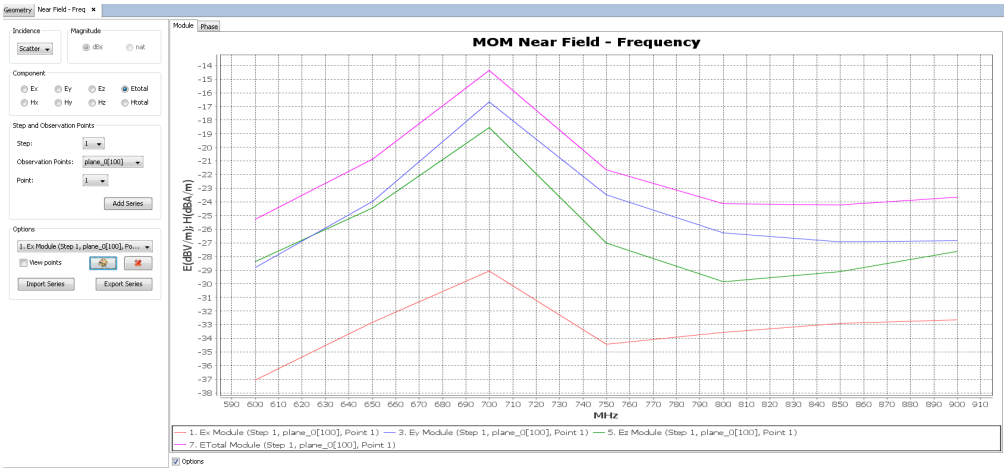


Figure 1150: View Observation Points By Frequency panel (Module)

Within this window, the following options are available:

- **Incidence** The near field may be plotted by considering different contributions on the observation points.
  - **Scatter** Scattered field components.
  - **Direct** Direct field components.
  - **Total** Combination of the scattered field plus the direct field components.
- **Magnitude** The near field may be represented in natural or dBs magnitudes.
  - **dBs** The near field is represented in logarithmic scale, given in dB-Amperes per meter for magnetic field, or in dB-Volts per meter for electric field.
  - **nat** The near field is represented in natural scale, given in Amperes per meter for magnetic field, or in Volts per meter for electric field.
- **Component** To specify the field component to plot. Only one component may be selected:
  - **Ex** Plot the X component of electric field.
  - **Ey** Plot the Y component of electric field.
  - **Ez** Plot the Z component of electric field.
  - **ETotal** Plot the total electric field, which is the combination of the X, Y and Z components.
  - **Hx** Plot the X component of magnetic field.
  - **Hy** Plot the Y component of magnetic field.
  - **Hz** Plot the Z component of magnetic field.
  - **HTotal** Plot the total magnetic field, which is the combination of the X, Y and Z components.
- **Step and Observation Points** Select the observation points to plot.
  - **Step** To specify the step results files when a parametric simulation has been run.
  - **Observation points** List with all the observation points entities defined. Select the desired ones to load the next list.
  - **Point** Specify the index point within the observation entities selected above to compare its values along the frequency.

  - **Add series**:Confirm the observation points to plot and add its curve.
- **Options** Having inserted several plots, the **Options** section allow to edit the result curves:
  - **List of Series** All the plotted field components are listed. Note that two options are included for every plot for most of components both the amplitude and the phase. However, **ETotal** and **HTotal** only represent the module.
  - **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
  - **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
  - **Remove** To delete the selected curve in the above **List of Series**.
  - **Import Series** To add a new plot from an external file. The file must have ". **txt**" extension, and it must contain in every new line the point index and the values to be plotted.
  - **Export Series**:To export the selected curve in the above Plot List in a "**.txt**" file, with the point index and the value of the plot in a new line.
- **Options** Disable this option to hide the left menu and visualize just the near field curves.

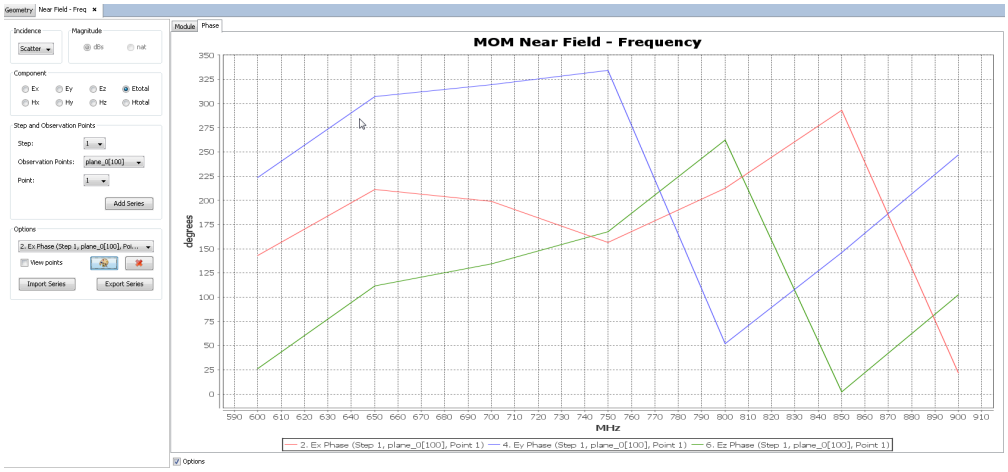


Figure 1151: View Observation Points By Frequency panel (Phase)

View Observation Points By Step

This option allow to compare the near field values in a given observation point according to the geometric configuration (or parametric step). It is only useful if the geometry depends on one or more parameters which have multiple values. Click on **Show Results - Near Field - View Observation Points By Step**, and the next window will appear:

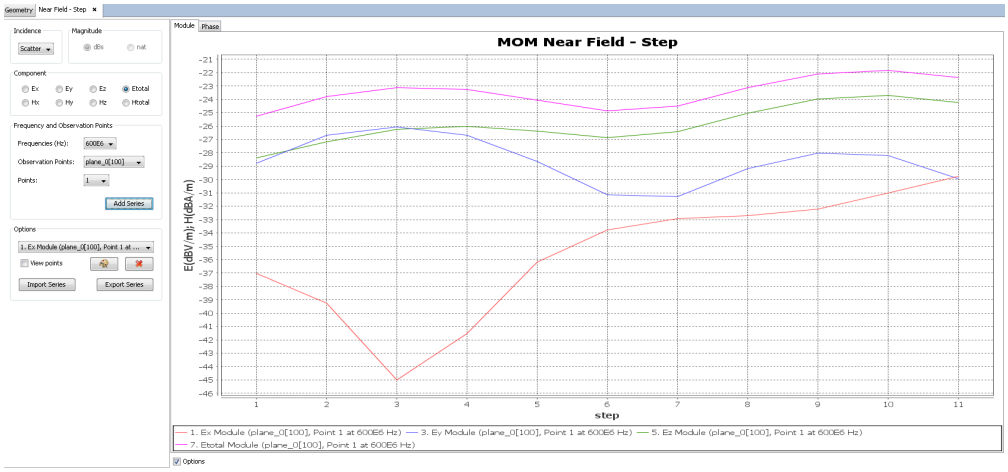


Figure 1152: View Observation Points By Step panel (Module)

Within this window, the following options are available:

- **Incidence** The near field may be plotted by considering different contributions on the observation points.
  - **Scatter** Scattered field components.
  - **Direct** Direct field components.
  - **Total** Combination of the scattered field plus the direct field components.
- **Magnitude** The near field may be represented in natural or dBs magnitudes.
  - **dBs** The near field is represented in logarithmic scale, given in dB-Amperes per meter for magnetic field, or in dB-Volts per meter for electric field.
  - **nat** The near field is represented in natural scale, given in Amperes per meter for magnetic field, or in Volts per meter for electric field.
- **Component** To specify the field component to plot. Only one component may be selected:
  - **Ex** Plot the X component of electric field.
  - **Ey** Plot the Y component of electric field.
  - **Ez** Plot the Z component of electric field.
  - **ETotal** Plot the total electric field, which is the combination of the X, Y and Z components.
  - **Hx** Plot the X component of magnetic field.
  - **Hy** Plot the Y component of magnetic field.
  - **Hz** Plot the Z component of magnetic field.
  - **HTotal** Plot the total magnetic field, which is the combination of the X, Y and Z components.
- **Frequency and Observation Points** Select the observation points to plot.
  - **Frequencies** To specify the desired frequency results files when a frequency sweep has been run.
  - **Observation points** List with all the observation points entities defined. Select the desired ones to load the next list.
  - **Point** Specify the index point within the observation entities selected above to compare its values along the frequency.

  - **Add series**:Confirm the observation points to plot and add its curve.
- **Options** Having inserted several plots, the **Options** section allow to edit the result curves:
  - **List of Series** All the plotted field components are listed. Note that two options are included for every plot for most of components both the amplitude and the phase. However, **ETotal** and **HTotal** only represent the module.
  - **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
  - **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
  - **Remove** To delete the selected curve in the above **List of Series**.
  - **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the point index and the values to be plotted.
  - **Export Series**:To export the selected curve in the above Plot List in a `.txt` file, with the point index and the value of the plot in a new line.
- **Options** Disable this option to hide the left menu and visualize just the near field curves.

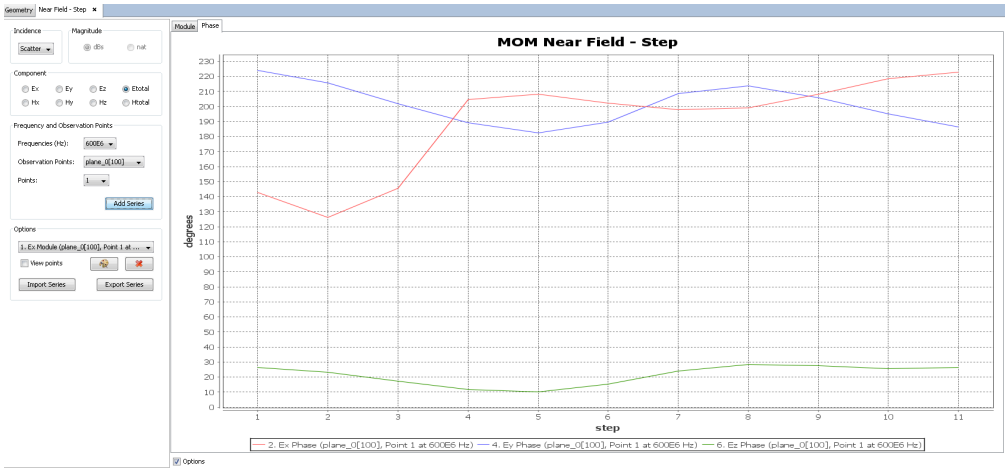


Figure 1153: View Observation Points By Step panel (Phase)

View Radial Points

This option is only enabled when the user has set up several Sphere within the **Output - Observation Points** section. It plots the near field for each observation spheres in the desired direction versus the radius. After clicking on **Show Results - Near Field - View Radial Points**, the next window is open.

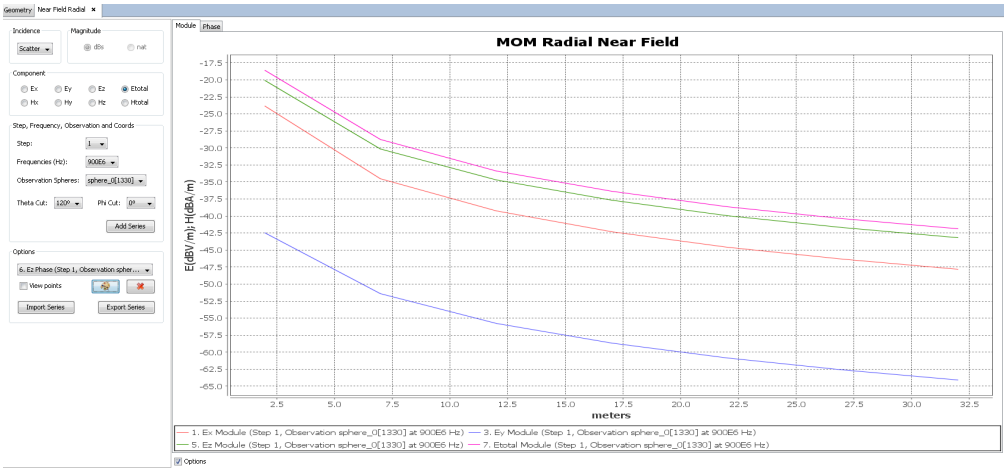


Figure 1154: Radial Field results (Module)

Within this window, the following options are available:

- **Incidence** The near field may be plotted by considering different contributions on the observation points.
  - **Scatter** Scattered field components.
  - **Direct** Direct field components.
  - **Total** Combination of the scattered field plus the direct field components.
- **Magnitude** The near field may be represented in natural or dBs magnitudes.
  - **dBs** The near field is represented in logarithmic scale, given in dB-Amperes per meter for the magnetic field, or in dB-Volts per meter for the electric field.
  - **nat** The near field is represented in natural scale, given in Amperes per meter for the magnetic field, or in Volts per meter for the electric field.
- **Component** To specify the field component to the plot. Only one component may be selected:
  - **Ex** Plot the X component of the electric field.
  - **Ey** Plot the Y component of the electric field.
  - **Ez** Plot the Z component of the electric field.
  - **ETotal** Plot the total electric field, which is the combination of the X, Y and Z components.
  - **Hx** Plot the X component of magnetic field.
  - **Hy** Plot the Y component of magnetic field.
  - **Hz** Plot the Z component of magnetic field.
  - **HTotal** Plot the total magnetic field, which is the combination of the X, Y and Z components.
- **Step, Frequency, Observation and Coords** Select the observation points to plot.
  - **Step** To specify the step results files when a parametric simulation has been run.
  - **Frequency** To specify the desired frequency results files when a frequency sweep has been run.
  - **Observation Spheres** List with all the different observation spheres entities that have been defined. Select the desired one to specify the direction in the next fields.

  - **Theta Cut** Spherical theta angle within the above-selected observation sphere, in degrees.
  - **Phi Cut** Spherical theta angle within the above-selected observation sphere, in degrees.
  - **Add series** Confirm the observation points to plot and add its curve.
- **Options** Having inserted several plots, the **Options** section allow to edit the result curves:
  - **List of Series** All the plotted field components are listed. Note that two options are included for every plot for most of the components both the amplitude and the phase. However, **ETotal** and **HTotal** only represent the module.
  - **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
  - **Change color** This option allows to change the color of the selected curve in the above **List of Series**.
  - **Remove** To delete the selected curve in the above **List of Series**.
  - **Import Series** To add a new plot from an external file. The file must have ". **txt**" extension, and it must contain in every new line the point index and the values to be plotted.
  - **Export Series** To export the selected curve in the above Plot List in a "**.txt**" file, with the point index and the value of the plot in a new line.
- **Options** Disable this option to hide the left menu and visualize just the near field curves.

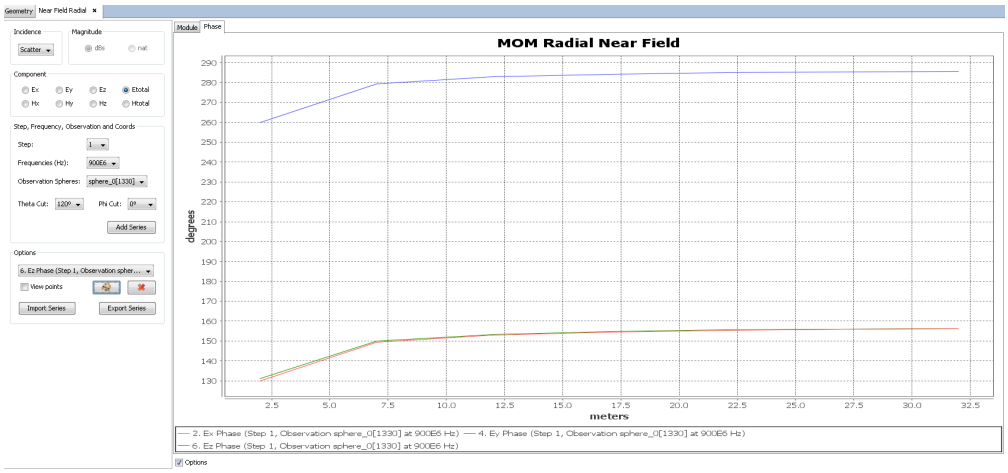


Figure 1155: Radial Field results (Phase)

View Text Files

The values of the near field components on the observation points may be visualized and exported as a text file within this menu. When click on this option, a new tab-panel is opened on right side with some options to select the results to visualize:

- **Step** to select the results corresponding to a given status of the parametric simulation.
- **Frequency** for the selected **Step**, a list of all the simulated frequencies is available.

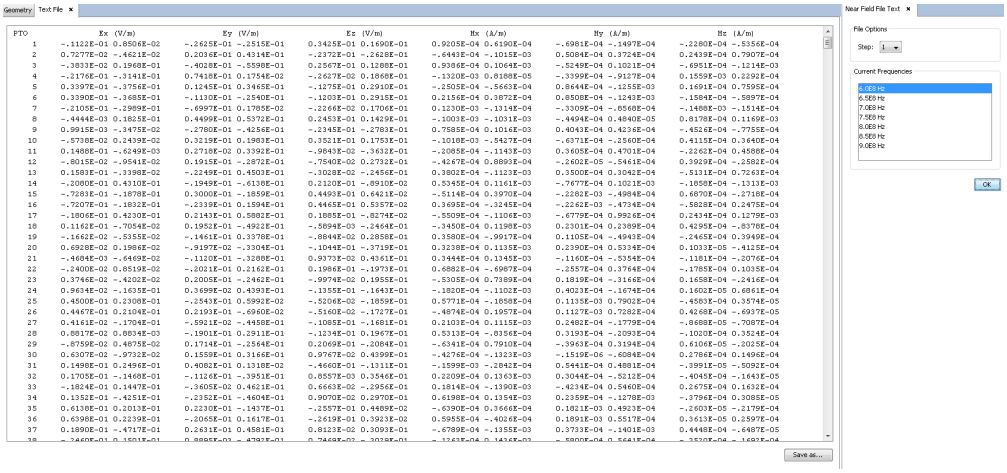


Figure 1156: Viewing near field values in text format

When a **Text File** is visualized, click on **Export** button to save it as a text file in the desired path.

11.12.6 View Radiation Map

The **Radiation Map** is a diagram that shows show the scattered field by each region in the specified directions. It is only available when the **Radiation Map** has been set as **Simulation type** in the **Simulation Parameters** section.

The signal, which may be set with different shapes, is generated by every **antenna source** available in the simulation, and the delay process may be computed in multiple far field observation directions.

After running a **Radiation Map** type simulation, click on **Show Results - View Radiation Map** to open the **Radiation Map** results, and the following panel is shown:

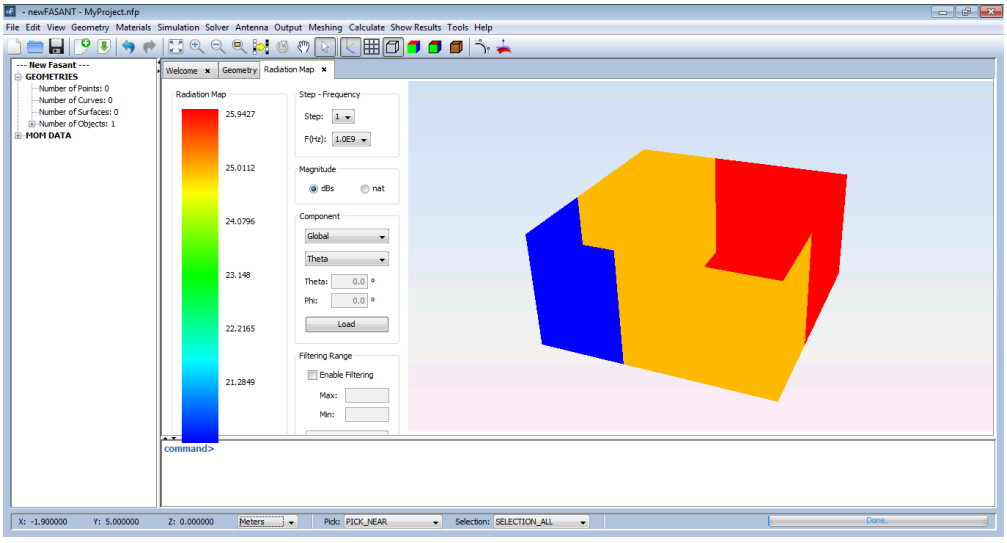


Figure 1157: Radiation Map results

Within this window, the following options are available:

Step - Frequency

Select the results files to plot.

- **Step** To specify the step results files when a parametric simulation has been run.
- **Frequency** To specify the desired frequency results files when a frequency sweep has been run.

Magnitude

The radiation map may be represented in natural or dBs magnitudes.

- **dBs** The radiation map is represented in logarithmic scale, given in dB-Volts per meter for electric field.
- **nat** The radiation map is represented in natural scale, given in Volts per meter for electric field.

Component

To specify the field component to plot and the observation direction to be considered:

- **Global/Directional** Select the observation direction.
  - **Global** Only the Z axis direction is considered.
  - **Directional** The user may introduce the observation direction, defined by its spherical coordinates.
- **Phi/Phi/Total** Select the field component of electric field to plot.
- **Theta** Spherical theta angle to define the observation direction, in degrees. It is only enabled when de **Directional** mode is selected.
- **Phi** Spherical phi angle to define the observation direction, in degrees. It is only enabled when de **Directional** mode is selected.
- **Load** Confirm the selection to plot and update the diagram representation.

Filtering Range

Use this section to skip results that are outside the desired range.

- **Enable Filtering** The filter only is applied when this check box is selected.
- **Max** To specify the maximum value of the range.
- **Min** To specify the minimum value of the range.
- **Filter** Click on this button to update the results filtering.

Bottom Options

Below the radiation map diagram, the following options are included:

- **Options** Disable this option to hide the left menu and visualize just the radiation map diagram.
- **Show Axis** Enable this option to show the coordinate axes.
- **Show Geometry** Enable this option to visualize the meshed geometry together the radiation map results.
- **Save as** To export the represented results as an image file.

11.12.7 View Delay Process

The **Delay Process** is a post-process computation that shows how a given signal is distorted in the time domain because of its environment. It is only available when the **Delay Process** has been set as **Simulation type** in the **Simulation Parameters** section.

The signal, which may be set with different shapes, is generated by every **antenna source** available in the simulation, and the delay process may be computed in multiple far field observation directions.

After running a **Delay Process** type simulation, click on **View Delay Process** to open the **Delay Process** window, and the following panel is shown on the right side:

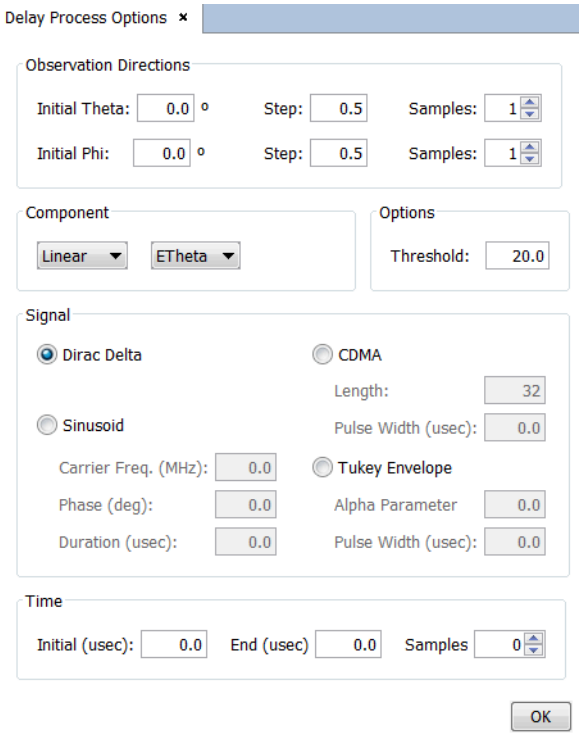


Figure 1158: Delay Process panel

This panel contains several options for plotting the Delay Process:

- **Observation Directions** to define the spherical far-field ranges where the **Delay Process** will be computed. The first row defines the theta interval and the second row the phi one.
  - **Initial Theta** first observation direction in theta, in degrees.
  - **Step (Theta)** angular increment for the theta sampling, in degrees.
  - **Samples (Theta)** total angles analyzed in theta direction, starting from the **Initial Theta** with **Step** equispaced intervals.
  - **Initial Phi** first observation direction in phi, in degrees.
  - **Step (Phi)** angular increment for the phi sampling, in degrees.
  - **Samples (Phi)** total angles analyzed in phi direction, starting from the **Initial Phi** with **Step** equispaced intervals.

- **Component** specify the field polarization and components considered for the **Delay Process** computation.
  - **Linear** or **Circular** polarization are differenciated.
  - **Etheta** or **Ephi** components may be chosen only when the **Linear** polarization is selected. It specifies the consideration of only the **Theta** or **Phi** component.
  - **Eul** or **Eur** components may be chosen only when the **Circular** polarization is selected. It specifies the consideration of only the **Left** or **Right** component.
- **Options - Threshold** only the contributions whouse level is below the strongest one up to this threshold, in dB, are taken into account.
- **Signal** the original signal may be generated by using different patterns. All of them are configured in the time domain.
  - **Dirac Delta** only a pulse is generated in the initial time.
  - **CDMA** a random distribution of pulses centred with positive (high) an negative (low) levels is generated. Note that a low or high level is considered as a **Pulse**.
    - **Length** number of pulses to be generated.
    - **Pulse Width** time duration of each **Pulse**, in microseconds.
  - **Sinusoid** a sinusoidal signal is generated.
    - **Carrier Freq.** frequency of the sinusoidal signal, in MHz.
    - **Phase** initial phase of the sinusoidal signal, in degrees.
    - **Duration** total length of the generated sinusoid in the time domain, in microseconds.
  - **Tukey Envelope** a Gaussian signal enveloped by a rectangular pulse is generated.
    - **Alpha Parameter** define the slope of the Gaussian curve. The higher is this parameter, the more enveloped by the rectangular pulse is the generated signal.
    - **Pulse Width** define the total length of the generated signal in the time domain, in microseconds.
- **Time** this section specifies the observation time range.
- *Initial*
  first observation instant in the time domain, in microseconds.

  End
  last observation instant in the time domain, in microseconds.

  Samples
  number of instants to evaluate de**Delay Process**that are equally equispaced in the time domain.

Click on **OK** button to plot de **Delay Process** results.

Only when the **CDMA** signal has been selected, the window represented below ask the selection of the **Correlation Signal** to be plotted with the following options.

- **Input (x[n])** the original CDMA signal is represented.
- **Output (y[n])** the received CDMA signal is represented
- **Correlation (c[n])** the correlation on the received signal is represented.

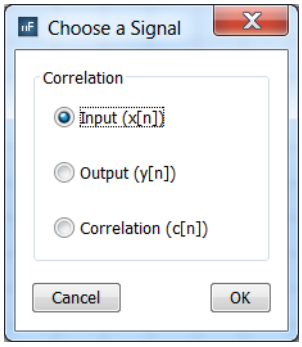


Figure 1159: Selection of signal to be plotted with CDMA

In the following, a **Coaxial Feed Dipole** has been simulated at 300 MHz. to compare the plots obtained by using the different signals:

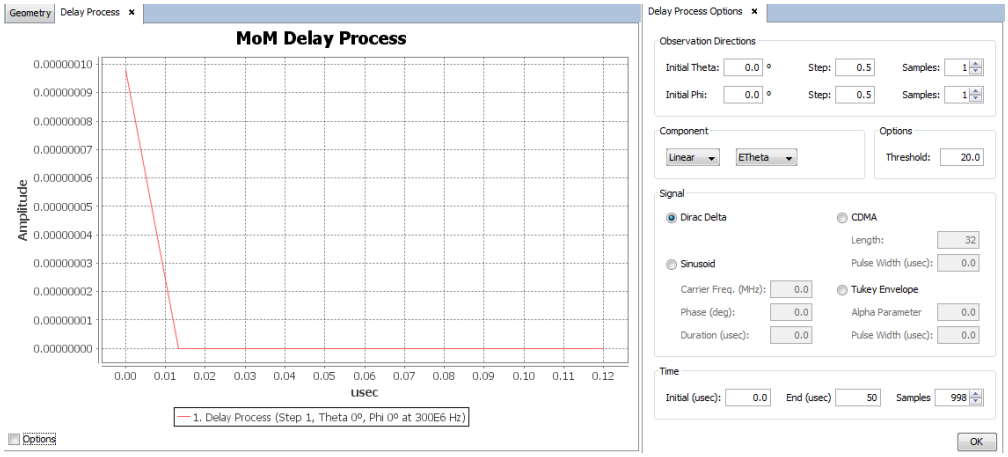


Figure 1160: Delay Process generated with a Dirac Delta signal

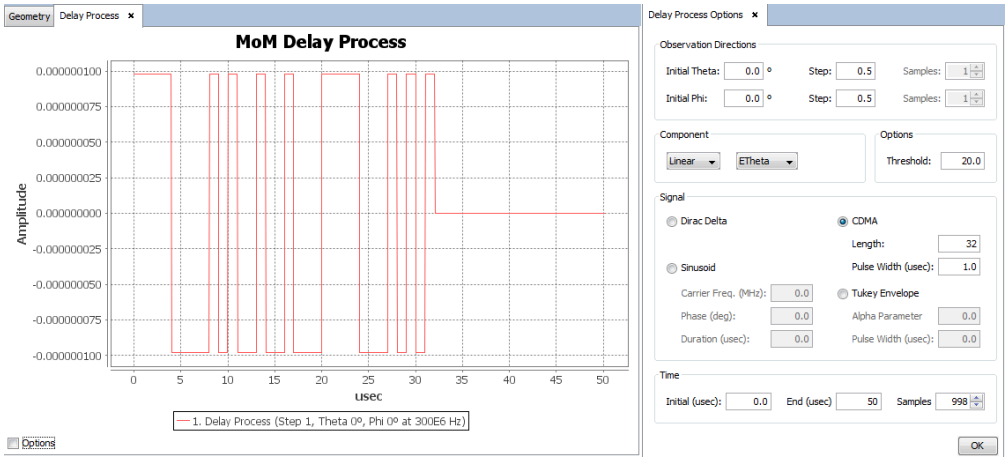


Figure 1161: Delay Process generated with a CDMA (output y[n]) signal

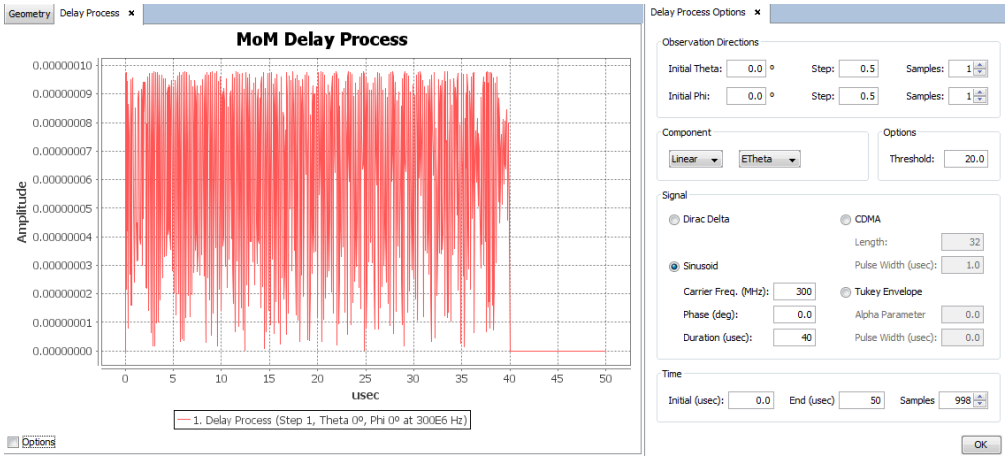


Figure 1162: Delay Process generated with a Sinusoid signal

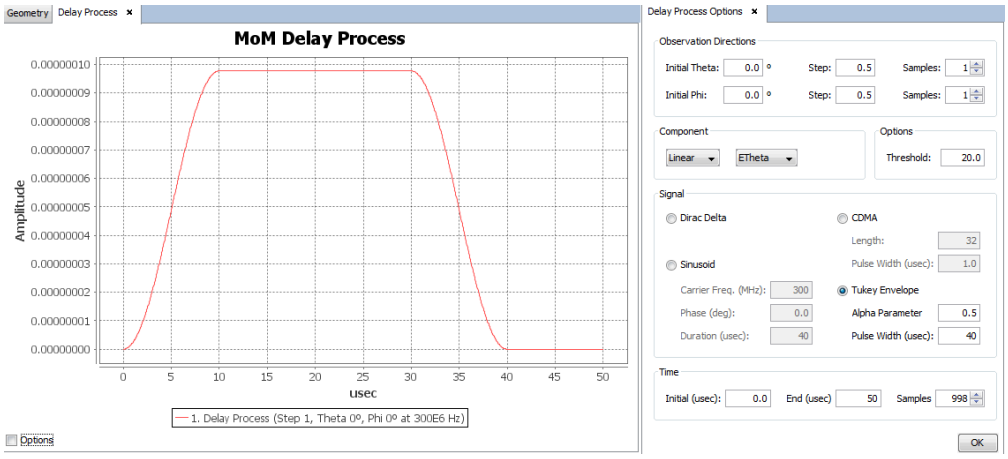
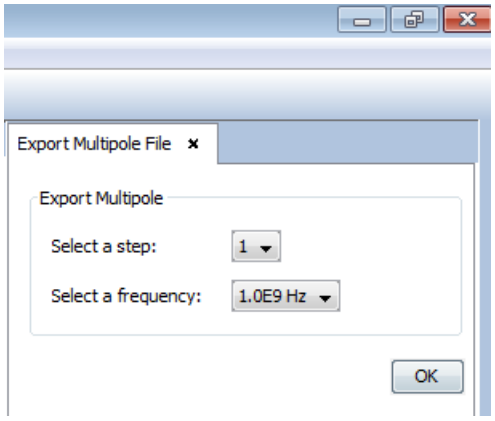


Figure 1163: Delay Process generated with a Tukey Envelope signal

11.12.8 Export Multipole File

This option is only enabled when the **Simulation Type** is set to **Multipole**. Then, use it to save the multipole simulation file of a project to be used in later simulations. When the geometry depends on parameters, the multipoles of every steps and frequencies are available:

- **Select a step** Select the parametric step to export its multipole results.
- **Select a frequency** Specify the frequency of the sweep to export its multipole results.



11.12.9 Antenna Impedance

With this option, the user can visualize the calculated impedance results for each antenna and for each user defined frequency.

- Select the parametric **Step** to consider its results.
- The user can specify the impedance used for normalizing the results with the **Normalized (SWR - Return Loss)** parameter, or using the local impedance at every frequency by using the **Auto** option. The table is updated whenever one of this parameters is modified.

Only the antennas fed with physical feeding (for example, a **Coaxial Feed** point) are included in this panel.

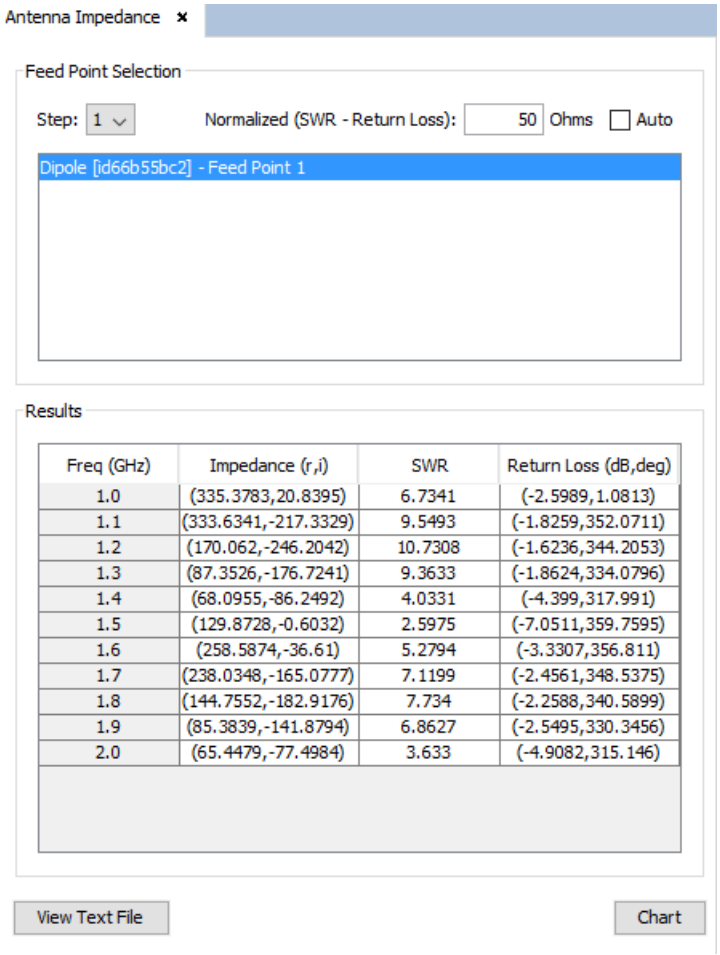


Figure 1164: Viewing Antenna Impedance results in table format

Clicking the **View Text File** button, the results shown as a table will appear in text file format:

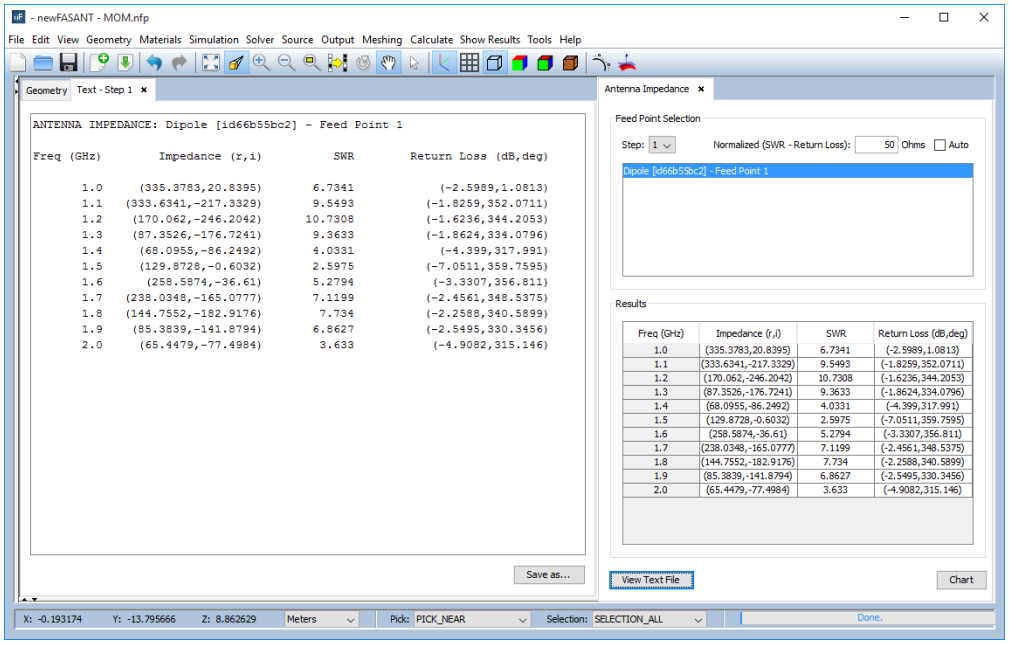


Figure 1165: Viewing Antenna Impedance results in text format

If user selects one column of the table and press **Chart** button, a representation of the selected values will appear in chart format.

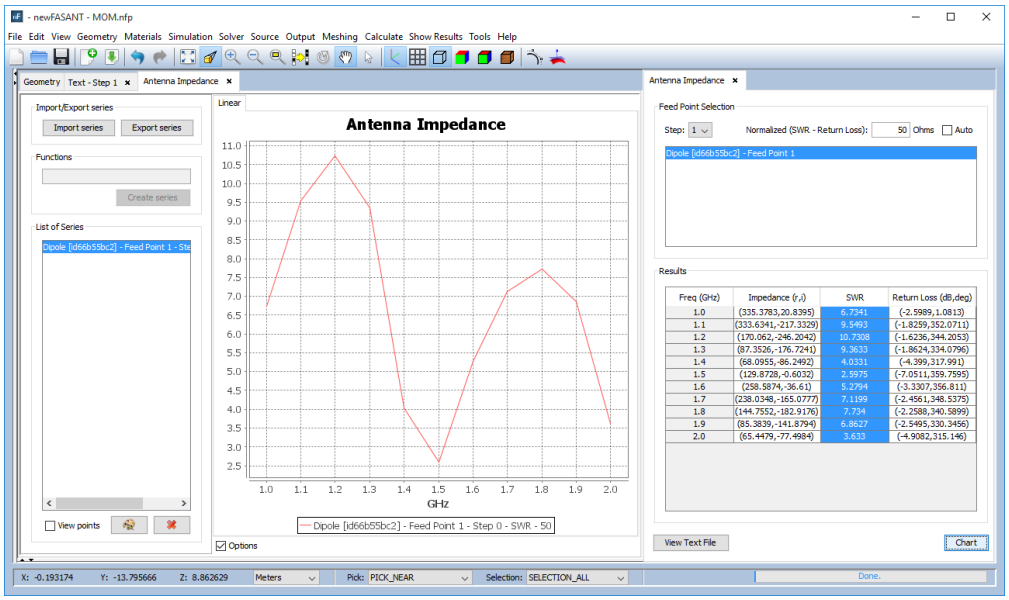


Figure 1166: Viewing Antenna Impedance results in chart format

11.12.10 S-Parameters

This option is only enabled when the **Simulation Type** is set to **S-Parameters**. Then, the user can visualize the calculated S-Parameters results for each defined coaxial feed antennas and for each user defined frequency.

Click on **Show Results - S-Parameters** menu and the **S-Parameters** panel shown in figure below is open.

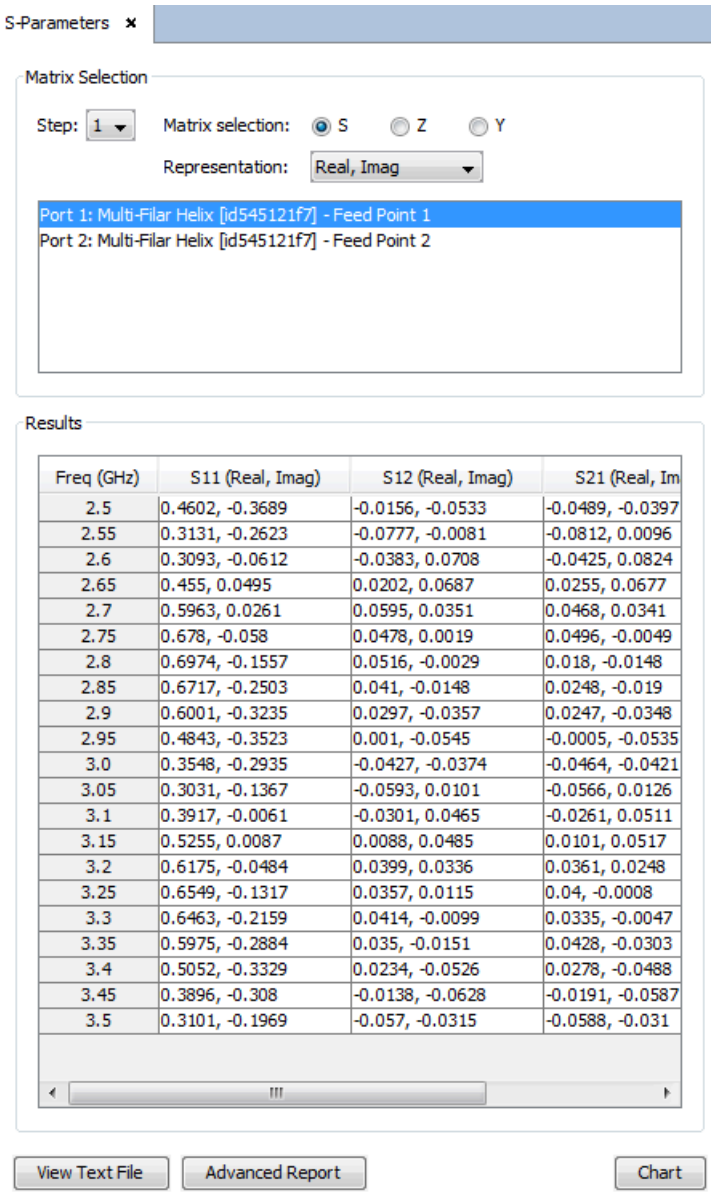


Figure 1167: S-Parameters panel

The **S-Parameters** panel contains the following options:

- **Matrix Selection**:
  - **Step** To select the geometric step in a parametric simulation. If the geometry does not contain any parameter, only one **Step** is available.

    - **Matrix Selection** The results may be represented in different magnitudes.
        - **S** scattering parameters.
        - **Z** impedance parameters.
        - **Y** admittance parameters.
    - **Representation** The results are represented as complex values, given by different components and units.
        - **Real, Imag** The **Real** and **Imaginary** parts are shown, in natural scale.
        - **Amplitude, Phase** The **Amplitude** is given in natural format, and the **Phase** in degrees.
        - **db, Phase** The amplitude, which is named **dB**, is given in logarithmic format, and the **Phase** in degrees.
    - **Ports section** It shows all the available ports. When a port is selected in this list, it is selected in the geometry panel, so it is useful to identify where each port is placed.
- **Results** This section contains a table with the results represented in the format specified according to the **Matrix Selection** configuration. Each row in the table represents a frequency with all the parameters given by columns. Within the table, the results may be selected by columns for further plotting.
- **Buttons below**:
    - **View Text File** Open a new tab panel with the table **Results** given in text file format. It can be exported by clicking on **Save as...** button.
    - **Advanced Report** Open a new window with multiple options to export all the results as a **html** file. The next information may be edited:
        - **Output Matrices** The results may be generated in one or all the magnitudes as mentioned above.
        - **Output Units** The results may be generated in one or all the magnitudes and units as mentioned above.
        - **Frequency** To select all frequency steps within a frequency sweep with the **Export All Frequencies** option, or just the desired one with the **Export Single Frequency** option.
        - **Step** To select all geometric steps in the parameteric simulation with the **Export All Steps** option, or just the desired one with the **Export Single Step** option.
        - **Output Format** To select the output format.
        - **Generate** Click on this button to confirm the selected configuration and generate the output file. Then, the **path** of the output file is required.
        - **Cancel** To close the **Advanced Report** window.

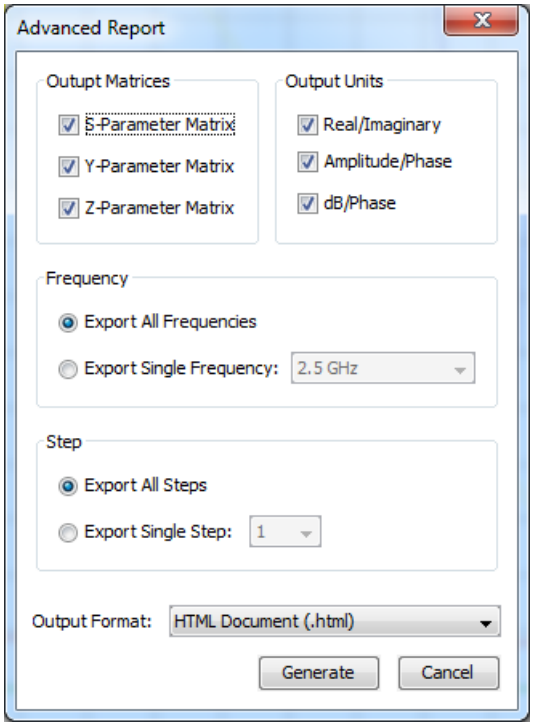


Figure 1168: Advanced Report window

- **Chart** Click on this button to generate a 2D-plot with the selected row within the **Results** section. Every time that this button is clicked on, the new plots are included in the same graph.

Some examples of results representation are included in next figures.

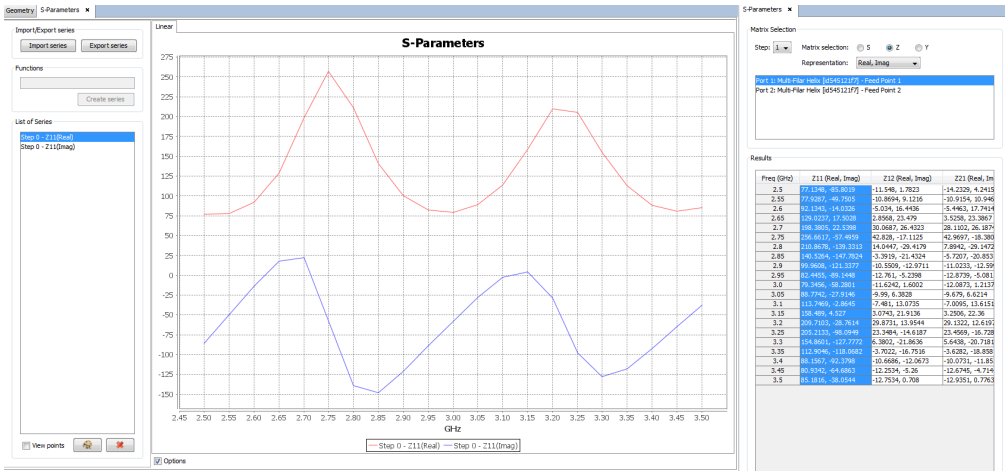


Figure 1169: S-Parameters plot

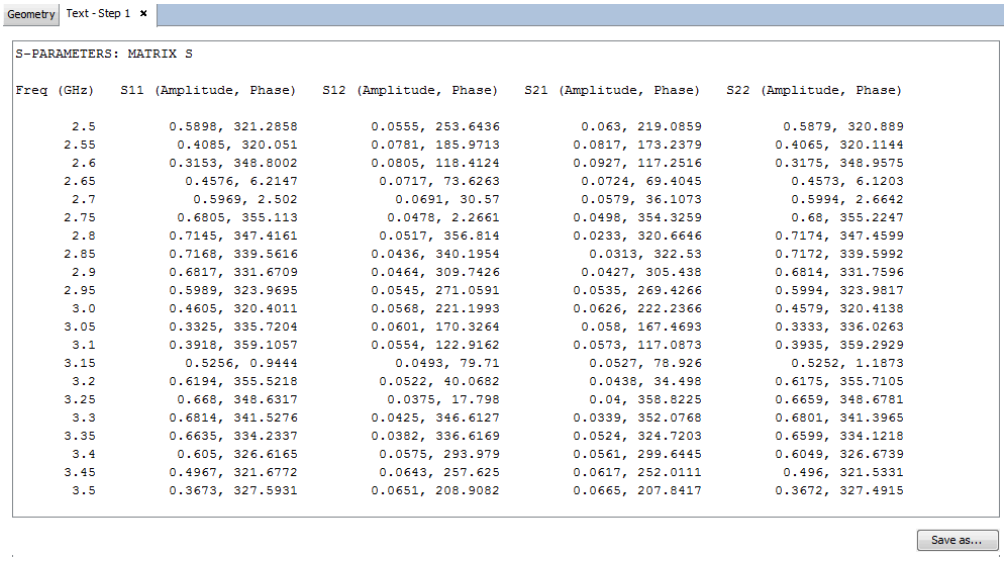


Figure 1170: S-Parameters as text file

Advanced Scattering Parameters Report

Step 1

S-Parameters

Frequency: 2.5E9

	Real/Imag	Amplitude/Phase	dB/Phase
S11	R = 0.4602 I = -0.3689	A = 0.5898 P = 321.2858	dB = -4.5854 P = 321.2858
S12	R = -0.0156 I = -0.0533	A = 0.0555 P = 253.6436	dB = -25.107 P = 253.6436
S21	R = -0.0489 I = -0.0397	A = 0.063 P = 219.0859	dB = -24.0123 P = 219.0859
S22	R = 0.4562 I = -0.3709	A = 0.5879 P = 320.889	dB = -4.6141 P = 320.889

Frequency: 2.55E9

	Real/Imag	Amplitude/Phase	dB/Phase
S11	R = 0.3131 I = -0.2623	A = 0.4085 P = 320.051	dB = -7.7772 P = 320.051
S12	R = -0.0777 I = -0.0081	A = 0.0781 P = 185.9713	dB = -22.1418 P = 185.9713
S21	R = -0.0812 I = 0.0096	A = 0.0817 P = 173.2379	dB = -21.7534 P = 173.2379
S22	R = 0.3119 I = -0.2606	A = 0.4065 P = 320.1144	dB = -7.8199 P = 320.1144

Frequency: 2.6E9

	Real/Imag	Amplitude/Phase	dB/Phase
S11	R = 0.3093 I = -0.0612	A = 0.3153 P = 348.8002	dB = -10.0247 P = 348.8002
S12	R = -0.0383 I = 0.0708	A = 0.0805 P = 118.4124	dB = -21.8805 P = 118.4124
S21	R = -0.0425 I = 0.0824	A = 0.0927 P = 117.2516	dB = -20.6571 P = 117.2516
S22	R = 0.3116 I = -0.0608	A = 0.3175 P = 348.9575	dB = -9.9647 P = 348.9575

Figure 1171: S-Parameters as html file

11.12.11 Insertion Losses

This option allows the user to visualize the values of the **Insertion Losses** only on simulations with some **radome** defined and the **Calculate antenna results** option activated on the radome definition window. User can visualize the **Insertion Losses** values for all frequencies of the selected simulation step.

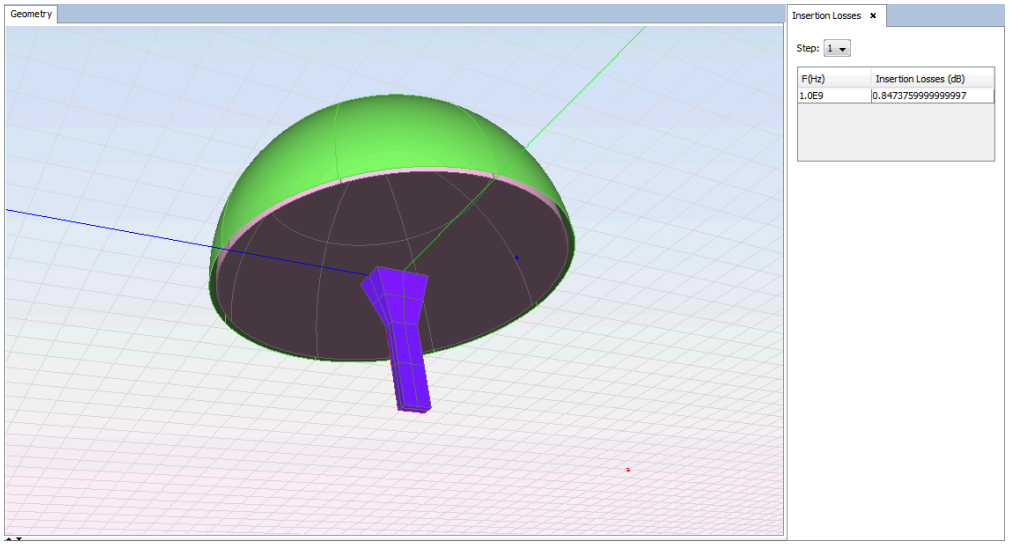


Figure 1172: Insertion Losses Results

11.12.12 Boresight Error

This option allows the user to visualize the values of the **Boresight Error**only on simulations with some **radome** defined and the **Calculate antenna results** option activated on the radome definition window. User can see the **Boresight Error** values for all frequencies of the selected simulation step.

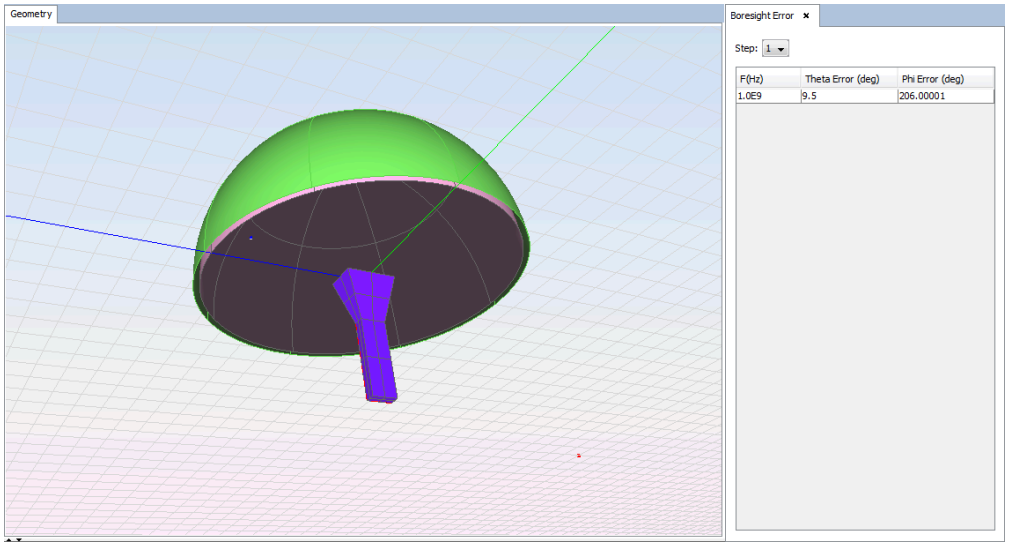


Figure 1173: Boresight Error Results

11.12.13 Post-Process

For information about the Post-Process menu, see Post-Process.

11.12.14 Amplitude-Phase Point Results

The user can choose between seeing the results divided by the different frequencies used, or by the number of steps committed.

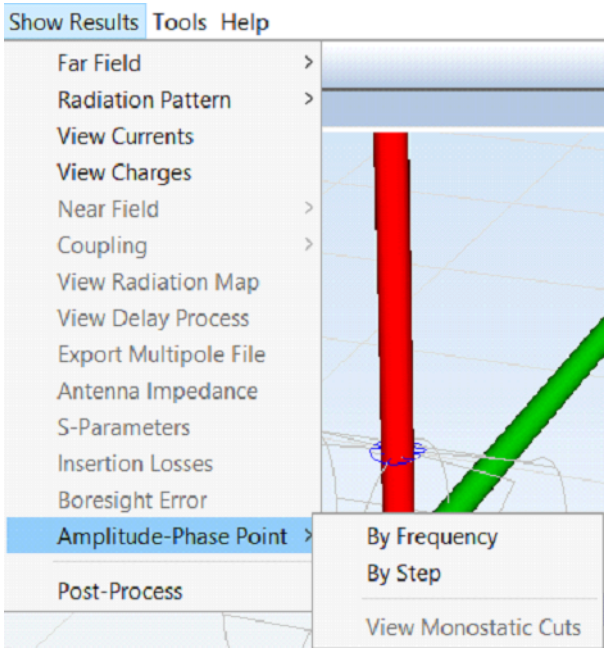


Figure 1174: Amplitude-Phase Point results menu

Amplitude-Phase Point **by frequency**

When the user selects the **by frequency** option the following tab will appear:

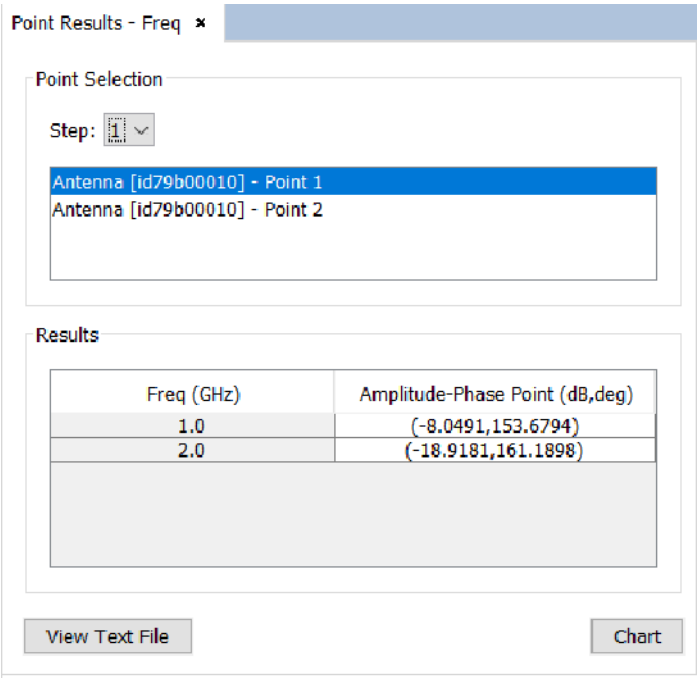


Figure 1175: Point results by frequency

In this menu the user can select which point will be represented in the chart. In order to do so, the user must select first the point that will be represented and click on the first cell of the Amplitude-Phase Point column of the Results chart. It is also posible to specify a step(Default value is 1). User can visualize in plain text the evolution of a point in a specified step with the View Text File button.

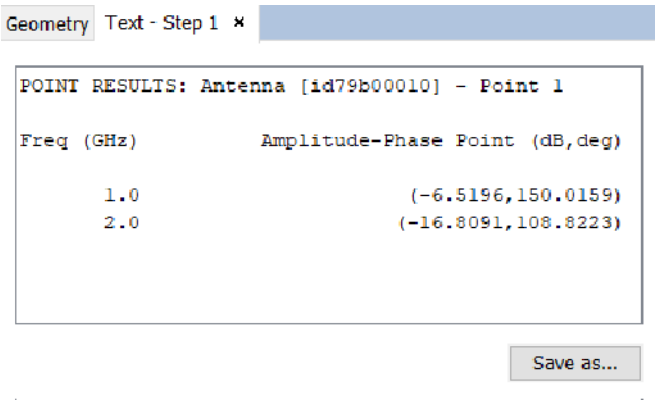


Figure 1176: Point results by frequency in plain text

Then click the Chart button and the following graph will appear:

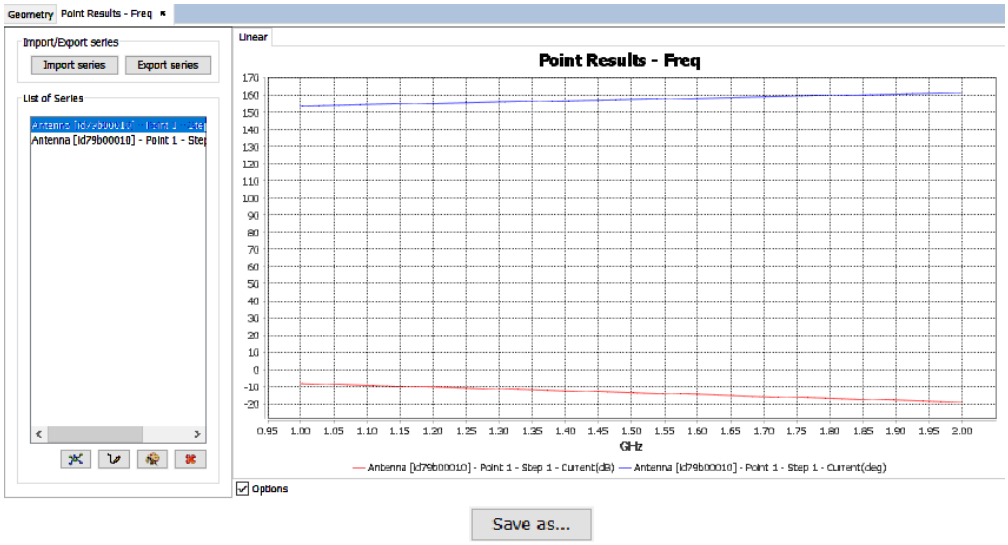


Figure 1177: Amplitude-Phase Point results by Frequency.

Import/Export Series: With these options the user is allowed to export or import a series. To export a series the user needs to click in the panel beneath which series will be exported (Amplitude or Phase). This will give the user a `.txt` file with the X and Y values of the plot in a new line. To import a series the user must submit a `.txt` file which must contain in every new line the X and Y values to be plotted.

List of Series In this panel will appear all the series that are being plotted into the graph. Here the user can customize each plot or remove the desired ones. The customization options are:

- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.
- **Change series stroke** Change the composition of the plots from a preset of options.
- **Change Color** Change the color of the selected plot.
- **Remove Plot** Delete a chosen plot from the graph.

Amplitude-Phase Point **by step**

When the user selects the **by step** option the following tab will appear:

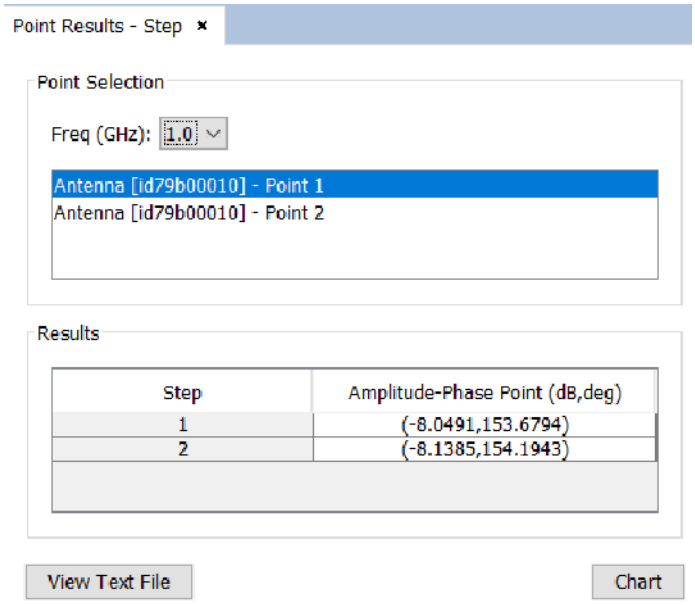


Figure 1178: Point results by step

On this menu the user can select which point will be represented in the chart. In order to do so, the user must select first the point that will be represented and click on the first cell of the Amplitude-Phase Point column of the Results chart. It is also possible to specify a frequency band(Default value is 1.0). User can visualize in plain text the evolution of a point with a specified frequency with the View Text File button.

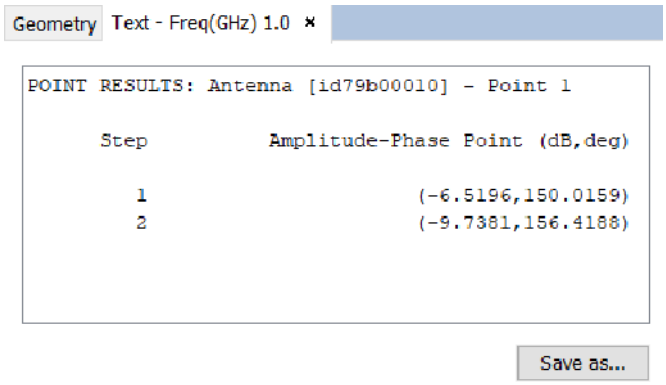


Figure 1179: Point results by step in plain text

Then click the Chart button and the following graph will appear:

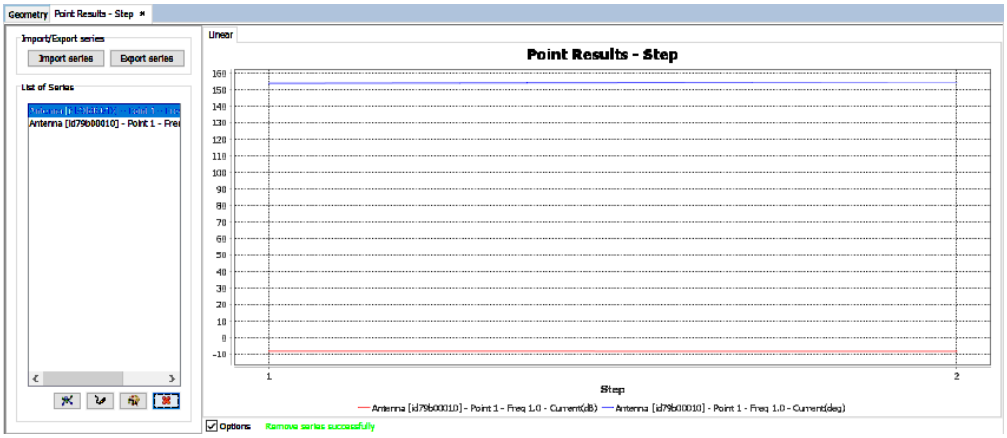


Figure 1180: Amplitude-Phase Point results by Steps.

Import/Export Series: With these options the user is allowed to export or import a series. To export a series the user needs to click in the panel beneath which series will be exported (Amplitude or Phase). This will give the user a `.txt` file with the X and Y values of the plot in a new line. To import a series the user must submit a `.txt` file which must contain in every new line the X and Y values to be plotted.

List of Series: In this panel will appear all the series that are being plotted into the graph. Here the user can customize each plot or remove the desired ones. The customization options are:

- View points: Enable this option to visualize the points where the field has been computed for every inserted plot.
- Change series stroke: Change the composition of the plots from a preset of options.
- Change Color: Change the color of the selected plot.
- Remove Plot: Delete a chosen plot from the graph.

View Monostatic Cuts

This menu is only enabled when the Method of Moments is selected on Solver section, and the option Monostatic is selected in the Source - Plane wave section.

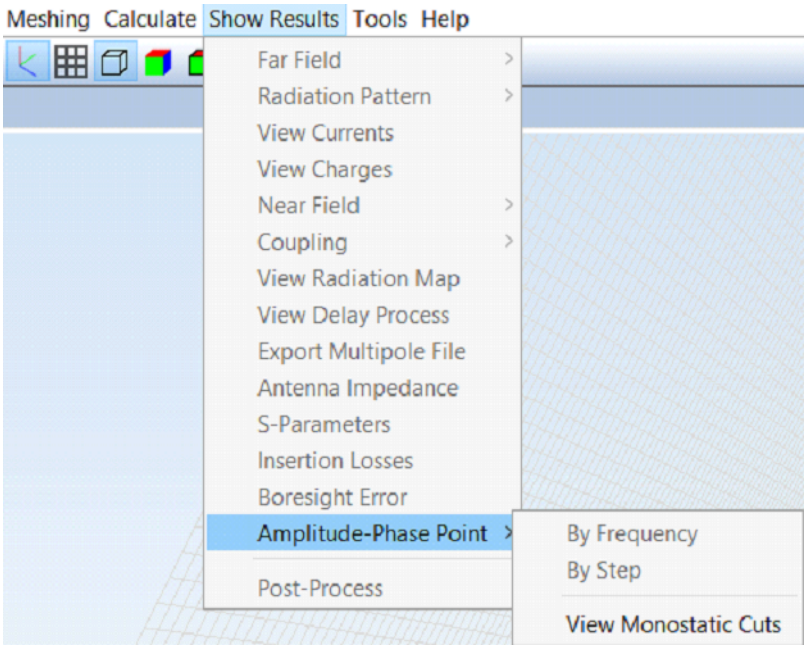


Figure 1181: Figure:Amplitude-Phase Point results menu

When the user clicks on the View Monostatic Cuts option, the following panel will appear:

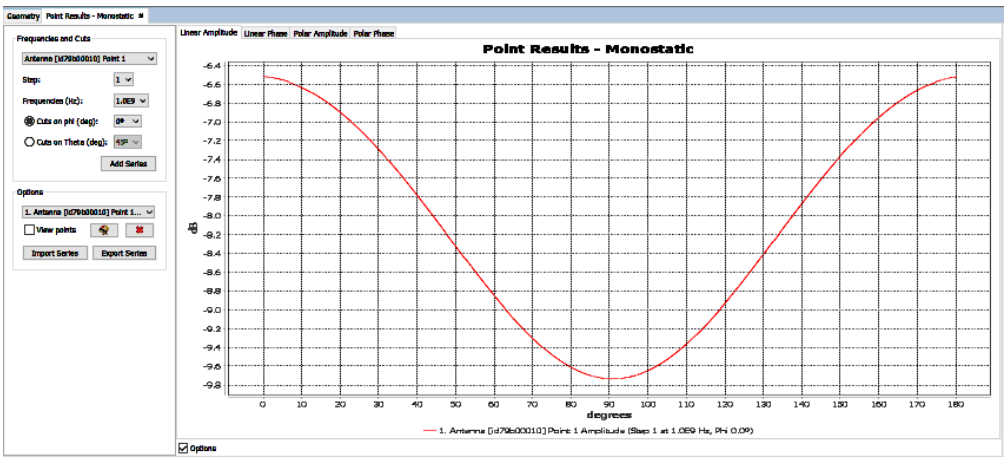


Figure 1182: Monostatic View Cuts panel. Linear Amplitude

To plot a graph, select the desired parameters: **Frequencies and Cuts** Here the user is able to select a feeding point, its Step and Frequencies and the list of Phi or Theta cuts to add the defined serie to the graph. Then, click on Add Series button to plot the selected curve.

Options This menu allow the user to edit the result curves.

- **List of Series** All the plotted polarizations and components are listed. Note that two options are included for every plot both the amplitude and the phase.
- **View points** Enable this option to visualize the points where the field has been computed for every inserted plot.

- **Change color** This option allows to change the color of the selected curve in the above List of Series.
- **Remove** To delete the selected curve in the above List of Series.
- **Import Series** To add a new plot from an external file. The file must have `.txt` extension, and it must contain in every new line the X and Y values to be plotted.
- **Export Series** To export the selected curve in the above Plot List in a `.txt` file, with the X and Y values of the plot in a new line.

The user can also switch between the **linear graph** and the **polar graph** to visualize far field results. It is possible to visualize Linear Amplitude, Linear Phase, Polar Amplitude and Polar Phase by clicking on their respective tabs.

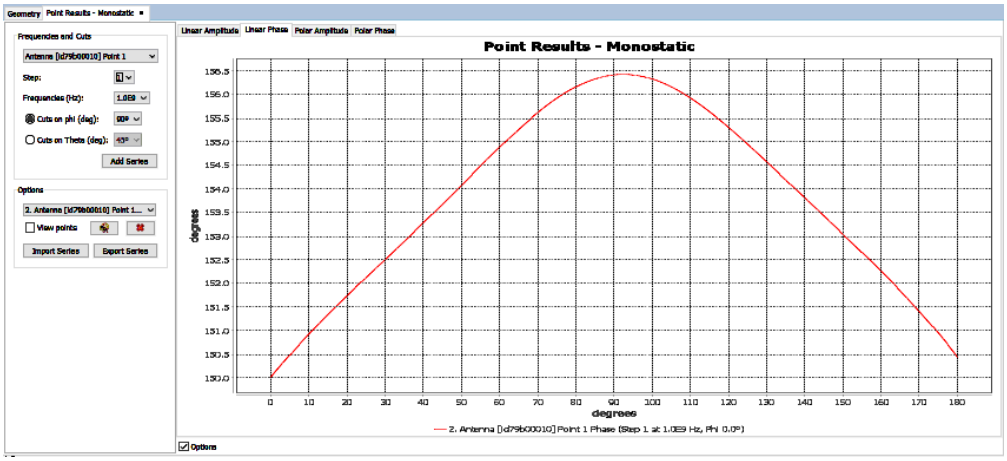


Figure 1183: Monostatic View Cuts panel. Linear Phase

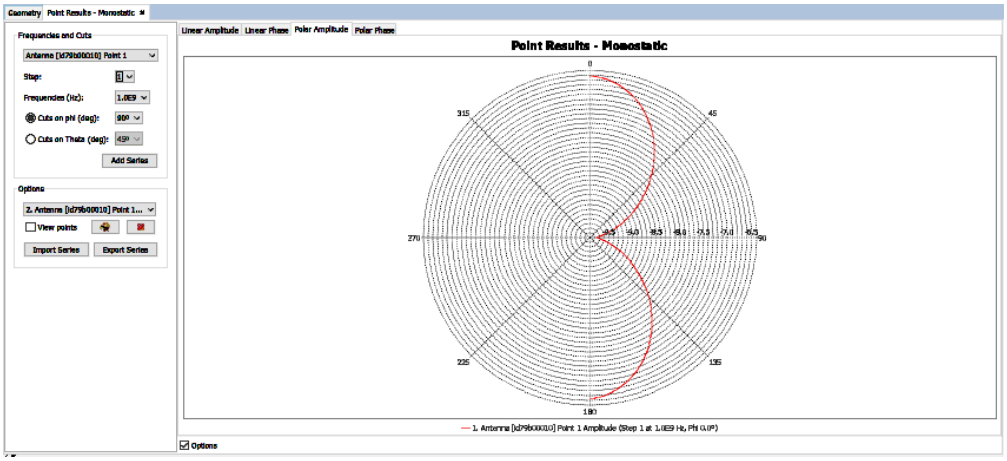


Figure 1184: Monostatic View Cuts panel. Polar Amplitude

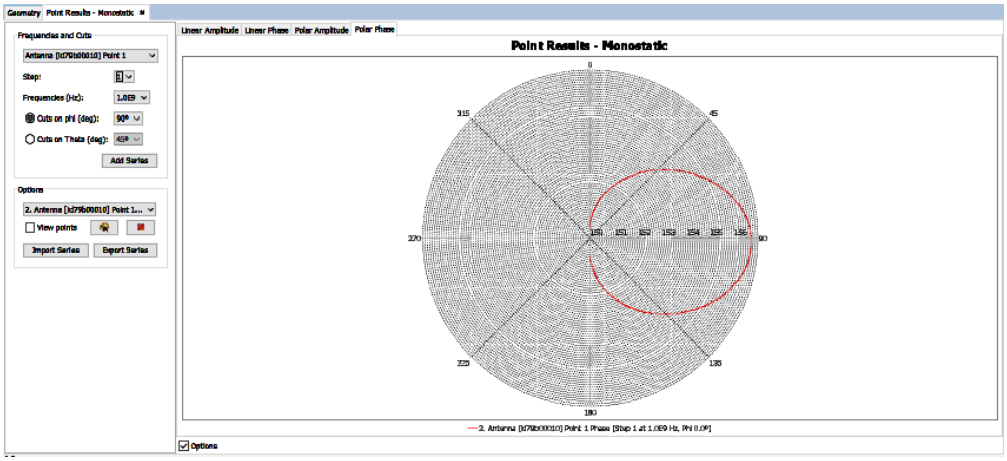


Figure 1185: Monostatic View Cuts panel. Polar Phase

11.13 Tools Menu

For information about the Tools menu, refer to the Tools.

11.14 Help Menu

For information about the Help menu, refer to Help Menu.

11.15 Training Examples

In this section, there are some examples showing the basic use of the MOM module. For further information about antennas simulations, see Antennas.

11.15.1 Example 1: Common Simulation

This case explains how to calculate far-field, radiation pattern, current density, charge density and near field of an open box with a dipole antenna insides the box.

Step 1: Create a new MOM Project.

Open 'New Fasant' and select 'File --> New' option.

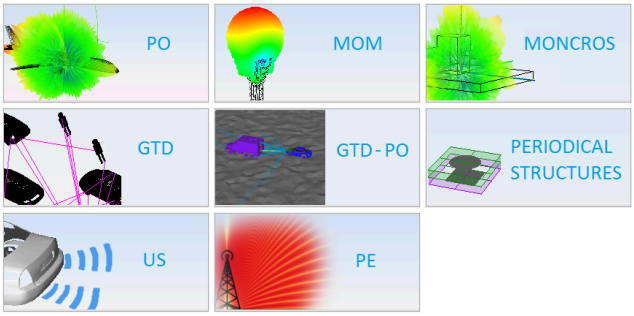


Figure 1186: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To get the complete information about geometries edition see 4. Geometry Menu.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

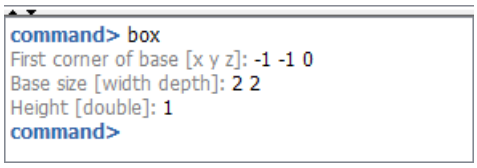


Figure 1187: Box parameters

Select box geometry on the main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

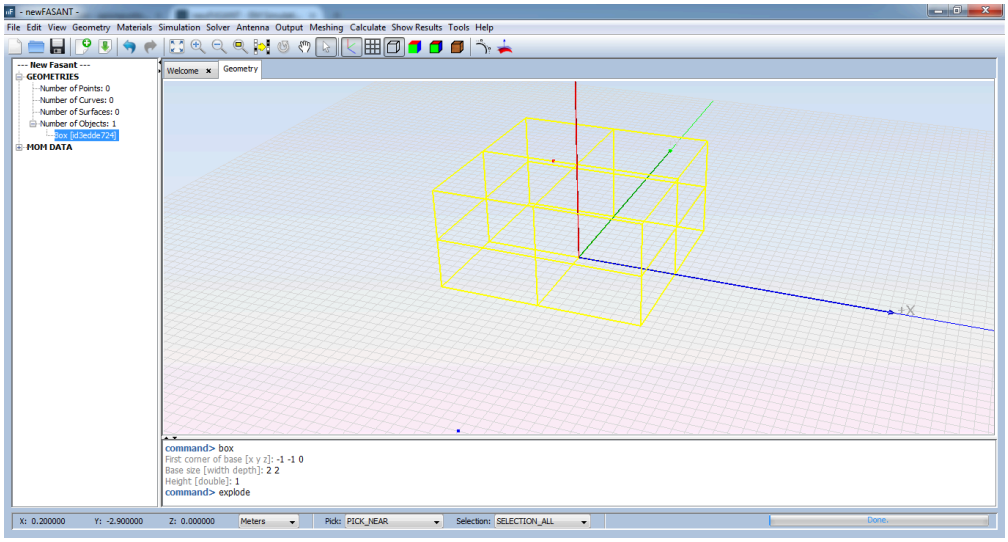


Figure 1188: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

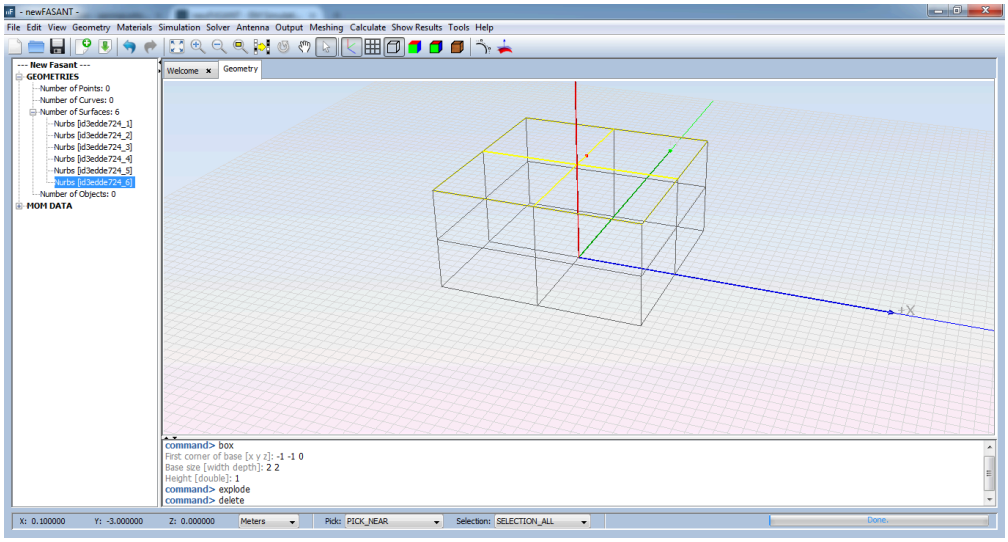


Figure 1189: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

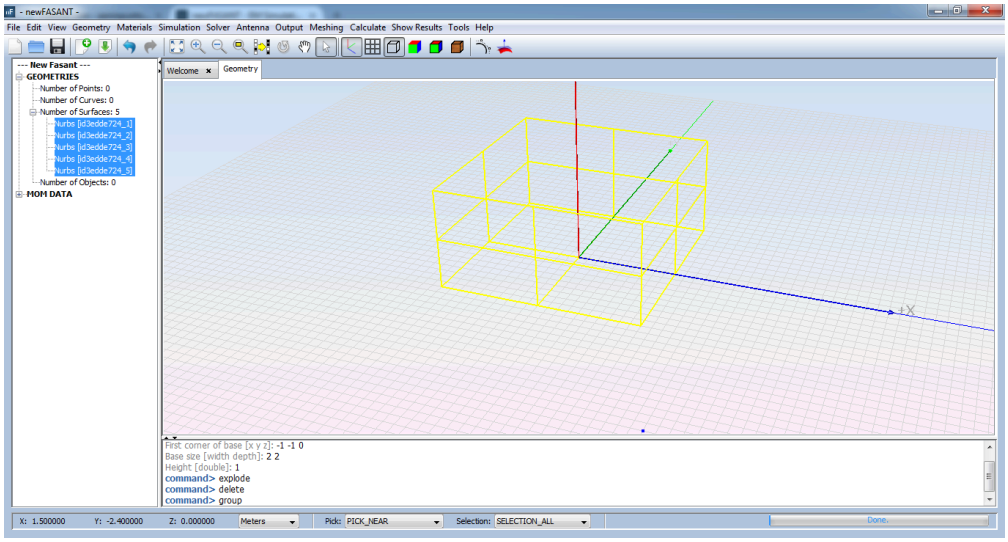


Figure 1190: Surfaces selection and 'group' command

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

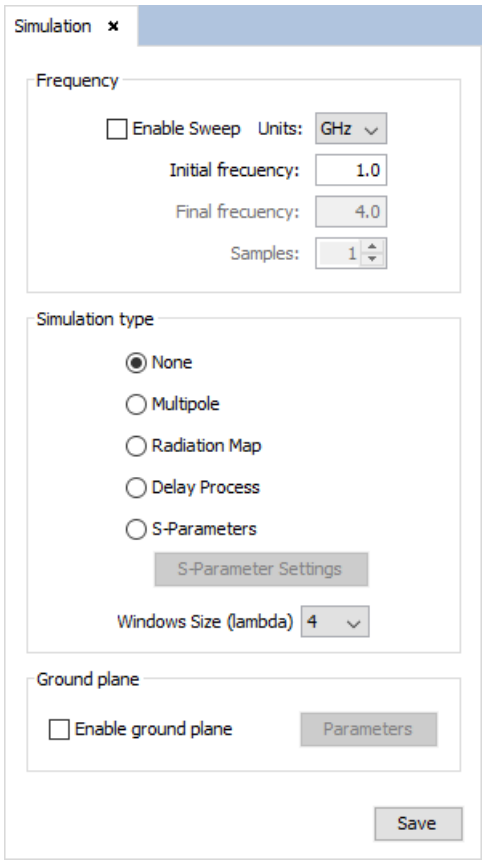


Figure 1191: Simulation Parameters panel

Step 4: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

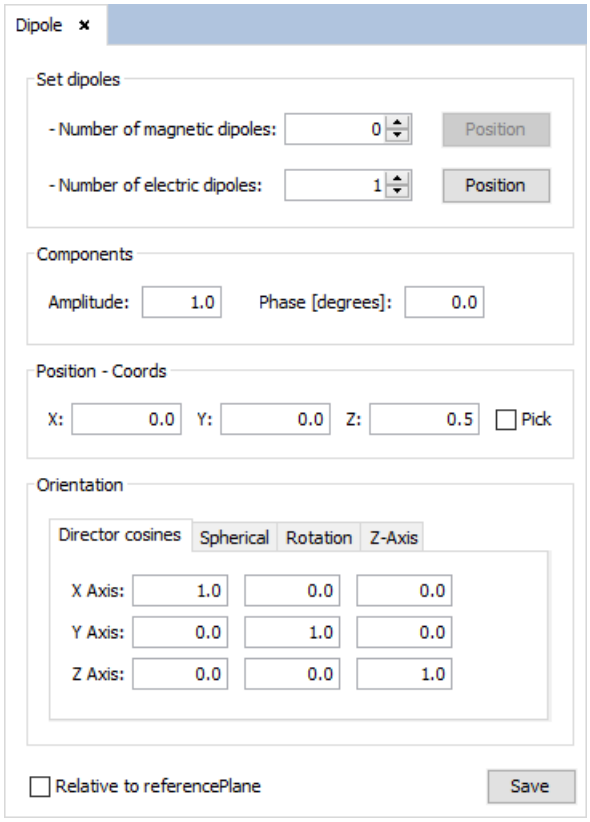


Figure 1192: Dipole Antenna panel

Step 5: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

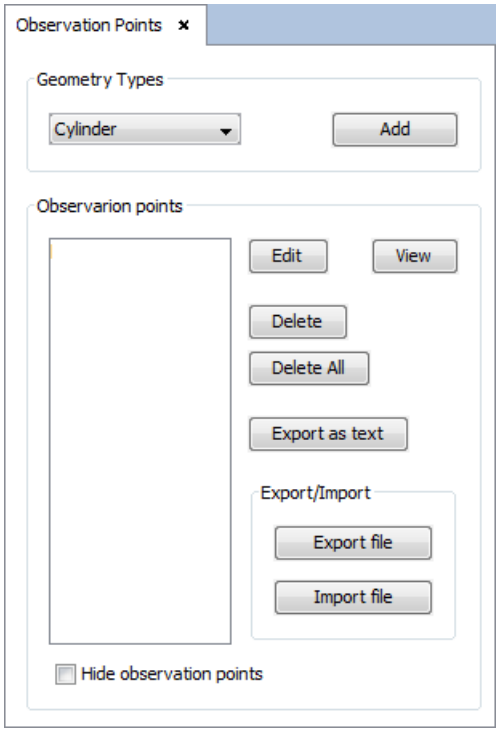


Figure 1193: Observation Points panel

To add a line visualization, select 'line' on the selector of 'Geometry Types' section and click on 'Add' button. The line parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

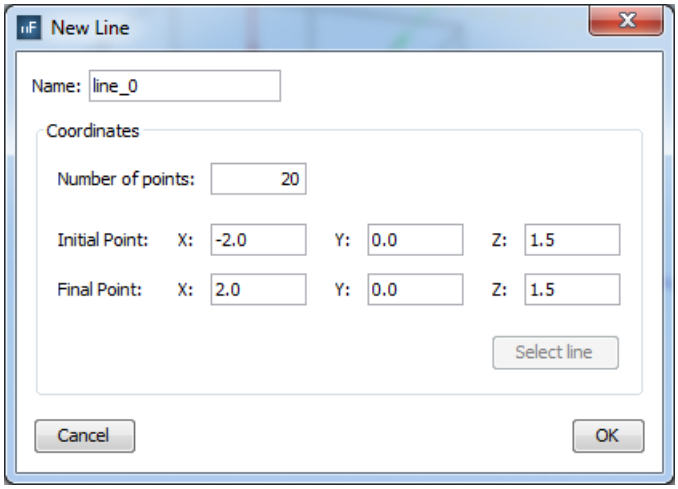


Figure 1194: Observation Line panel

The observation line will appear as a dashed line on the position configured.

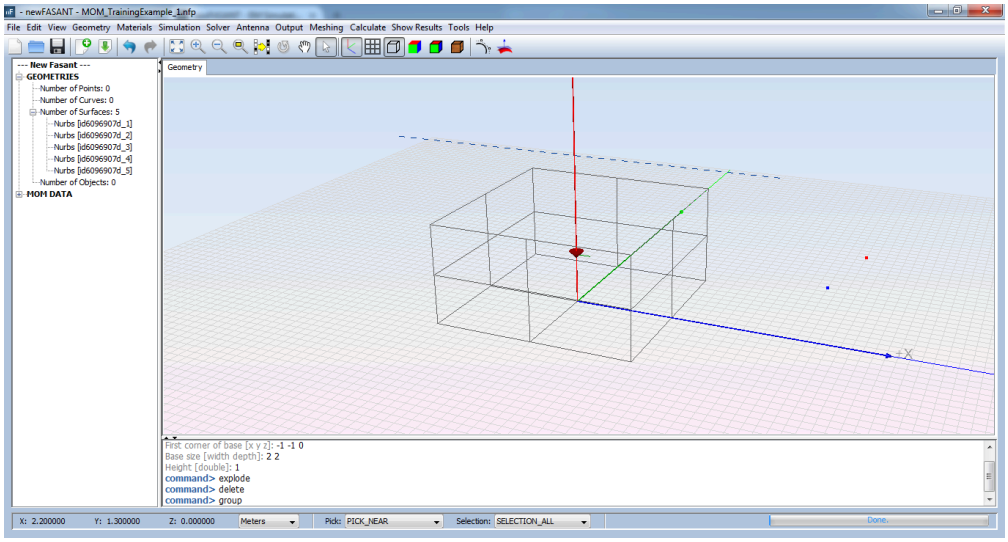


Figure 1195: Observation Line visualization

Step 6: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

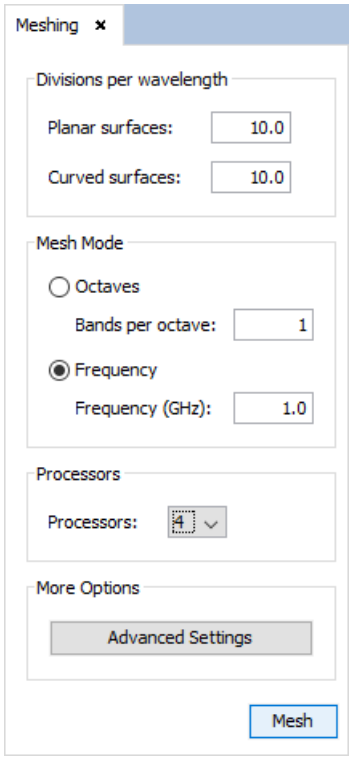


Figure 1196: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

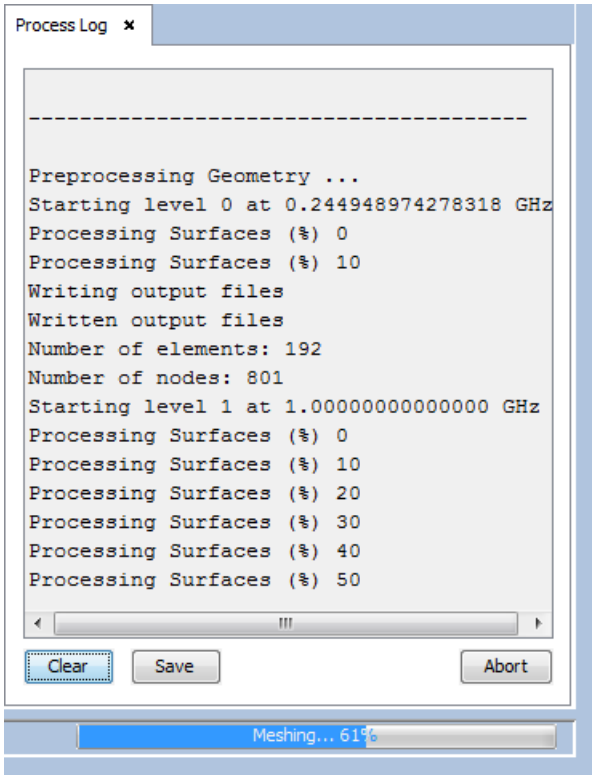


Figure 1197: Meshing process log

Step 7: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

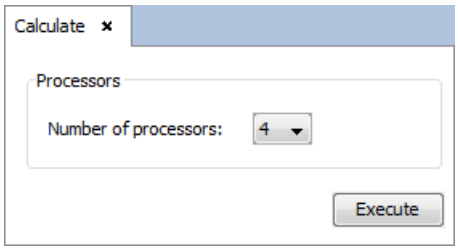


Figure 1198: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

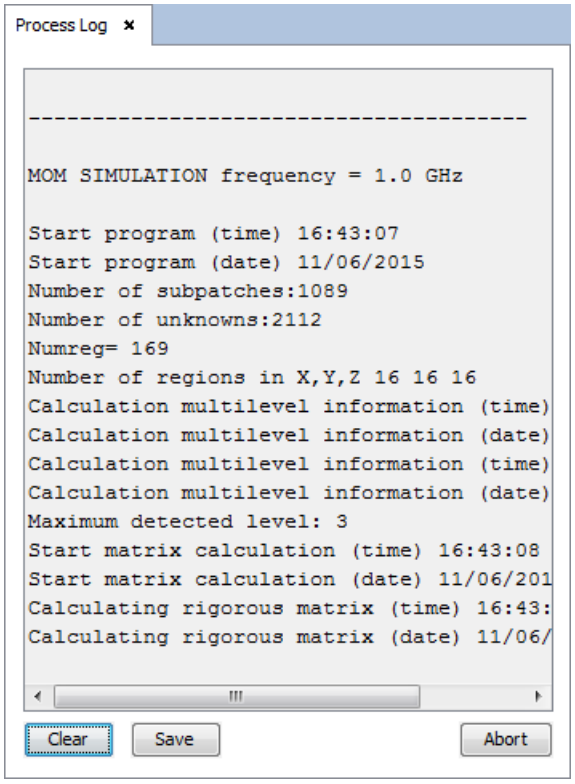


Figure 1199: Execute process log

Step 8: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

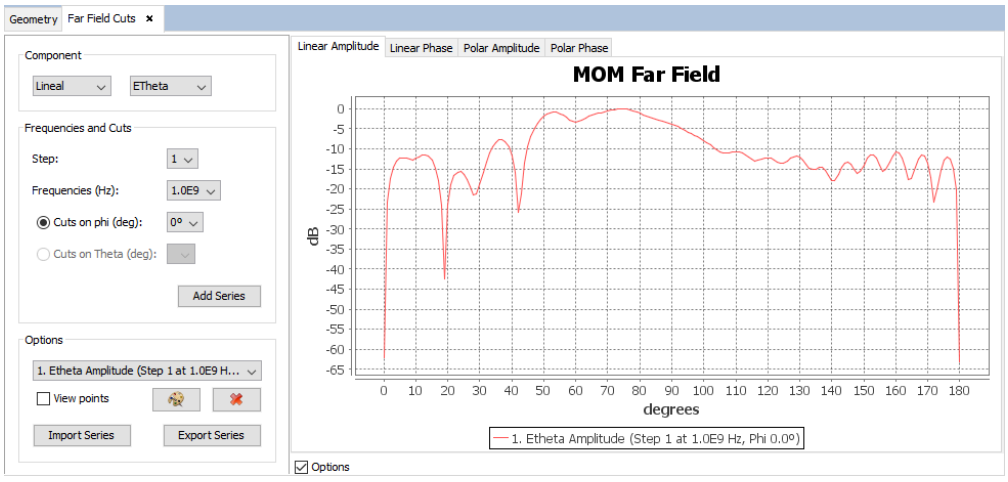


Figure 1200: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field'

menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

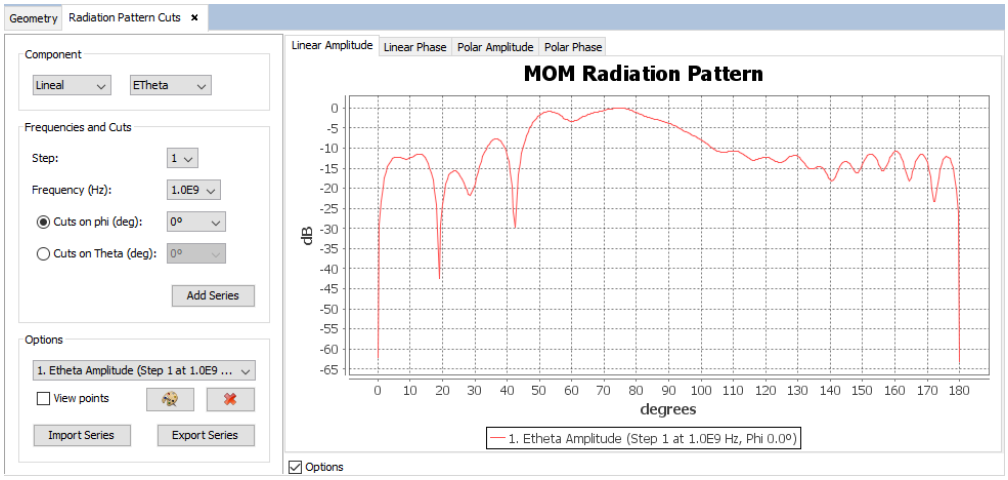


Figure 1201: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

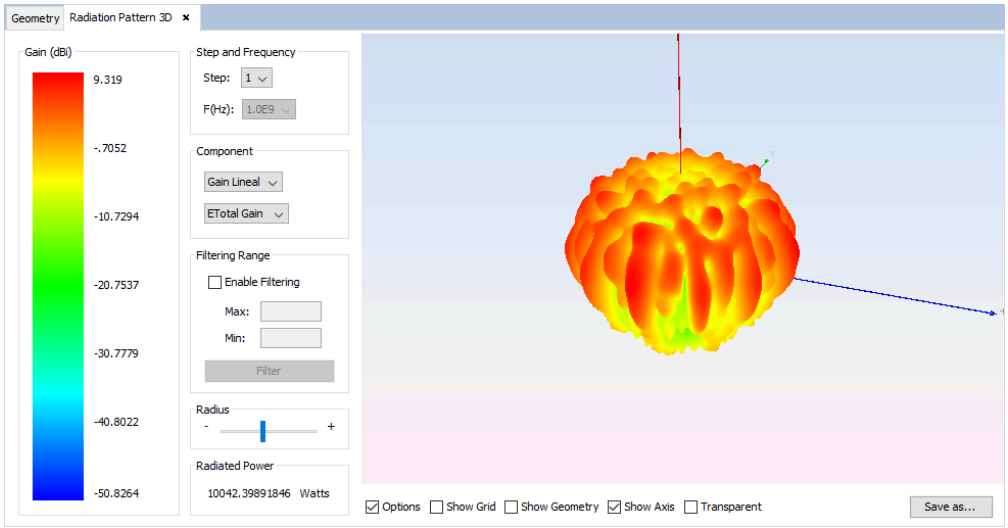


Figure 1202: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

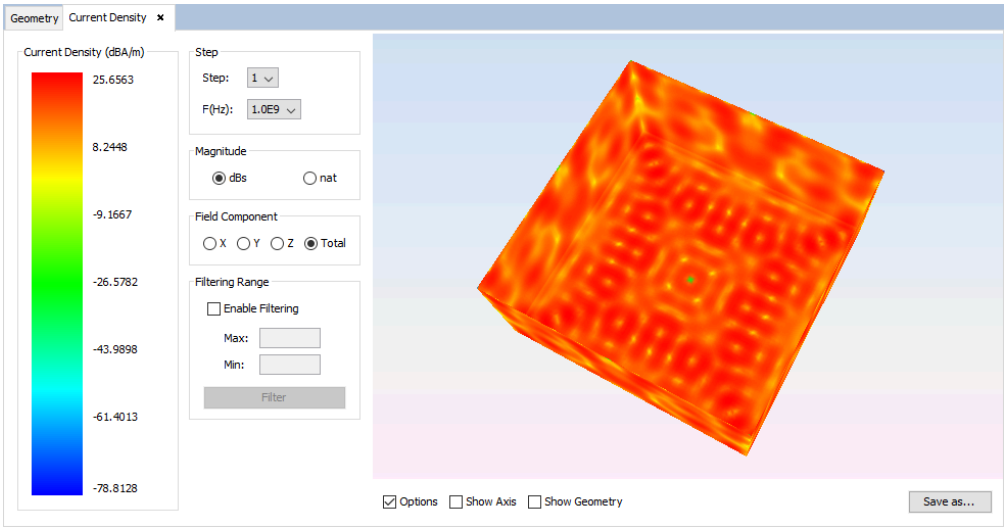


Figure 1203: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

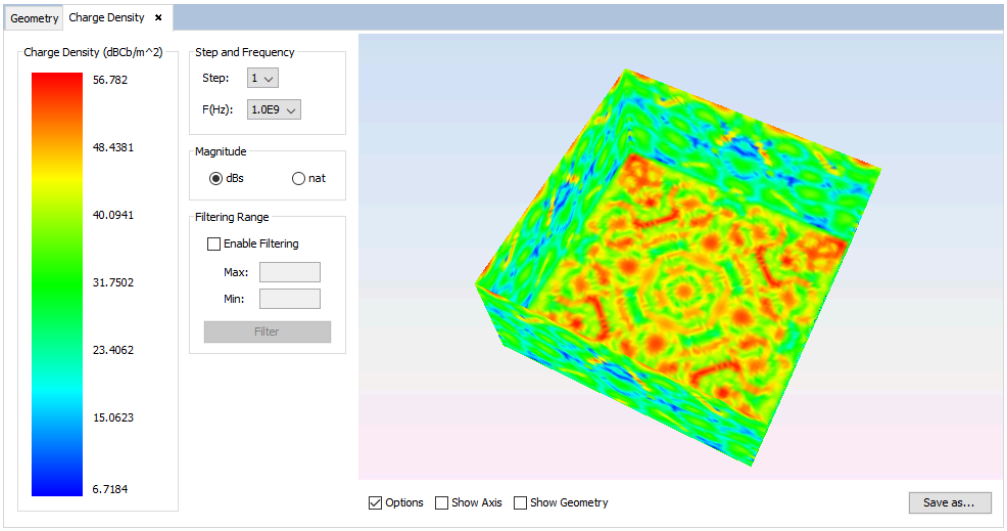


Figure 1204: Charge Density

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> Near Field --> View Near Field Diagram' option to show the observation points diagram. Previously, select the observation to visualize and the step and frequency on the next figure.

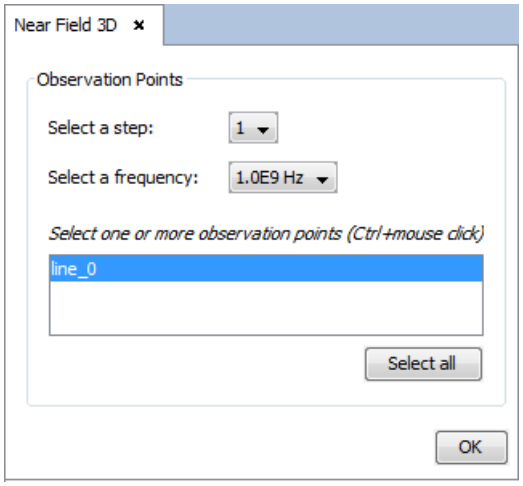


Figure 1205: Near Field diagram selection

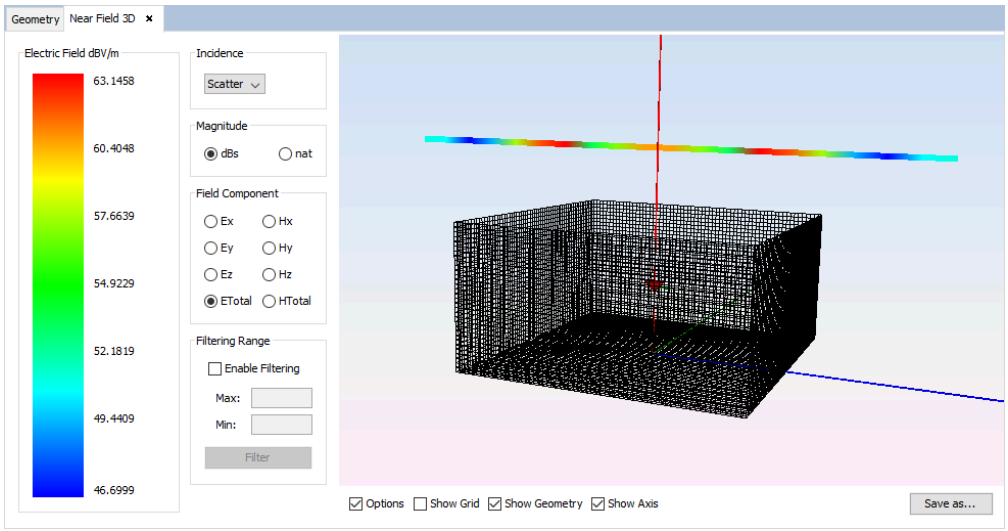


Figure 1206: Near Field Diagram

Changing values for incidence, magnitude, field or component parameters the visualization for the new parameters will be shown.

Select 'Show Results --> Near Field --> View Observation Points' option to show the observation points chart.

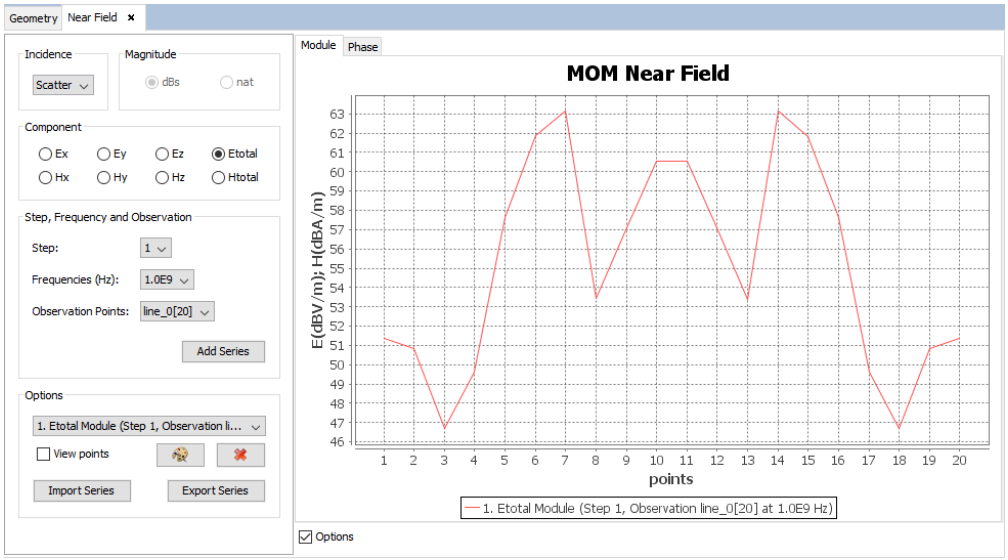


Figure 1207: Near Field Observation Points cuts

Selecting other values for incidence, component, step, frequency or observation parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

11.15.2 Example 2: Common Parametric Simulation with Frequency Sweep

This case explains how to calculate far-field, radiation pattern, current density, charge density, near field and coupling of a parametric open box with a dipole antenna insides the box and with a frequency sweep from 0.5 GHz to 1.5GHz with 3 samples.

Step 1: Create a new MOM Project.

Open newFASANT' and select 'File --> New' option.

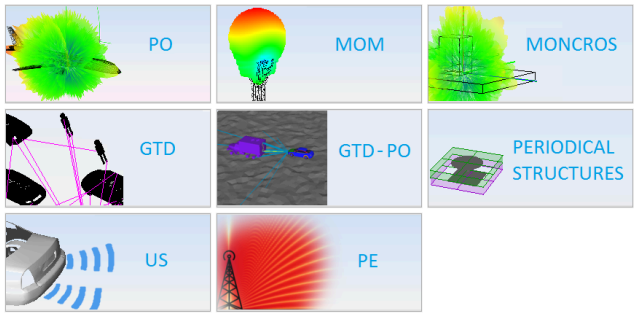


Figure 1208: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create parameters for the parametric geometry model.

Select 'Geometry --> Parameters --> Define Parameters' option from the menu bar. Then, on the panel, set the follow parameters. 'side' parameter will be the values for the size of the box in width and depth, 'h' parameter will be the height value on the box and 'initPoint' will be the value for the initial point coordinates on the base of the box. To obtain more information about parameters definition see Parameters.

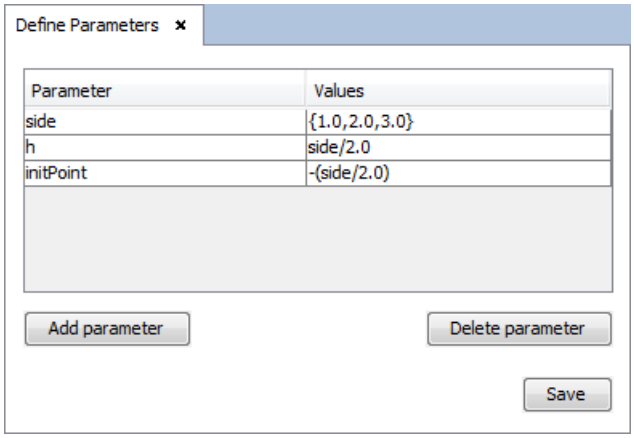


Figure 1209: Parameters panel

Step 3: Create the geometry model. To obtain more information about geometries generation see Parameters.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

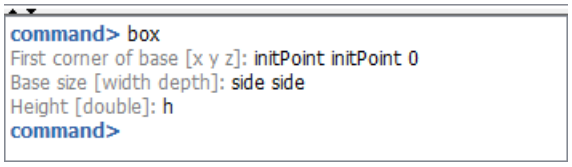


Figure 1210: Box parameters

Select box geometry on main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

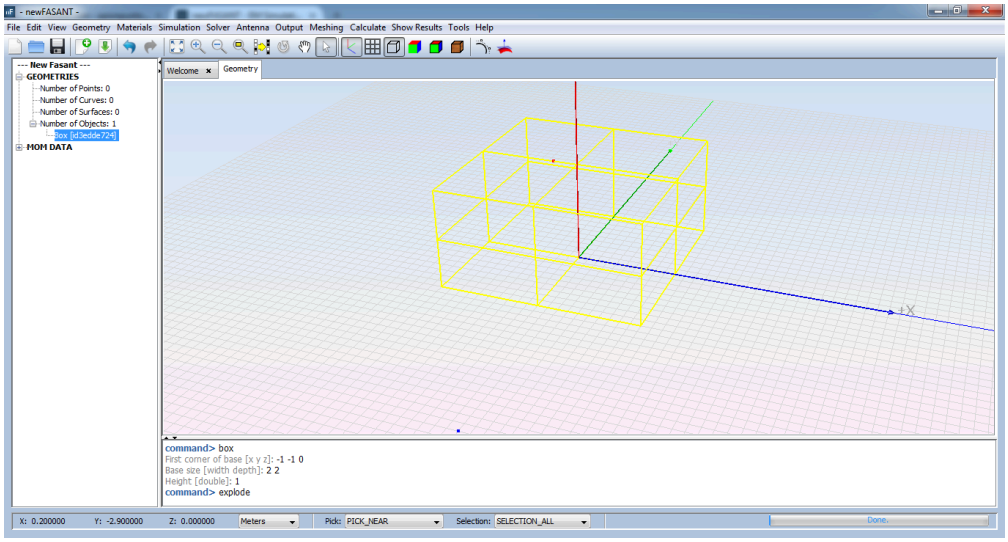


Figure 1211: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

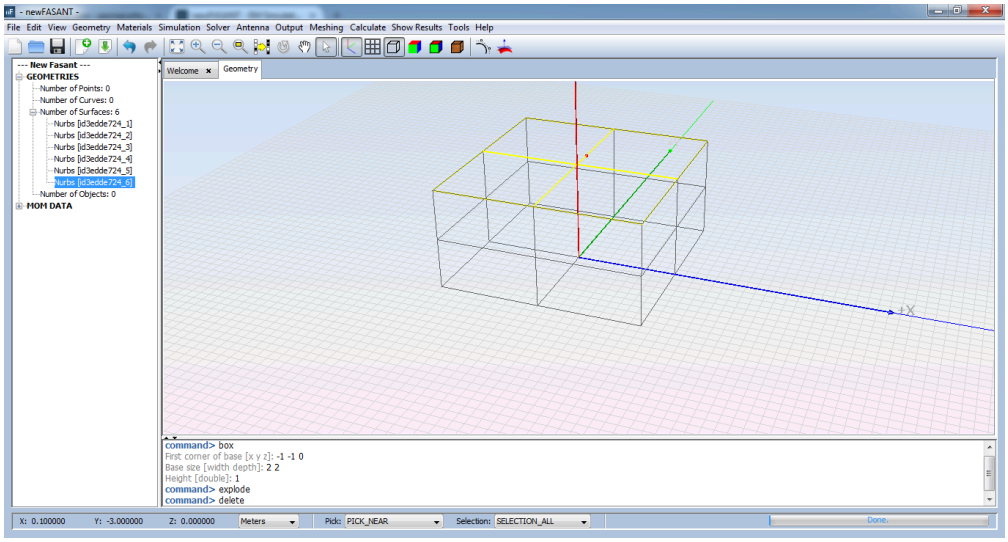


Figure 1212: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

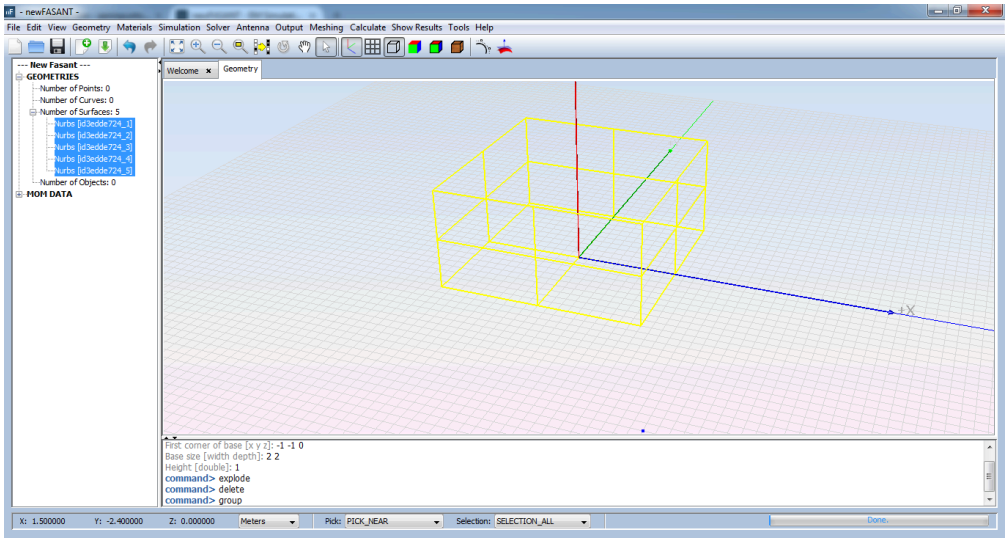


Figure 1213: Surfaces selection and 'group' command

Step 4: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

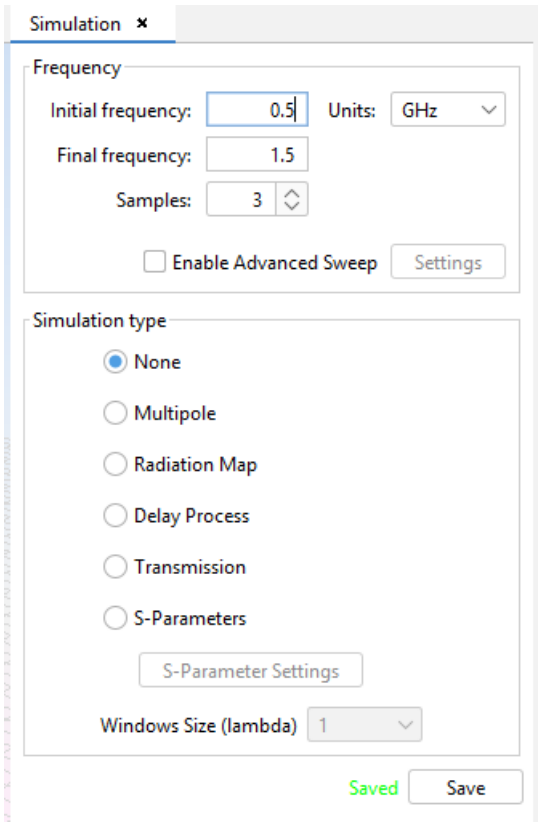


Figure 1214: Simulation Parameters panel

Step 5: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

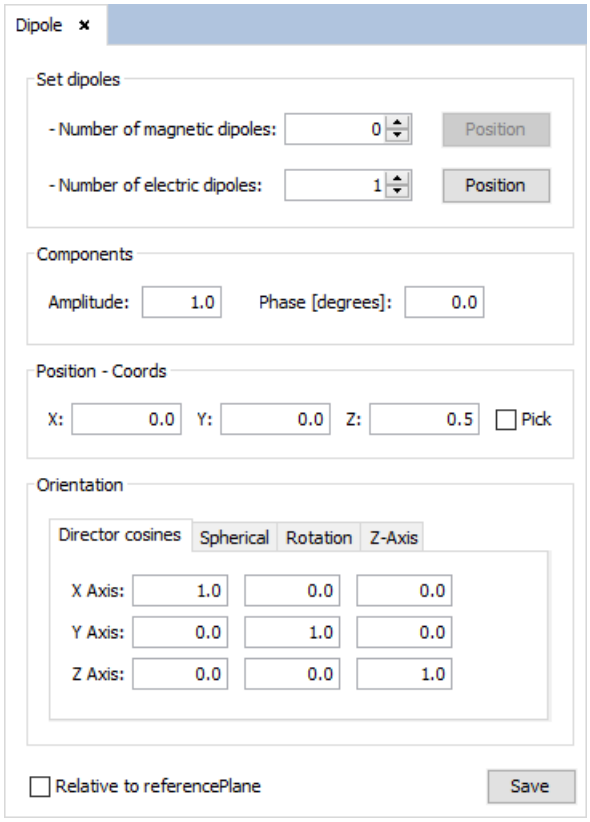


Figure 1215: Dipole Antenna panel

Step 6: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

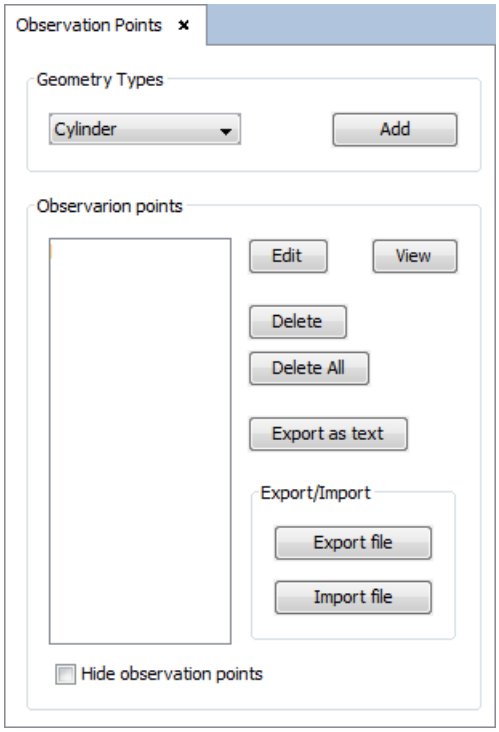


Figure 1216: Observation Points panel

To add a line visualization, select 'line' on the selector of 'Geometry Types' section and click on 'Add' button. The line parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

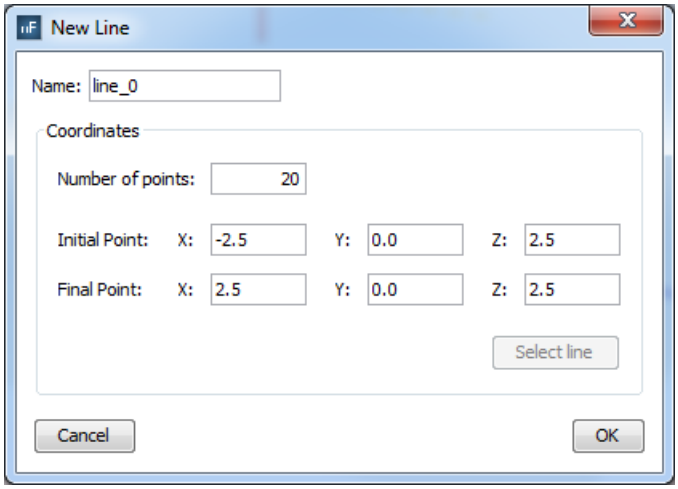


Figure 1217: Observation Line panel

The observation line will appear as a dashed line on the position configured.

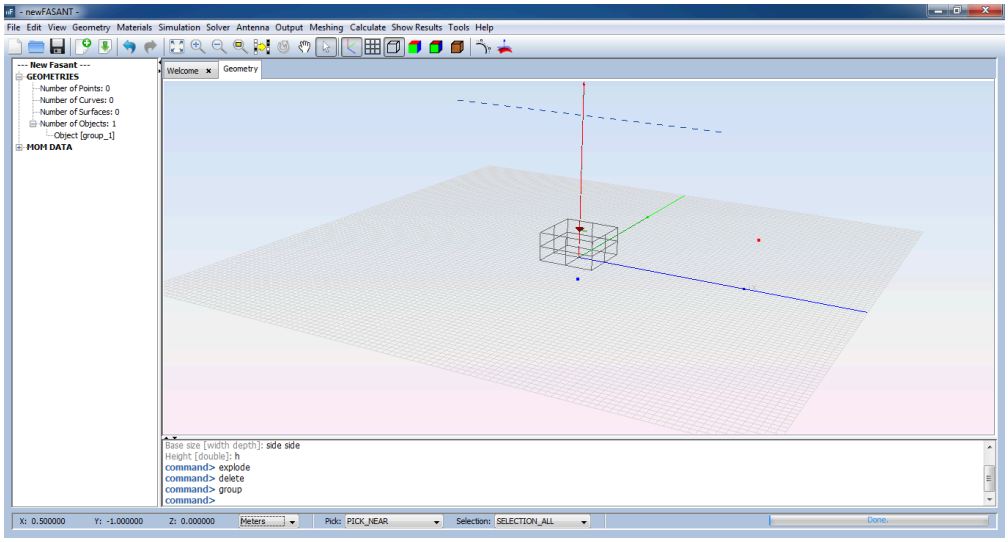


Figure 1218: Observation Line visualization

Step 7: Set Passive Antenna parameters for coupling results.

Select 'Output --> Passive Antenna' option. On the panel show, select the 'Enable / Disable Coupling' check box and set the parameters as the next figure. Then save the parameters and the passive antenna appears as a magenta dipole.

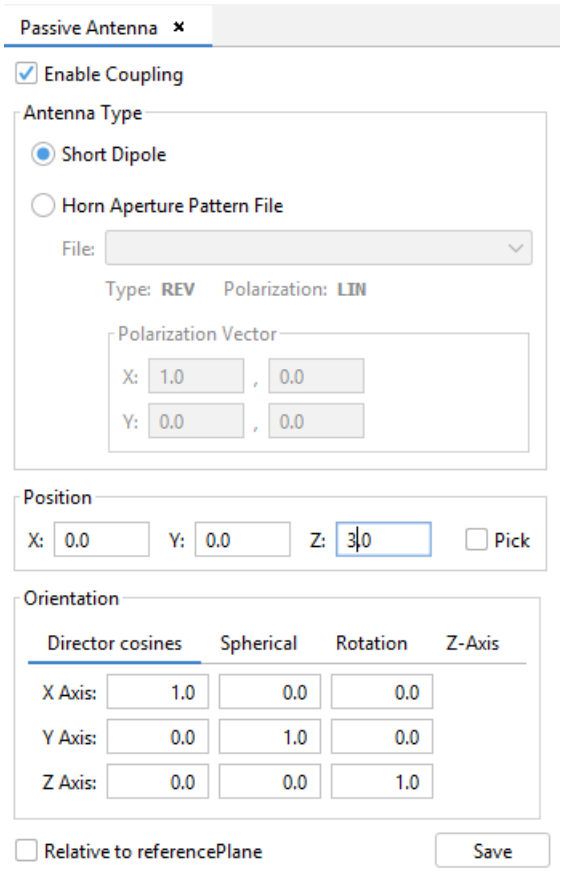


Figure 1219: Passive Antenna panel

Step 8: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

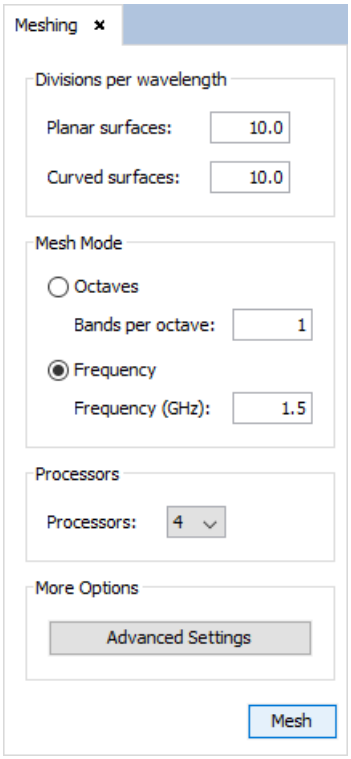


Figure 1220: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

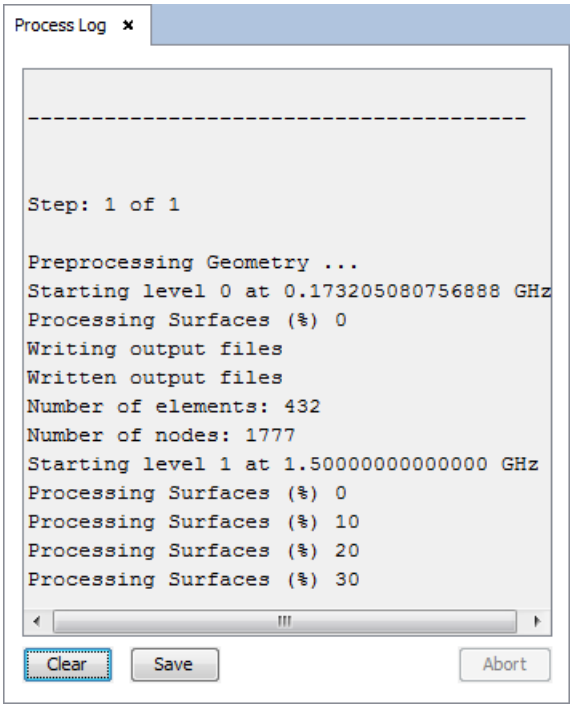


Figure 1221: Meshing process log

Step 9: Execute the simulation. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

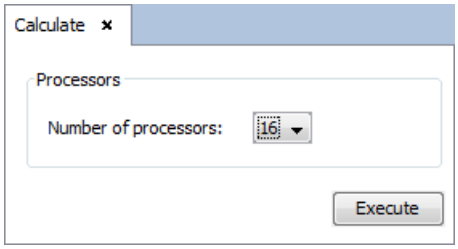


Figure 1222: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

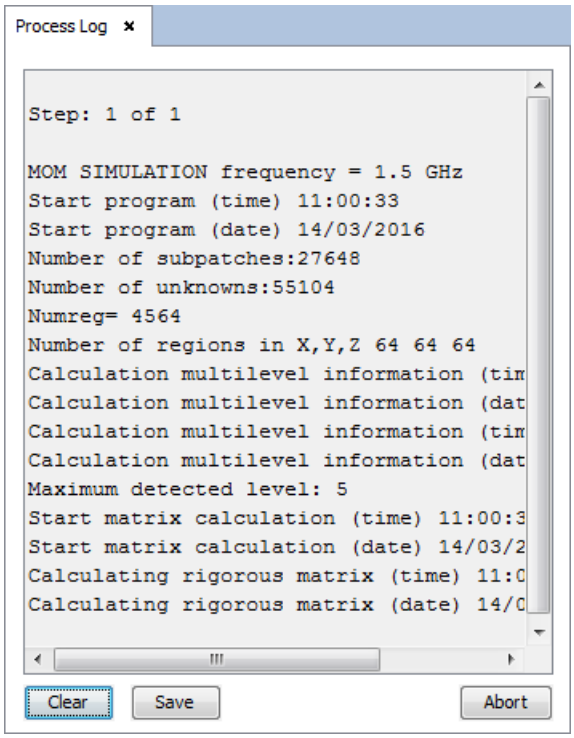


Figure 1223: Execute process log

Step 10: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

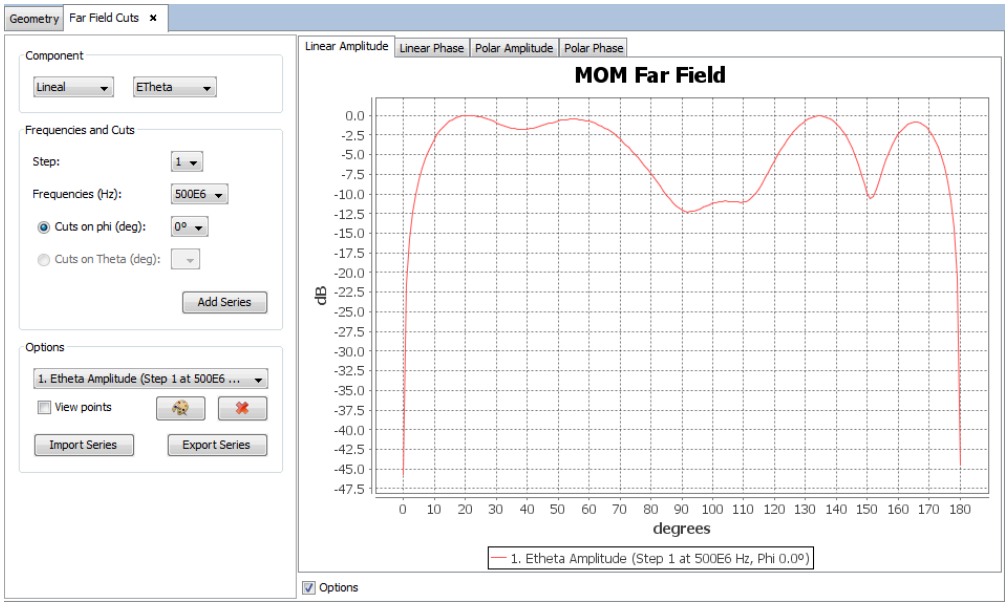


Figure 1224: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters.

On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

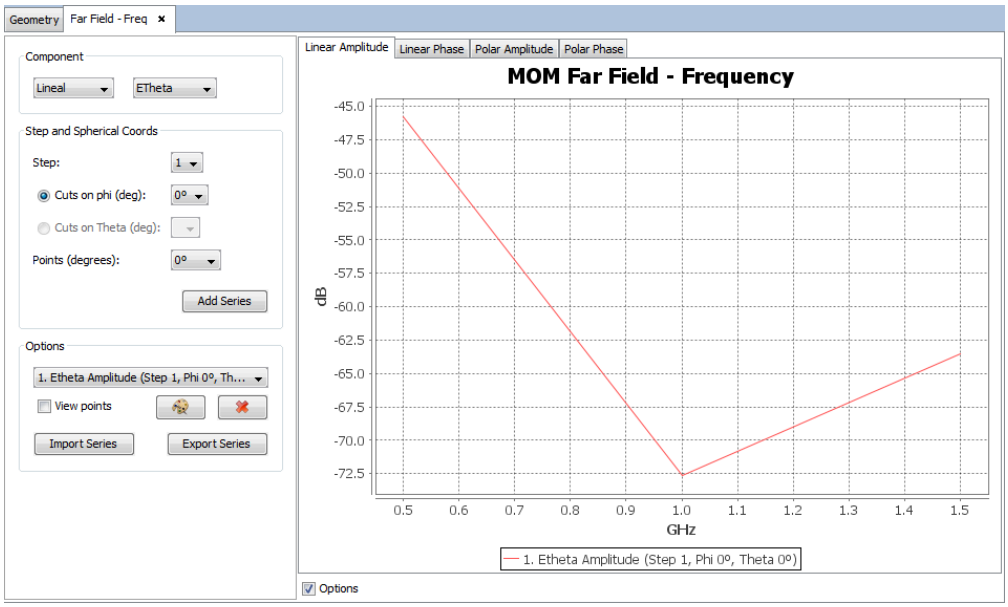


Figure 1225: Far Field by Frequency cuts

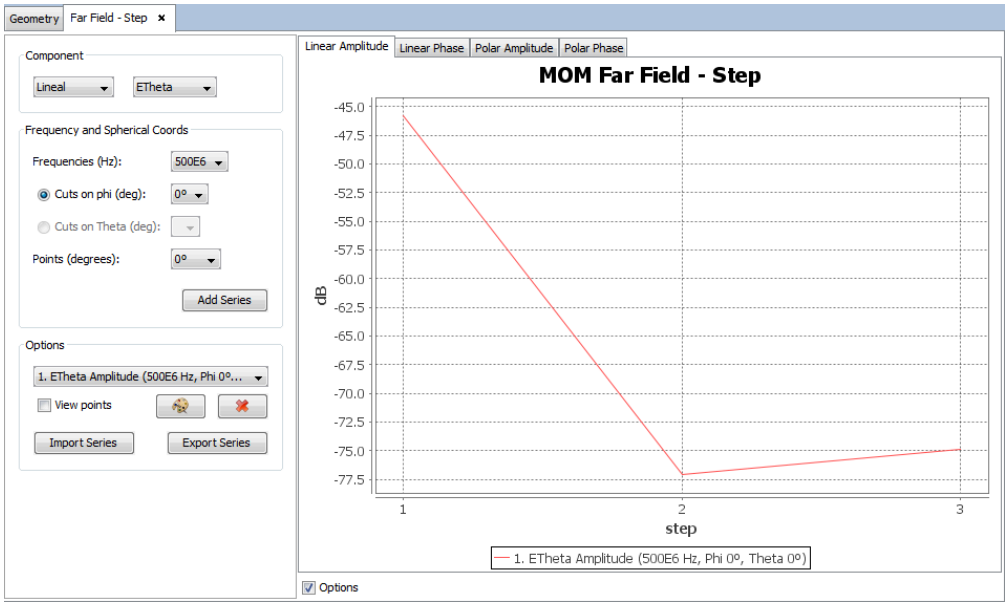


Figure 1226: Far Field by Step cuts

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

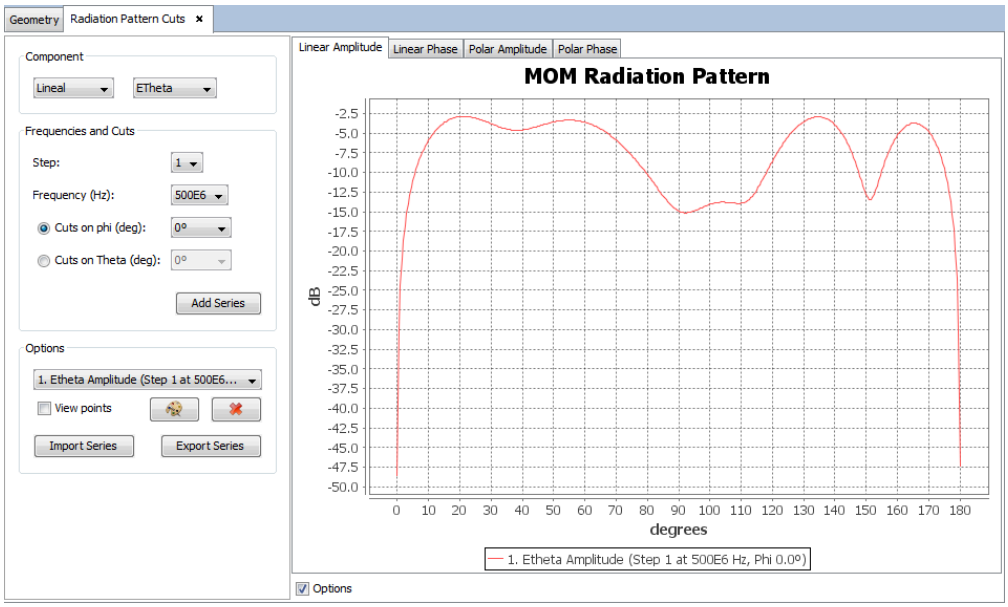


Figure 1227: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters.

On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

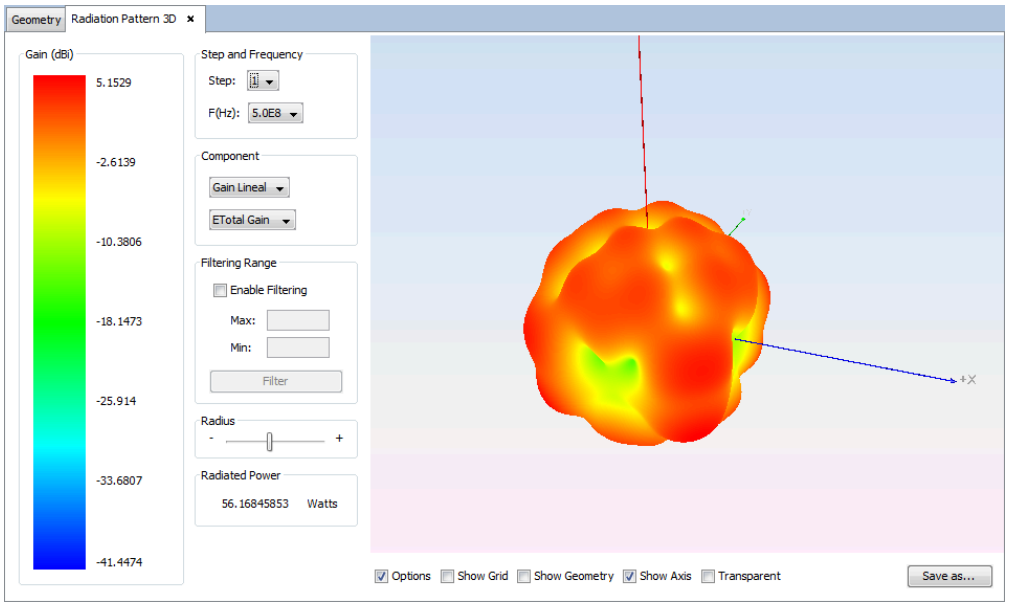


Figure 1228: Radiation Pattern 3D for step 1 and frequency 0.5 GHz

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

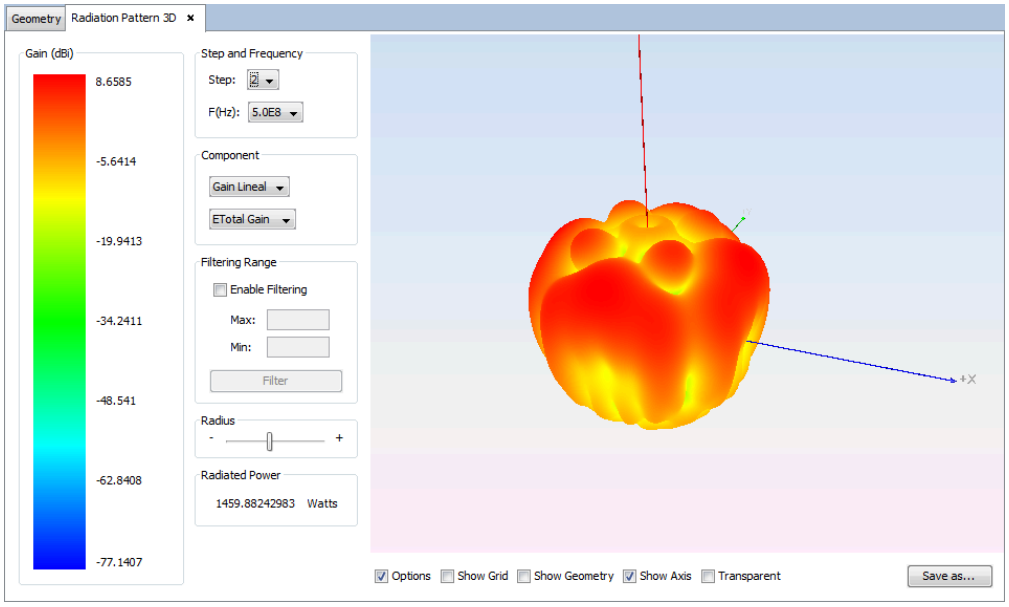


Figure 1229: Radiation Pattern 3D for step 2 and frequency 0.5 GHz

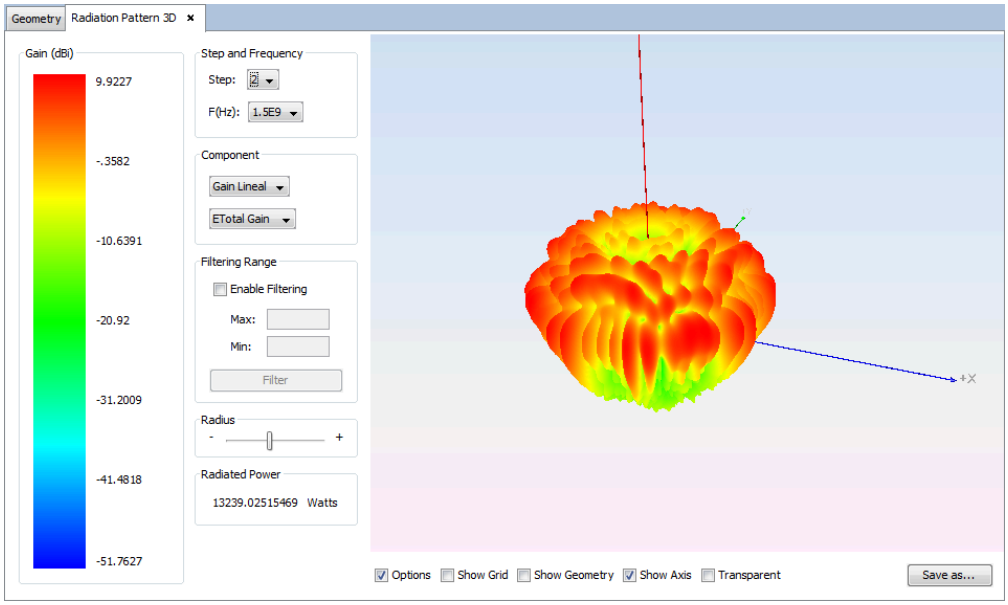


Figure 1230: Radiation Pattern 3D for step 2 and frequency 1.5 GHz

Select 'Show Results --> View Currents' option to show the current density.

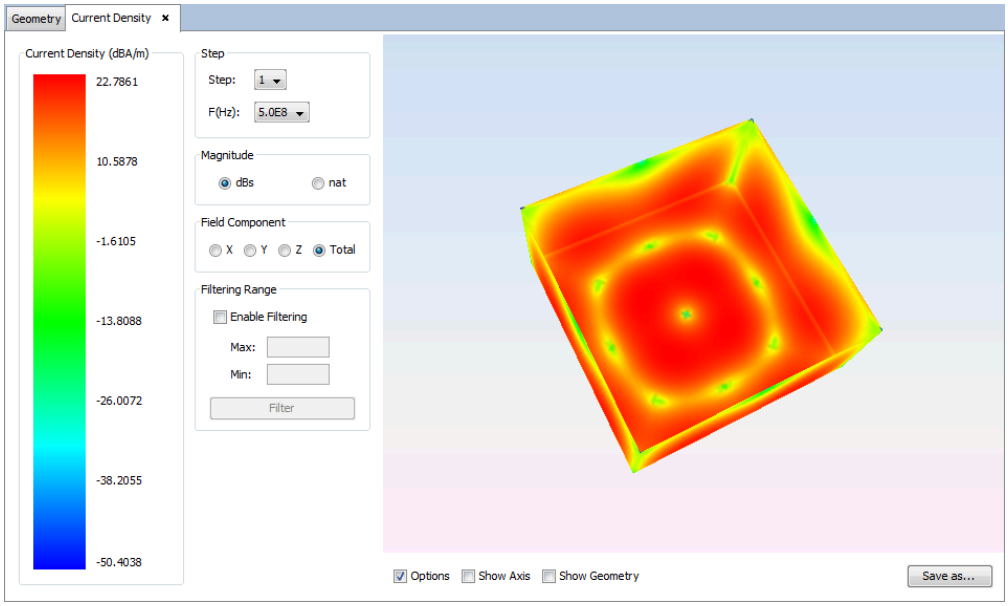


Figure 1231: Current Density for step 1 and frequency 0.5 GHz

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

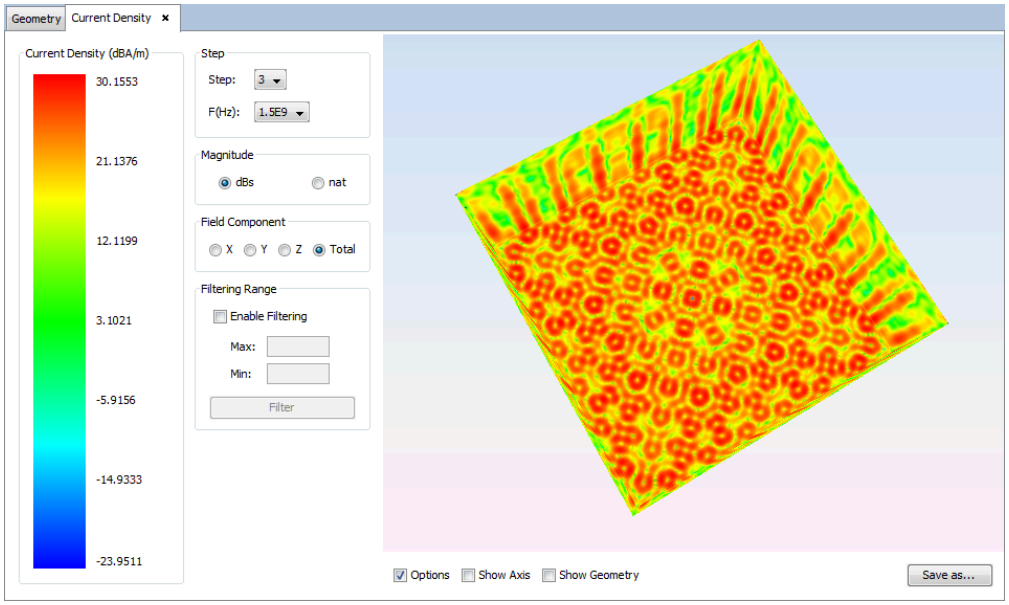


Figure 1232: Current Density for step 3 and frequency 1.5 GHz

Select 'Show Results --> View Charges' option to show the charge density.

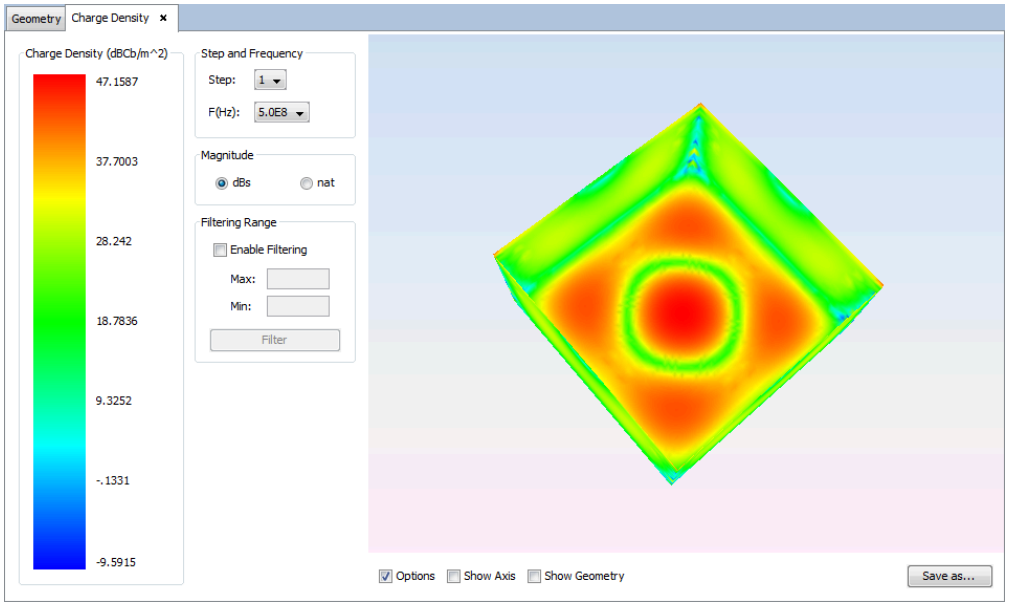


Figure 1233: Charge Density for step 1 and frequency 0.5 GHz

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

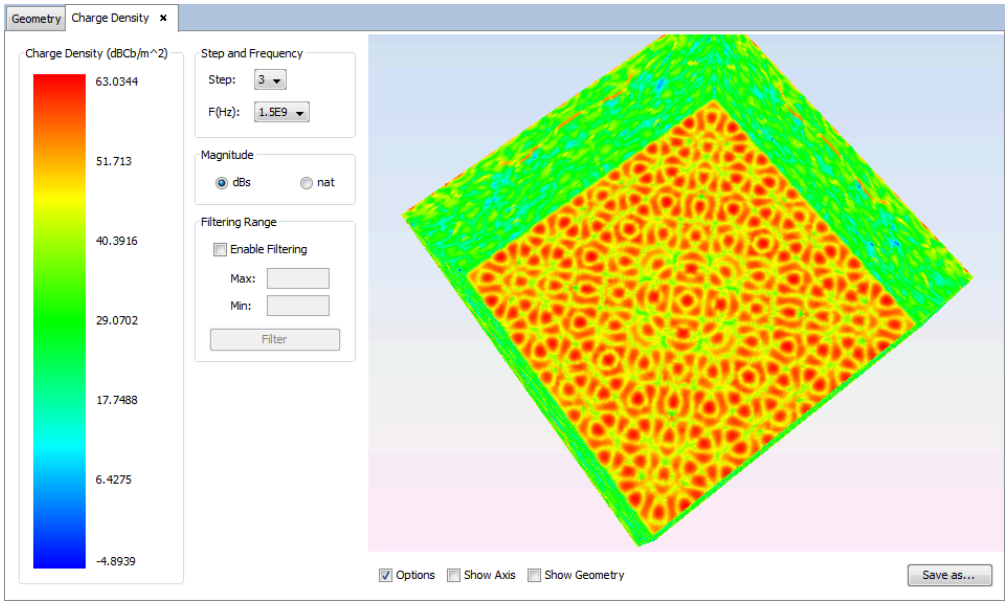


Figure 1234: Charge Density for step 3 and frequency 1.5 GHz

Select 'Show Results --> Near Field --> View Near Field Diagram' option to show the observation points diagram. Previously, select the observation to visualize and the step and frequency on the next figure.

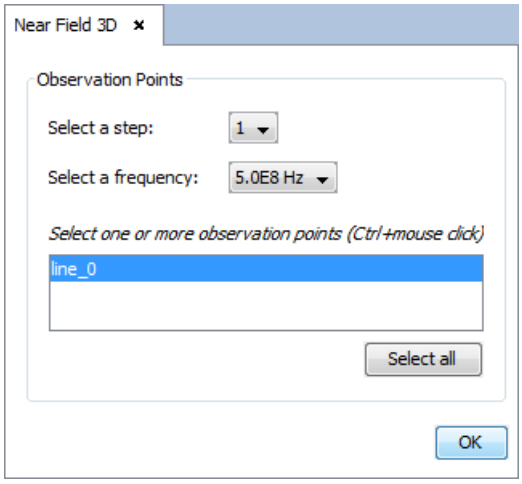


Figure 1235: Near Field diagram selection

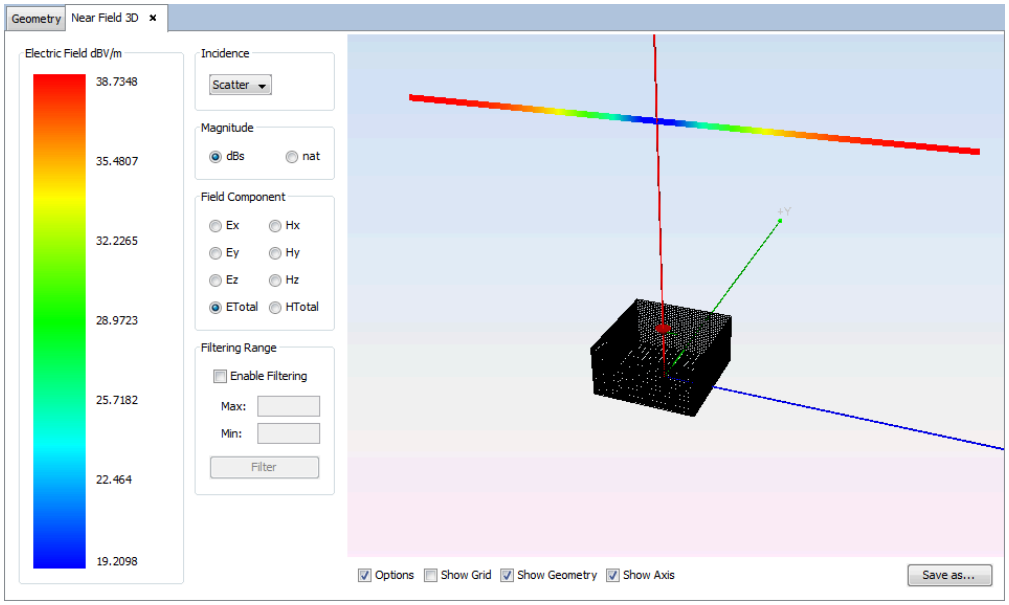


Figure 1236: Near Field Diagram for step 1 and frequency 0.5 GHz

Changing values for incidence, magnitude, field or component parameters the visualization for the new parameters will be shown.

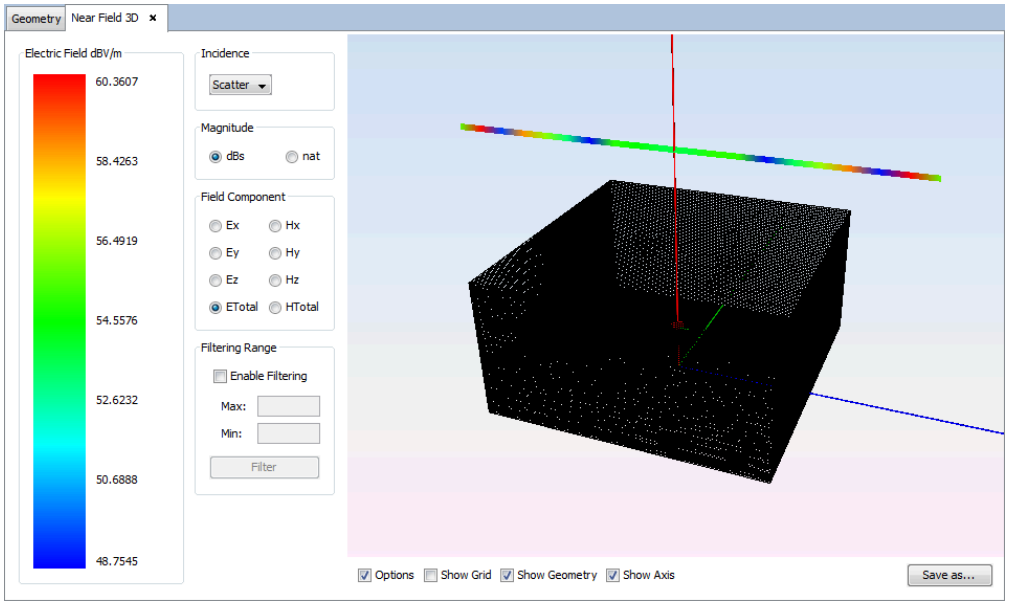


Figure 1237: Near Field Diagram for step 3 and frequency 1.5 GHz

Select 'Show Results --> Near Field --> View Observation Points' option to show the observation points chart.

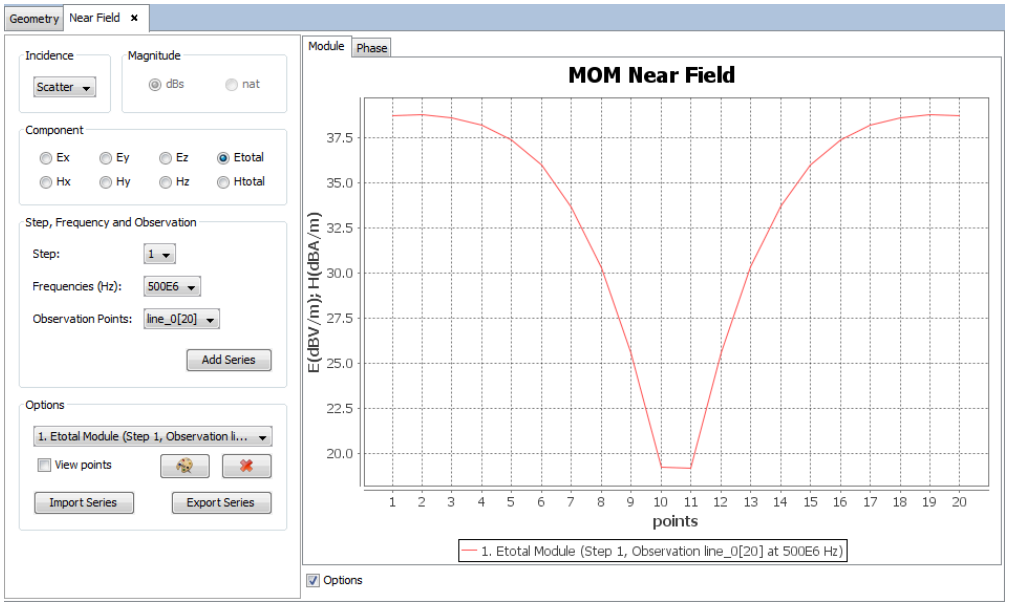


Figure 1238: Near Field Observation Points cuts

Selecting other values for incidence, component, step, frequency or observation parameters and clicking on 'Add Series' button a new cut will be added for the selected parameters.

On 'Show Results --> Near Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

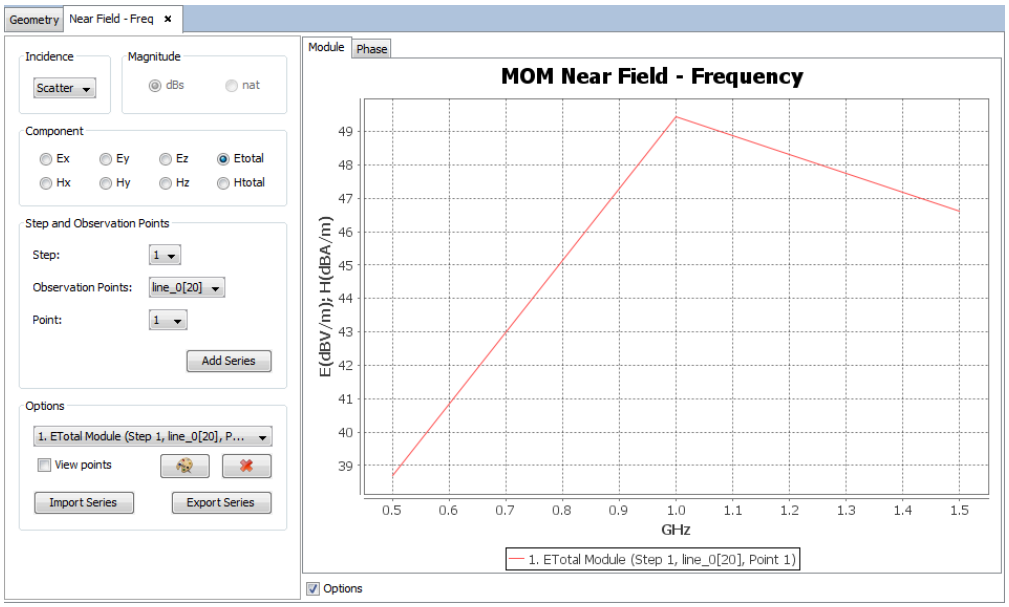


Figure 1239: Near Field Observation Points by Frequency cuts

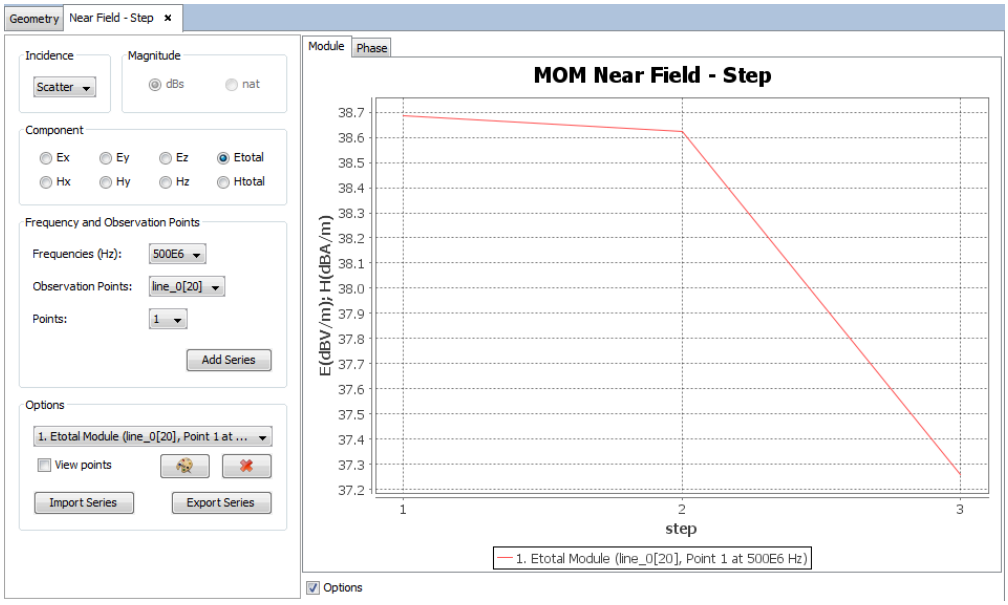


Figure 1240: Near Field Observation Points by Step cuts

Select 'Show Results --> Coupling' option to show the value of the coupling between antennas.

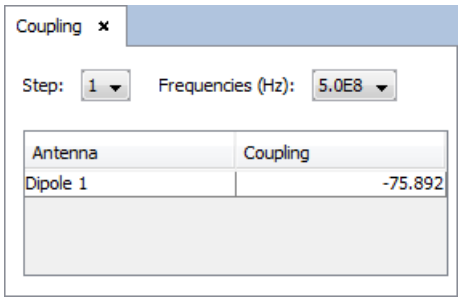


Figure 1241: Coupling values

Selecting other values step or frequency parameters new table of values for the coupling between antennas and passive antenna will be shown.

On 'Show Results --> Coupling' menu, other results are present such as 'View Coupling by Step' and 'View Coupling by Frequency' and this option display the values for one selected point for each step or frequency.

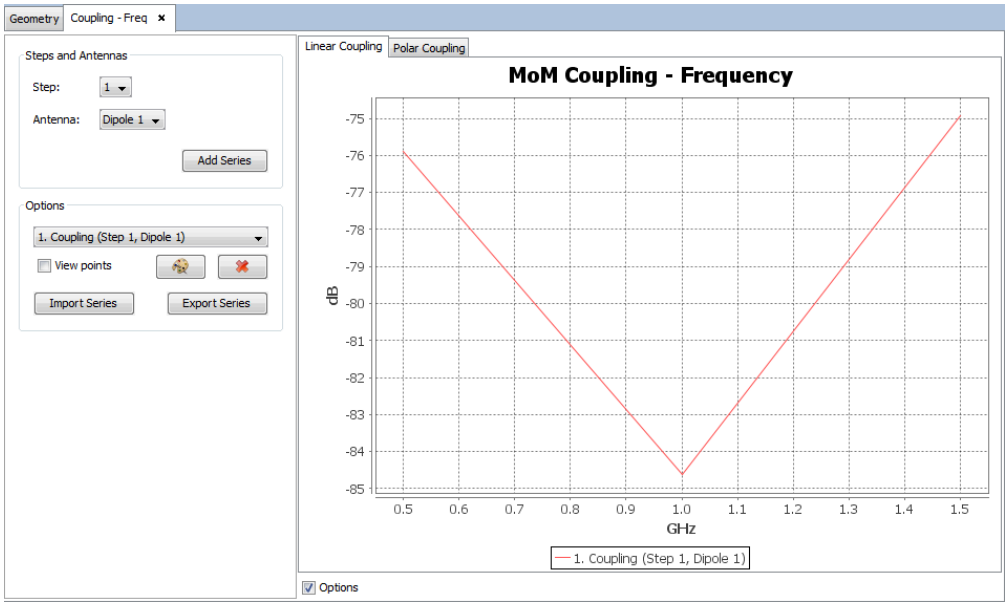


Figure 1242: Coupling by Frequency values

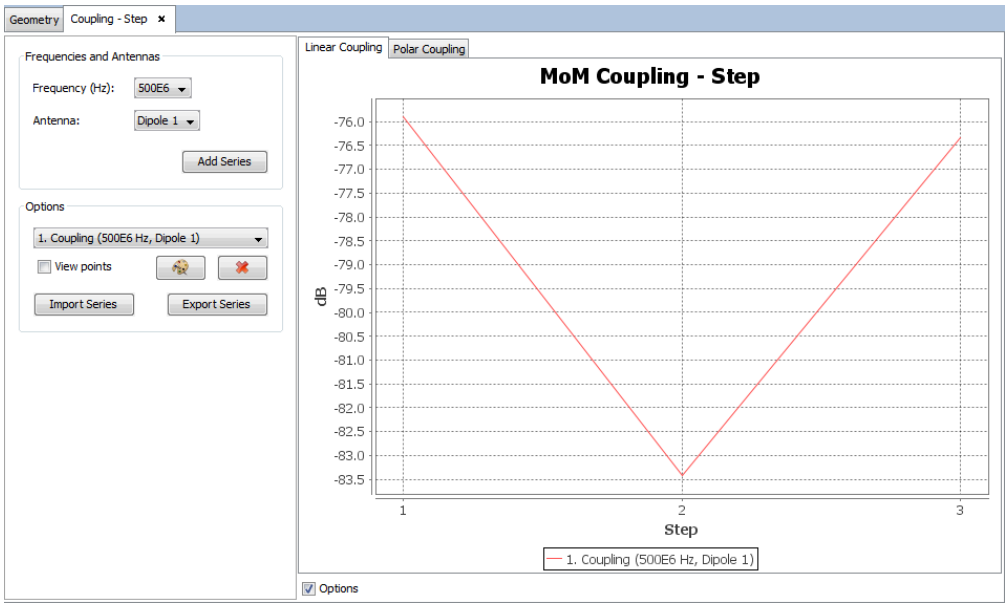


Figure 1243: Coupling by Step values

11.15.3 Example 3: Radiation Map Simulation with Frequency Sweep

This case explains how to calculate far-field, radiation pattern, current density, charge density, and radiation map of an open box with a dipole antenna insides the box and a frequency sweep from 0.5 GHz to 1.5 GHz with 3 examples.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

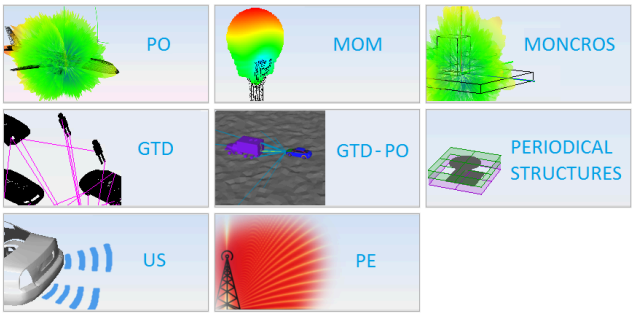


Figure 1244: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To obtain more information about geometries generation see Parameters.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

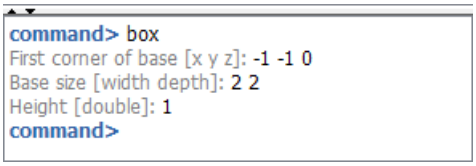


Figure 1245: Box parameters

Select box geometry on the main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

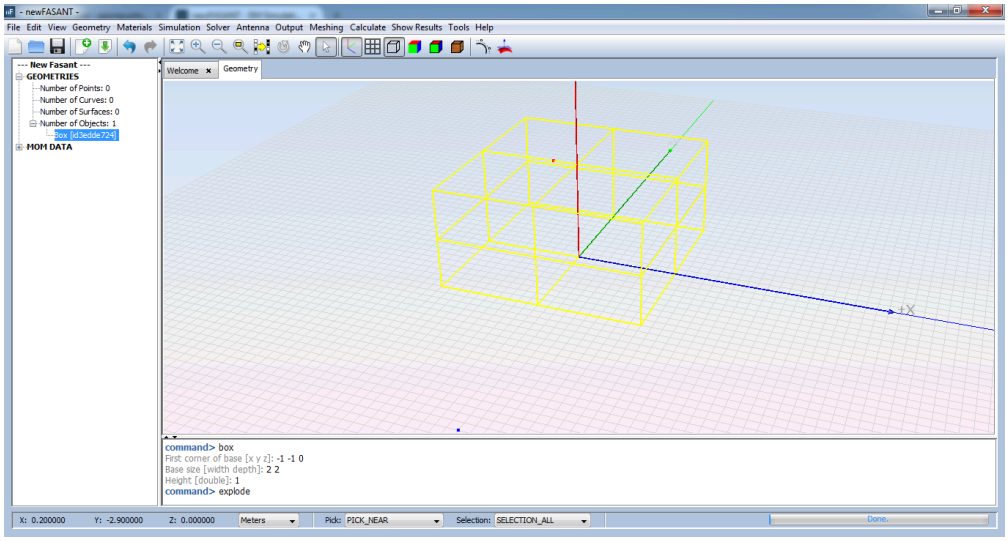


Figure 1246: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

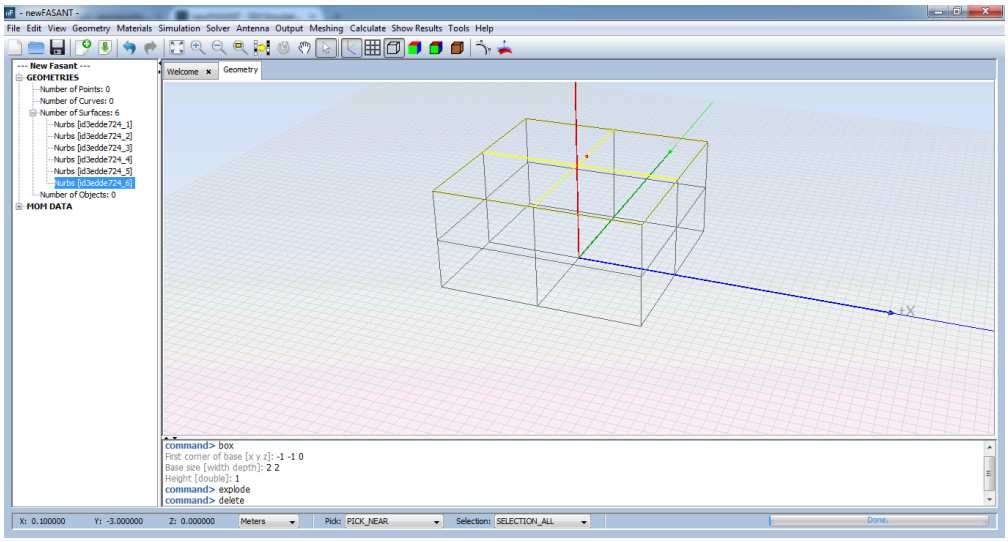


Figure 1247: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

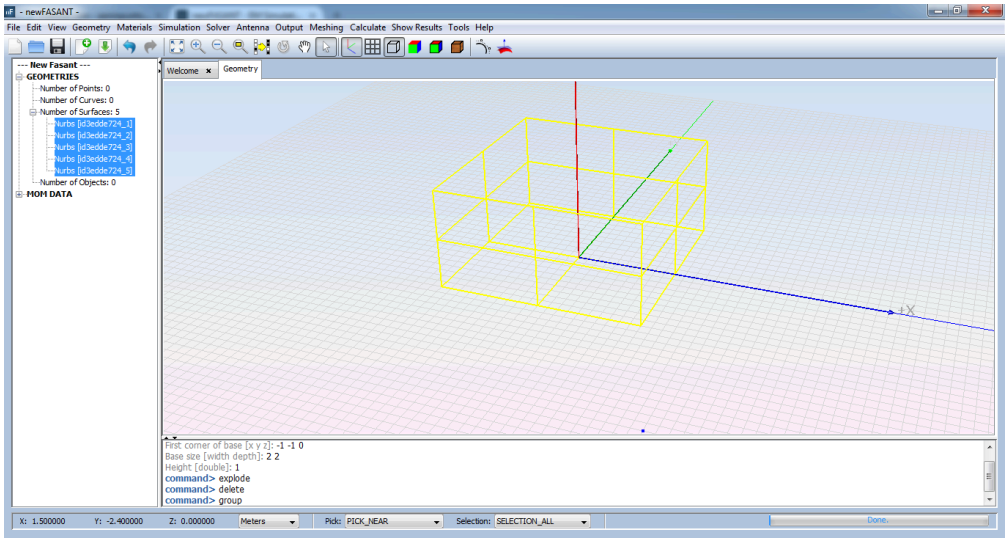


Figure 1248: Surfaces selection and 'group' command

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

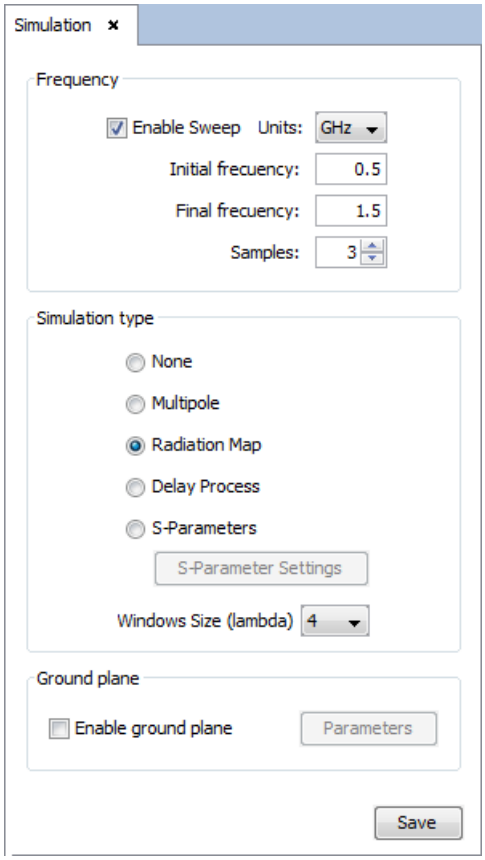


Figure 1249: Simulation Parameters panel

Step 4: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

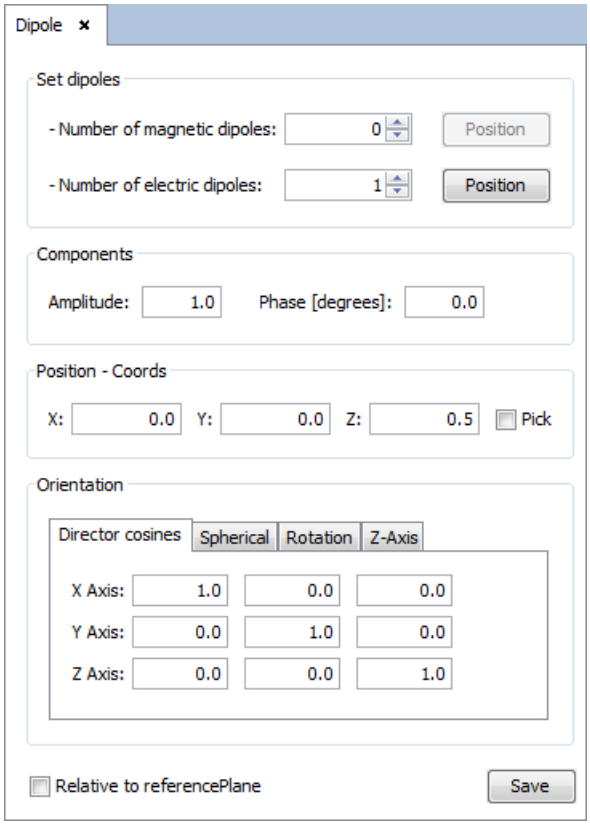


Figure 1250: Dipole Antenna panel

Step 5: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

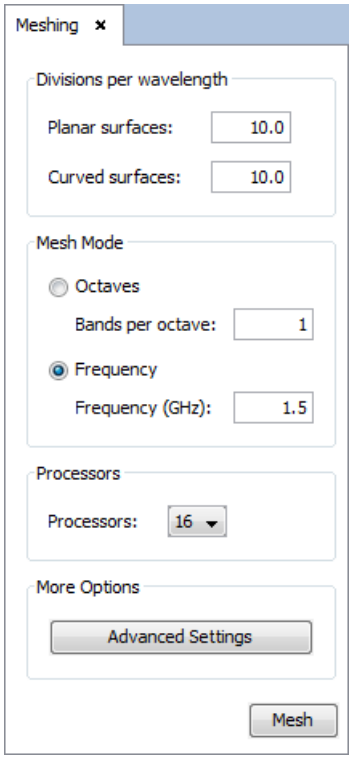


Figure 1251: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

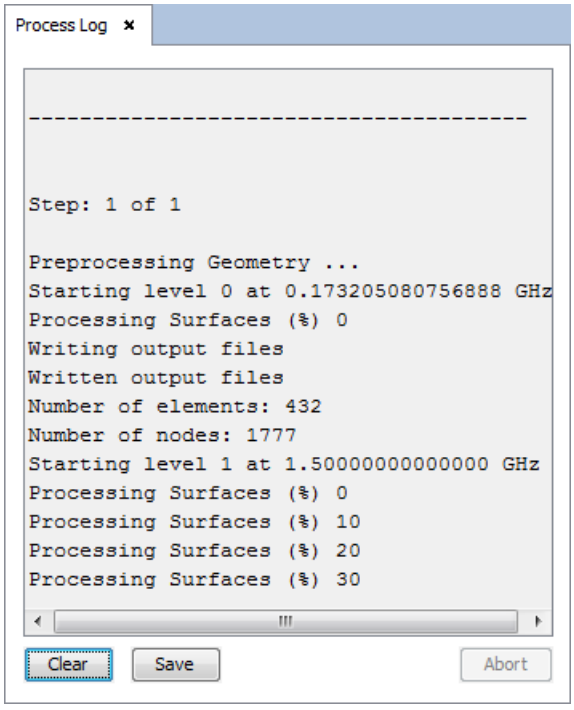


Figure 1252: Meshing process log

Step 6: Execute the simulation. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

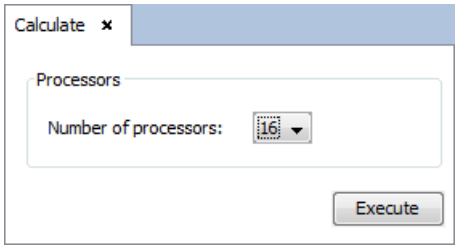


Figure 1253: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

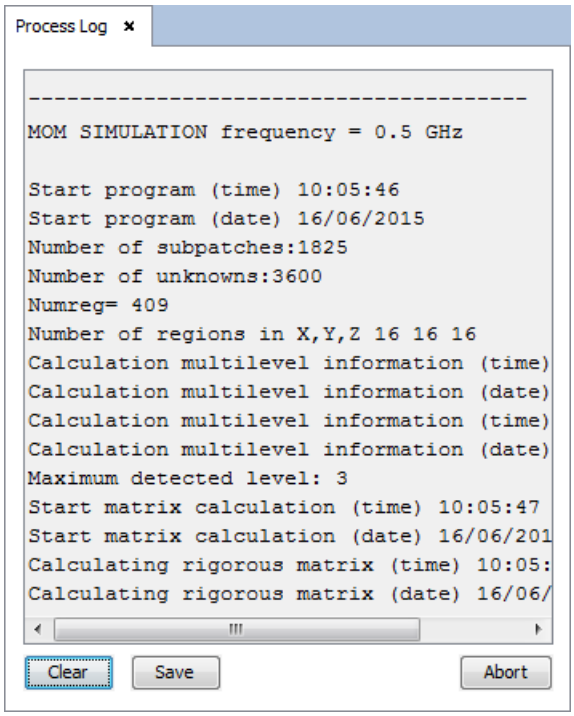


Figure 1254: Execute process log

Step 7: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

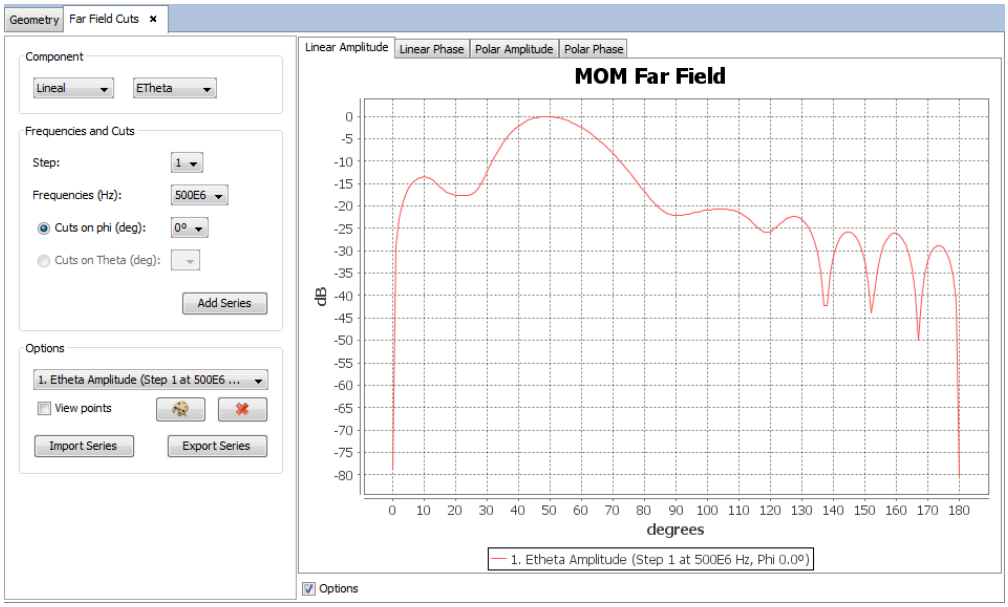


Figure 1255: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

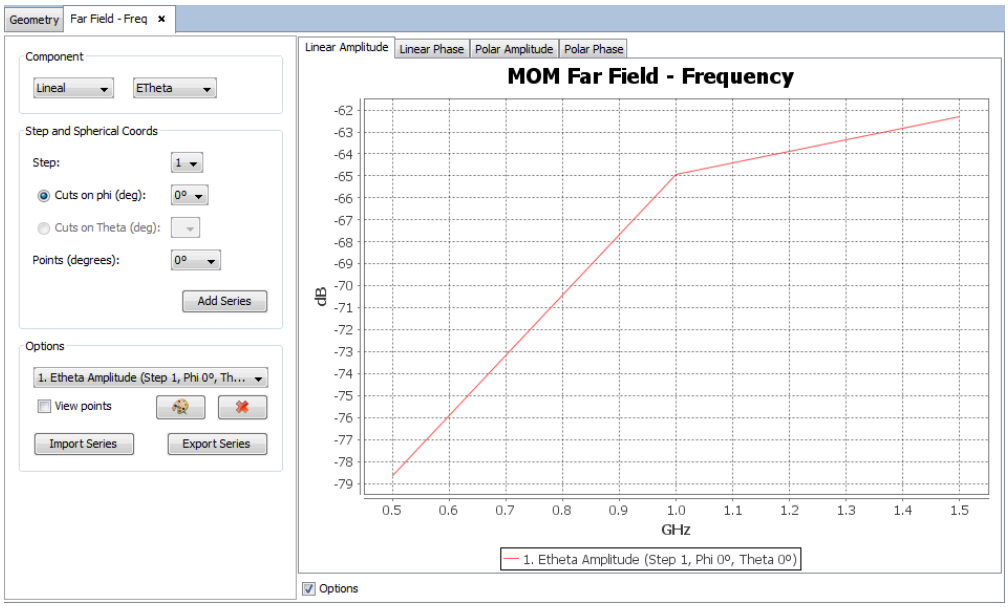


Figure 1256: Far Field by Frequency cuts

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

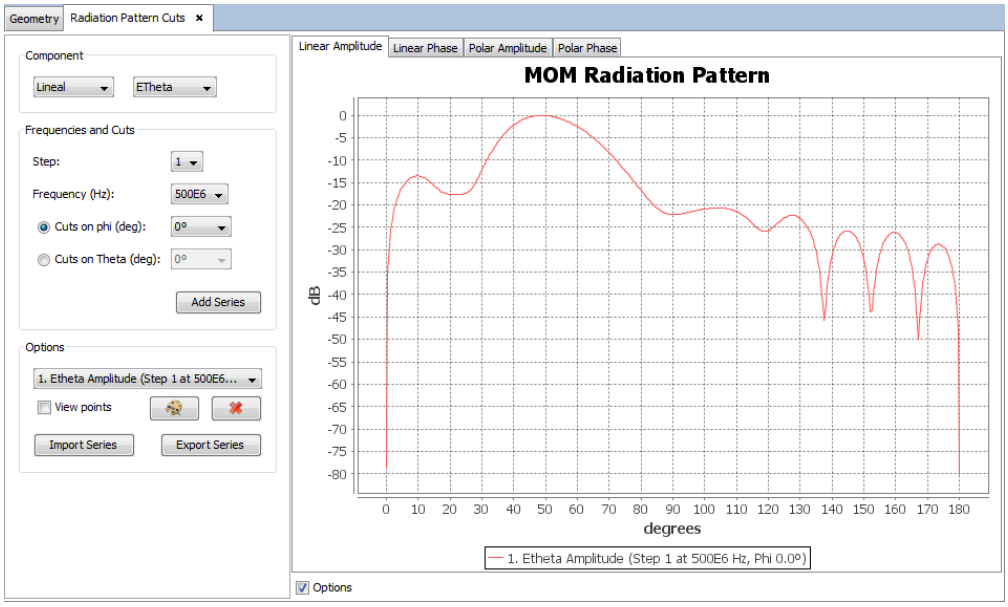


Figure 1257: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

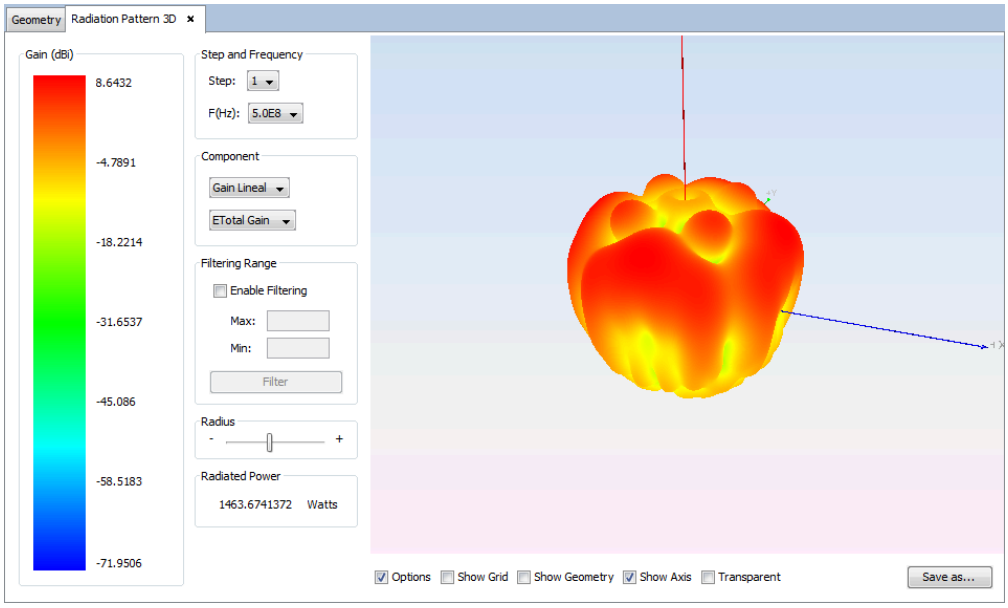


Figure 1258: Radiation Pattern 3D for frequency 0.5 GHz

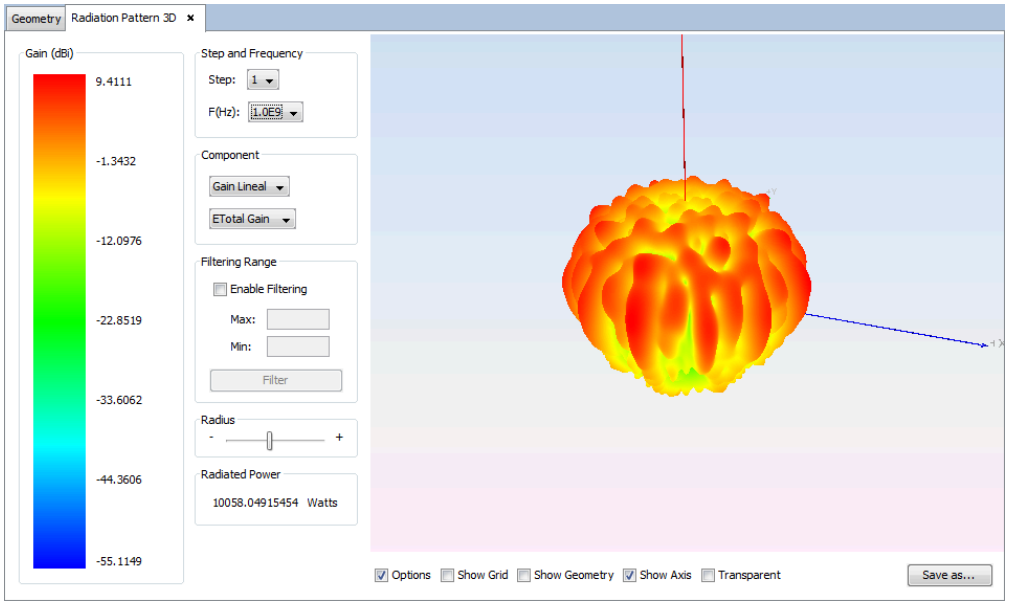


Figure 1259: Radiation Pattern 3D for frequency 1.0 GHz

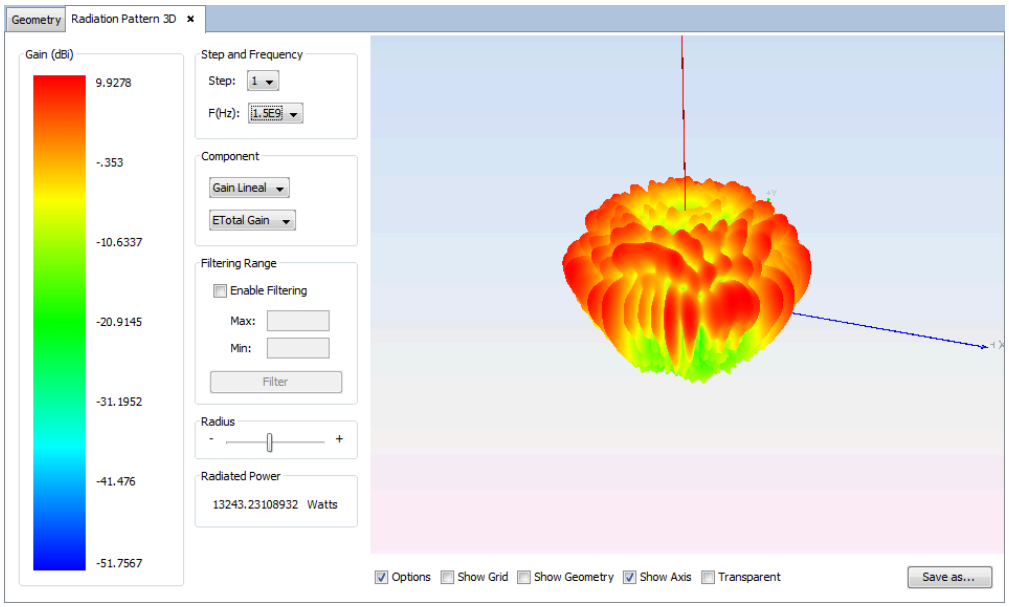


Figure 1260: Radiation Pattern 3D for frequency 1.5 GHz

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

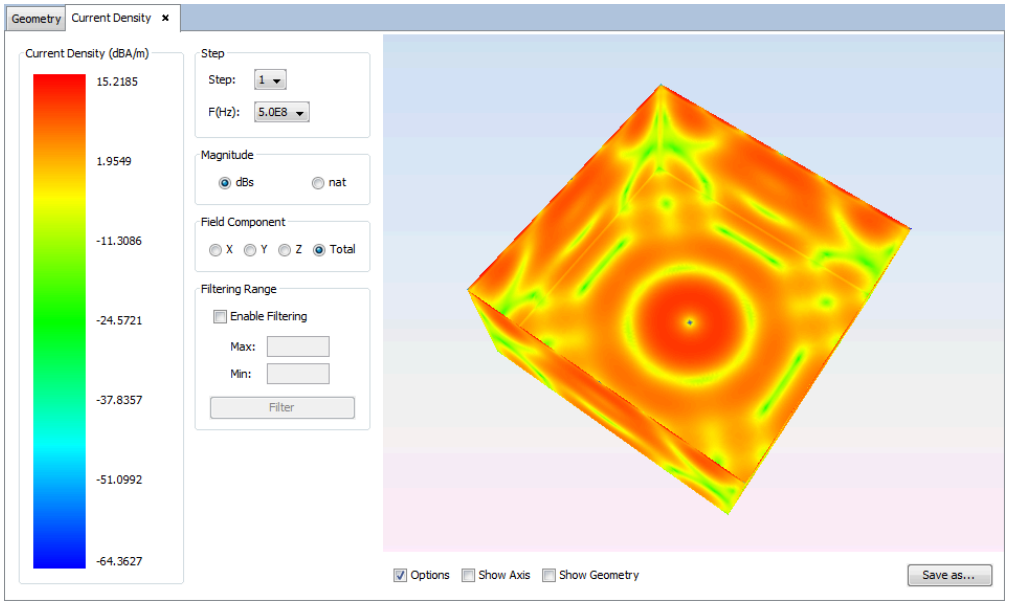


Figure 1261: Current Density for frequency 0.5 GHz

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

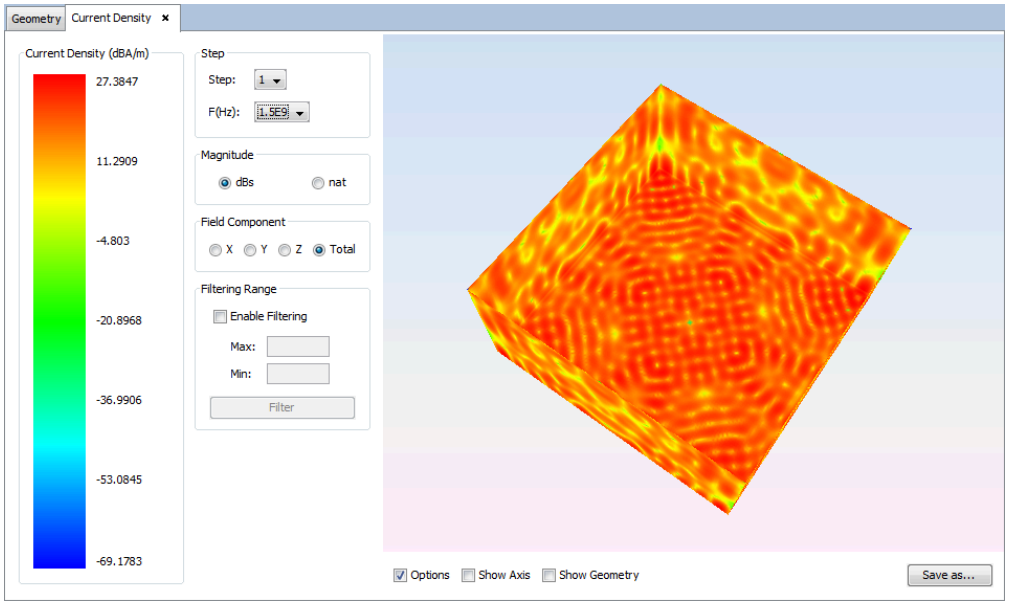


Figure 1262: Current Density for frequency 1.5 GHz

Select 'Show Results --> View Charges' option to show the charge density.

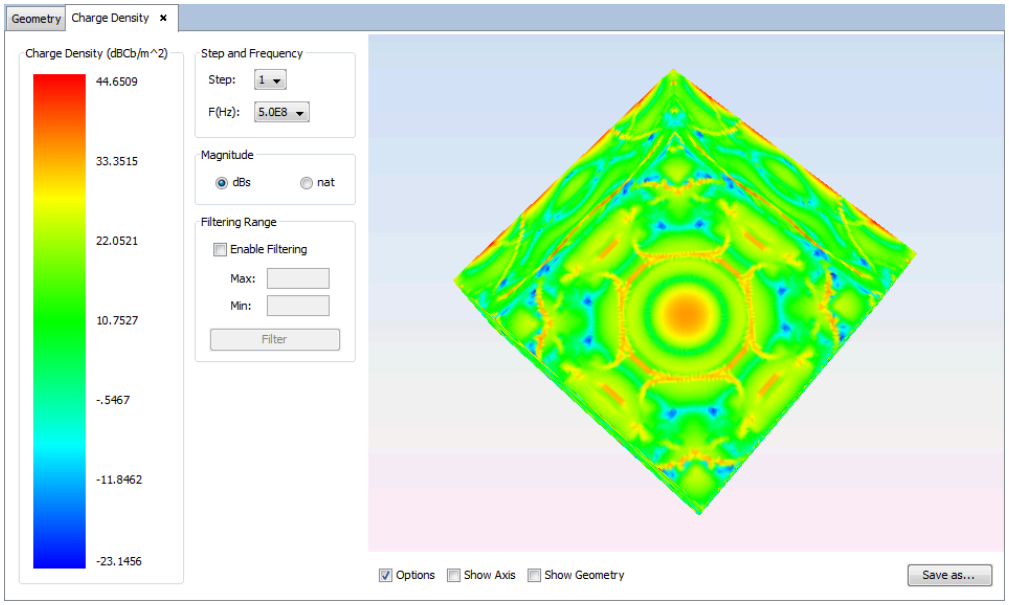


Figure 1263: Charge Density for frequency 0.5 GHz

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

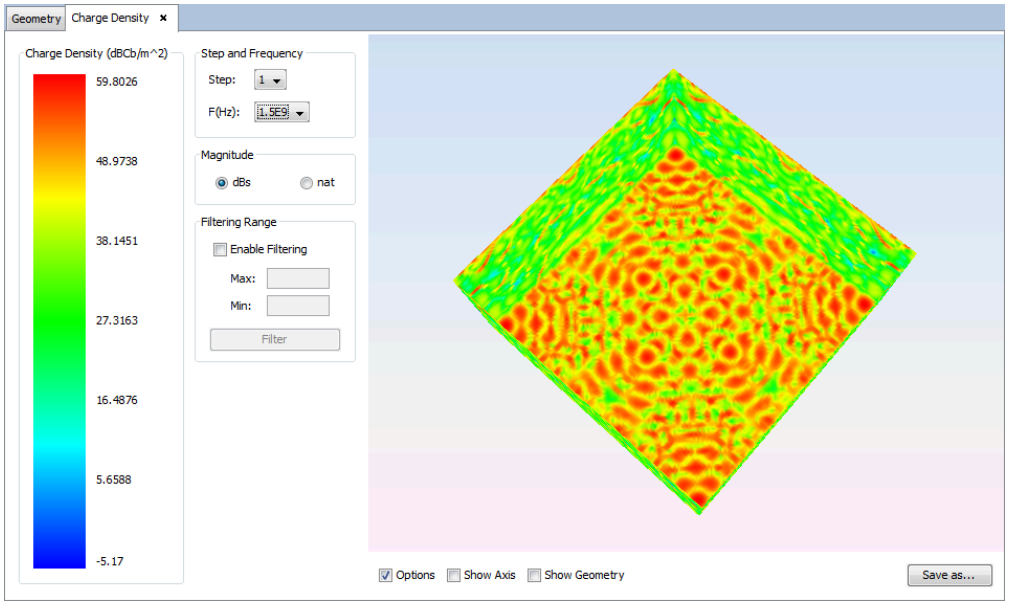


Figure 1264: Charge Density for frequency 1.5 GHz

Select 'Show Results --> View Radiation Map' option to show the radiation map.

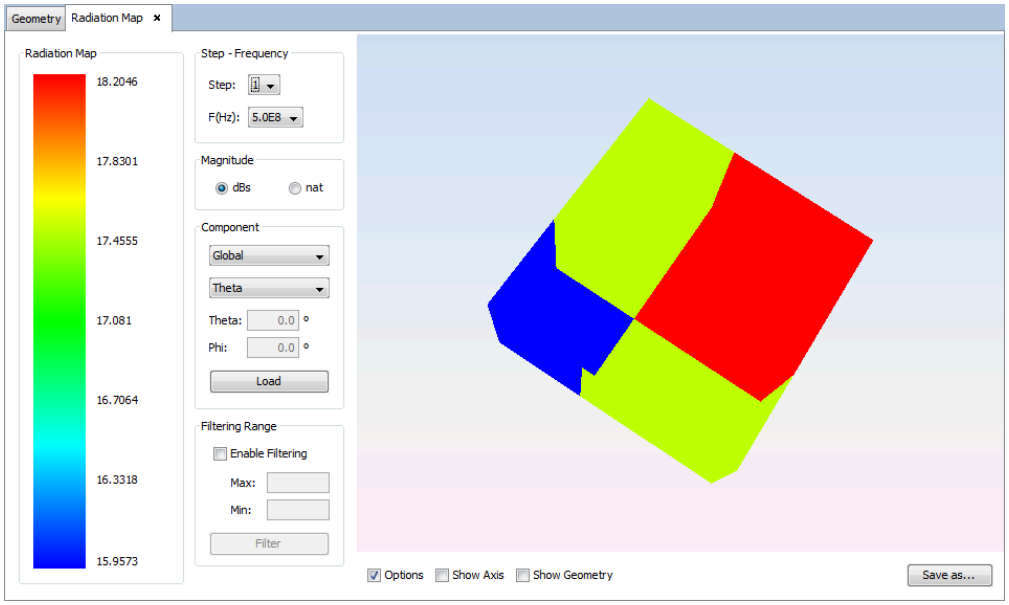


Figure 1265: Radiation Map for frequency 0.5 GHz

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

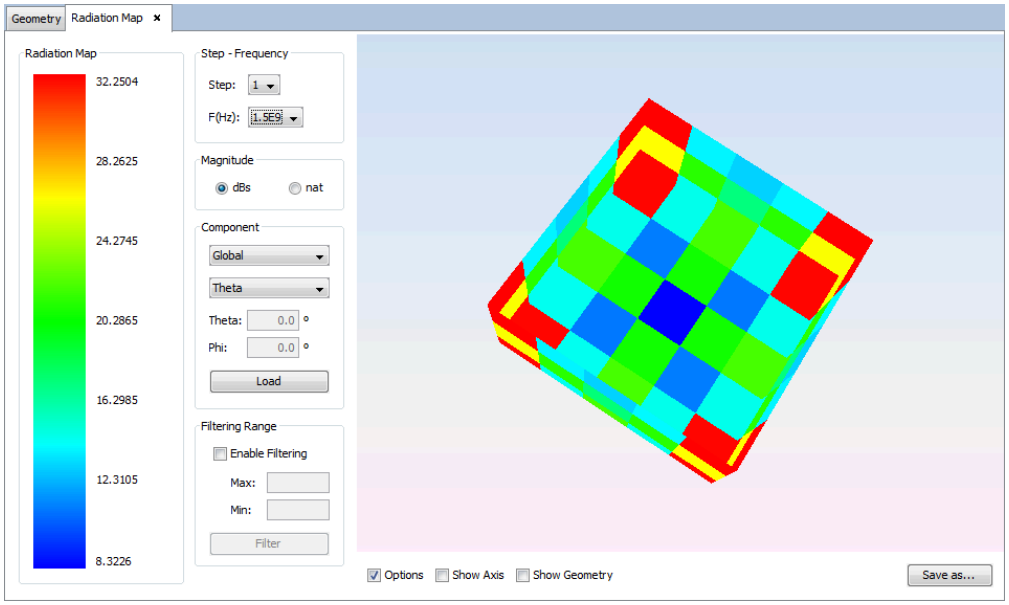


Figure 1266: Radiation Map for frequency 1.5 GHz

11.15.4 Example 4: Delay Process Simulation with Frequency Sweep

This case explains how to calculate far field, radiation pattern, current density, charge density and delay process of a open box with a dipole antenna insides the box and a frequency sweep from 0.5 GHz to 1.5 GHz with 3 examples.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

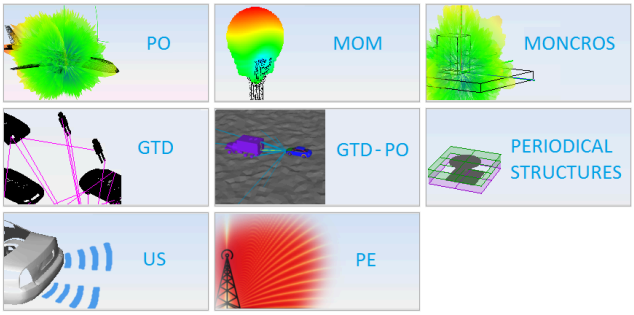


Figure 1267: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To obtain more information about geometries generation see Parameters.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

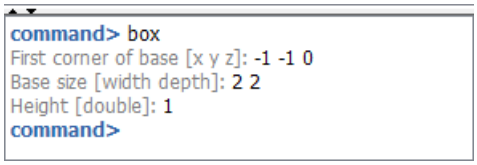


Figure 1268: Box parameters

Select box geometry on the main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

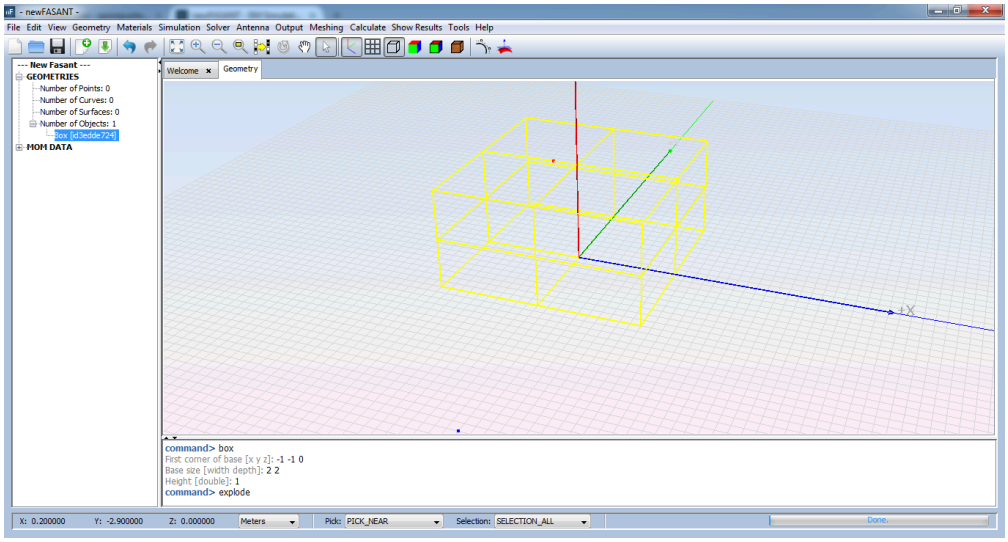


Figure 1269: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

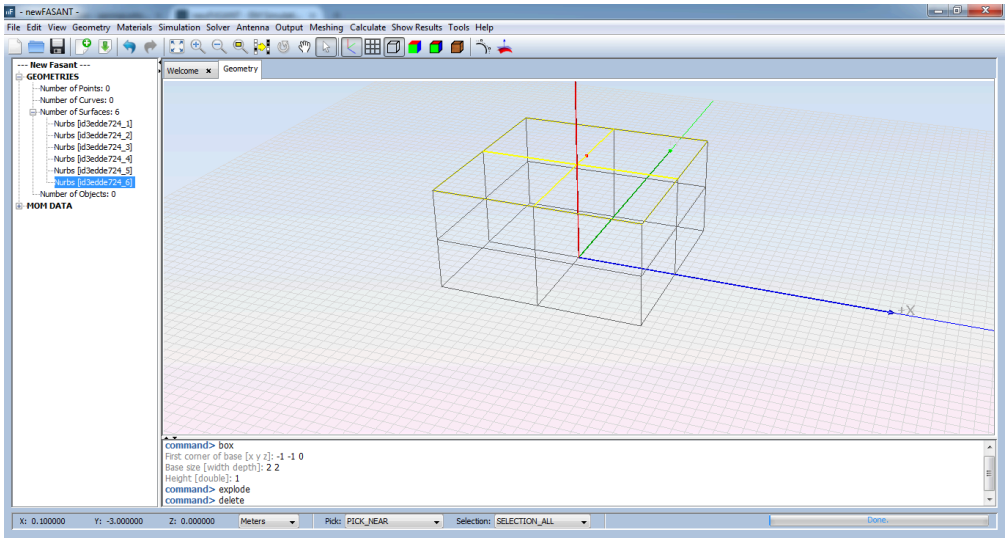


Figure 1270: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

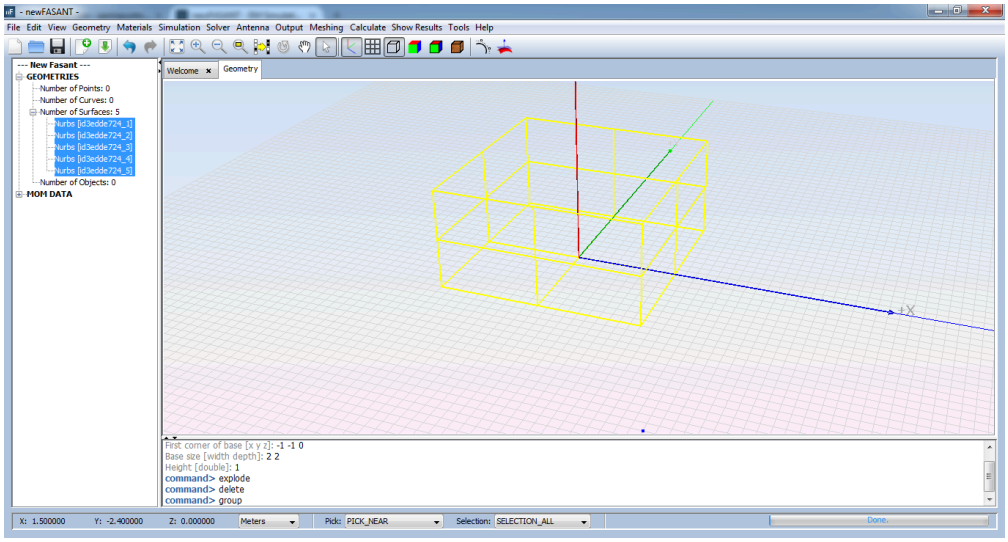


Figure 1271: Surfaces selection and 'group' command

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

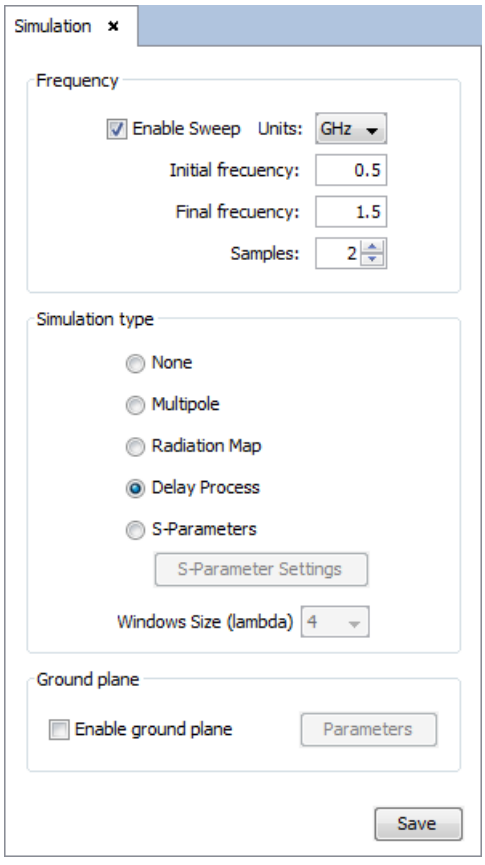


Figure 1272: Simulation Parameters panel

Step 4: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

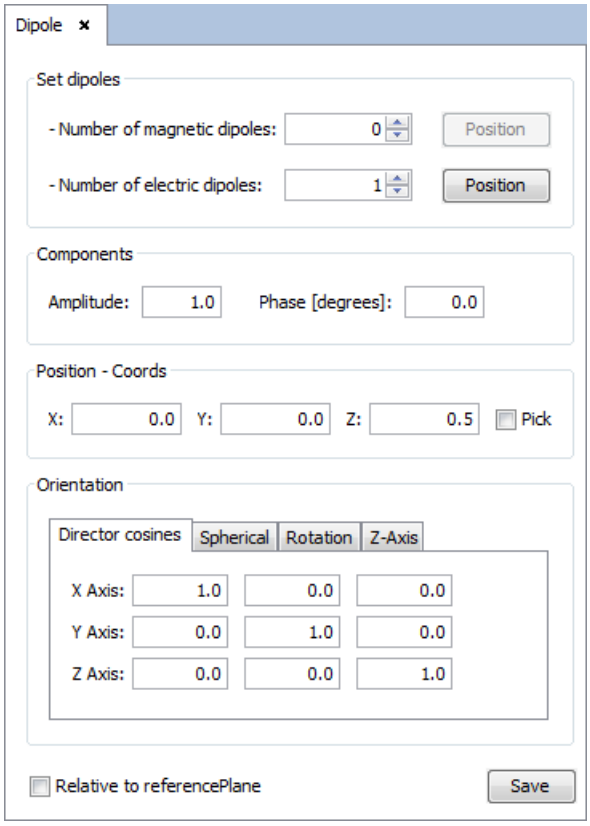


Figure 1273: Dipole Antenna panel

Step 5: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

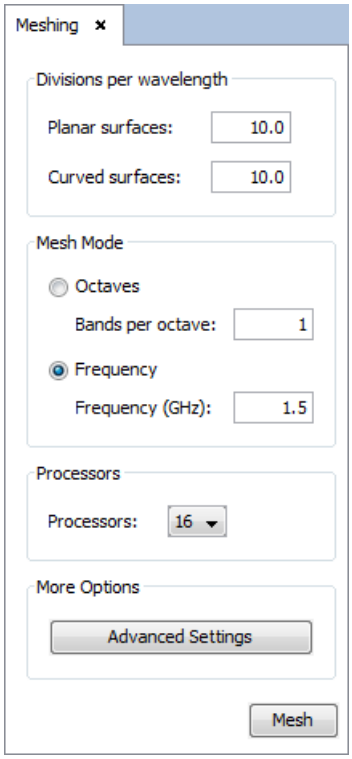


Figure 1274: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

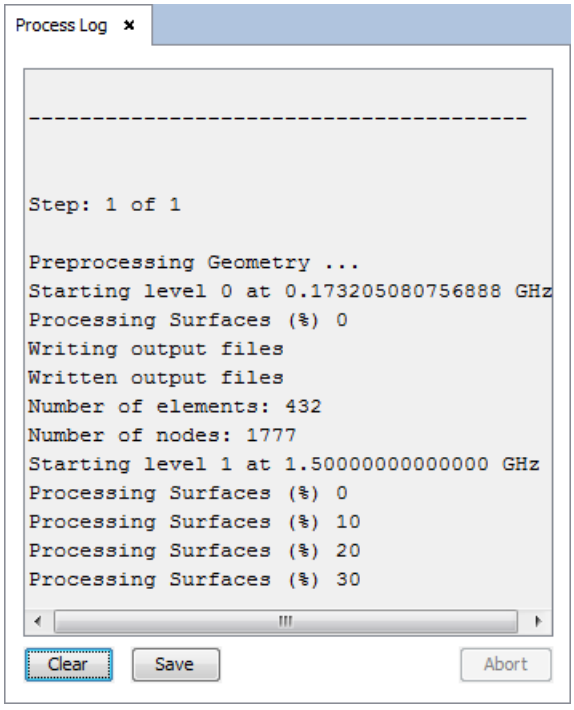


Figure 1275: Meshing process log

Step 6: Execute the simulation. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

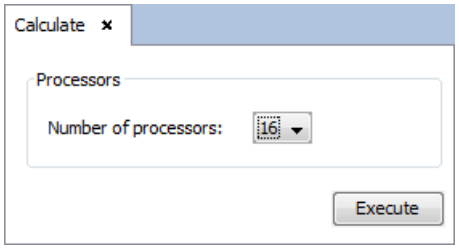


Figure 1276: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

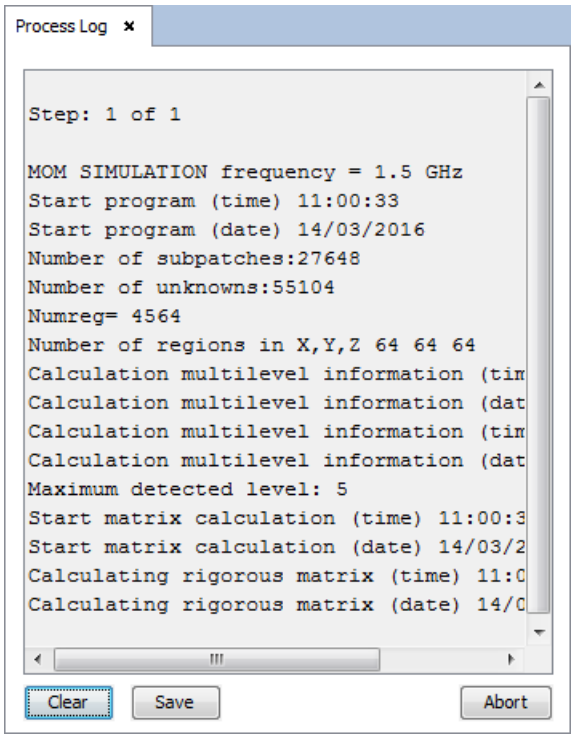


Figure 1277: Execute process log

Step 7: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

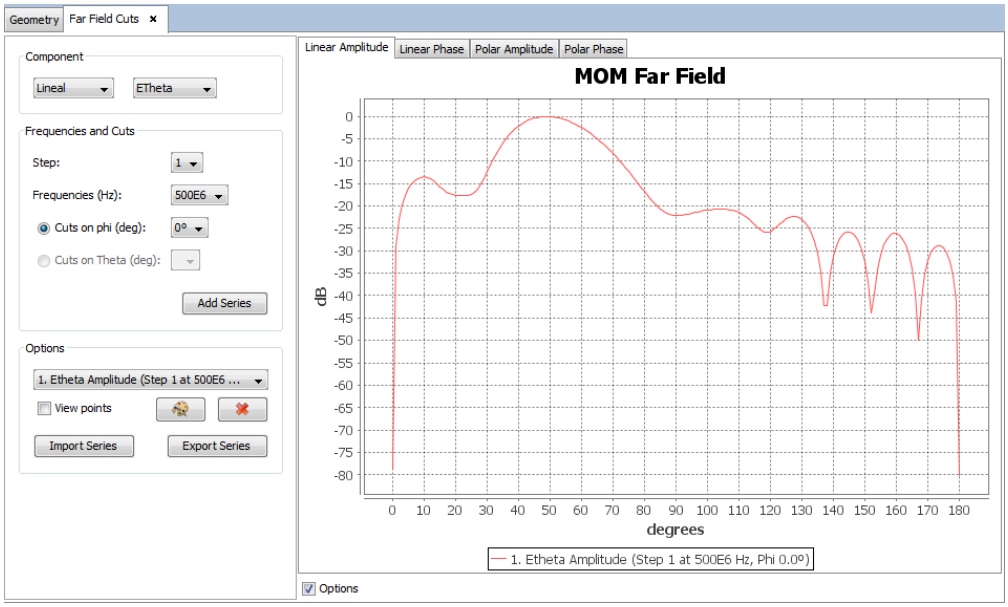


Figure 1278: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

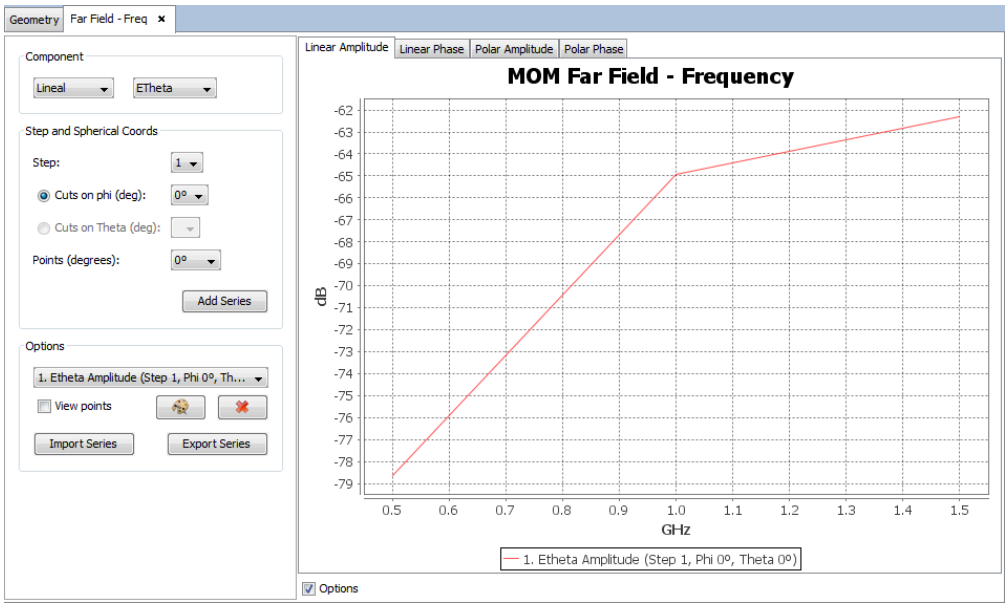


Figure 1279: Far Field by Frequency cuts

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

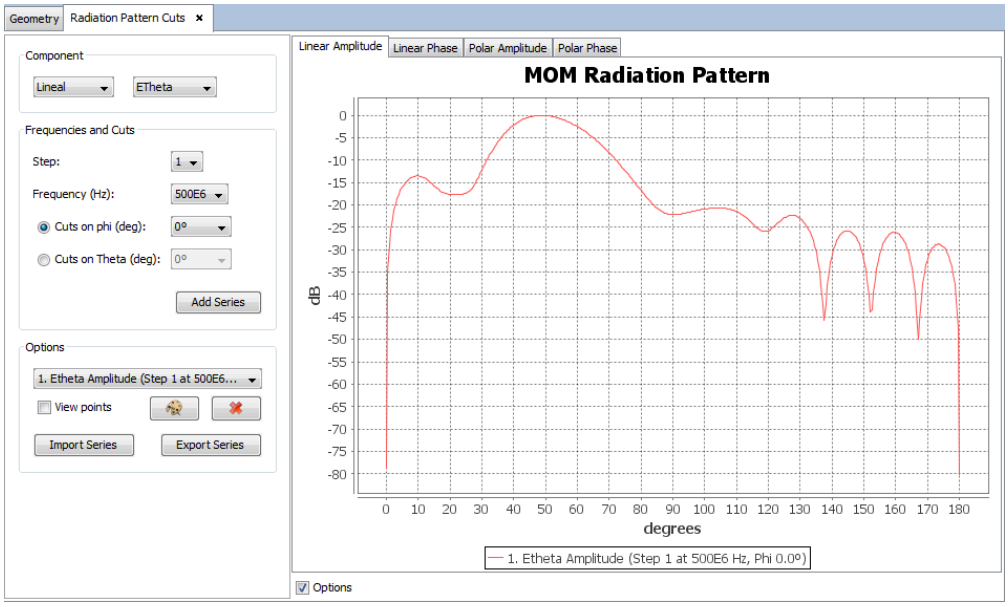


Figure 1280: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

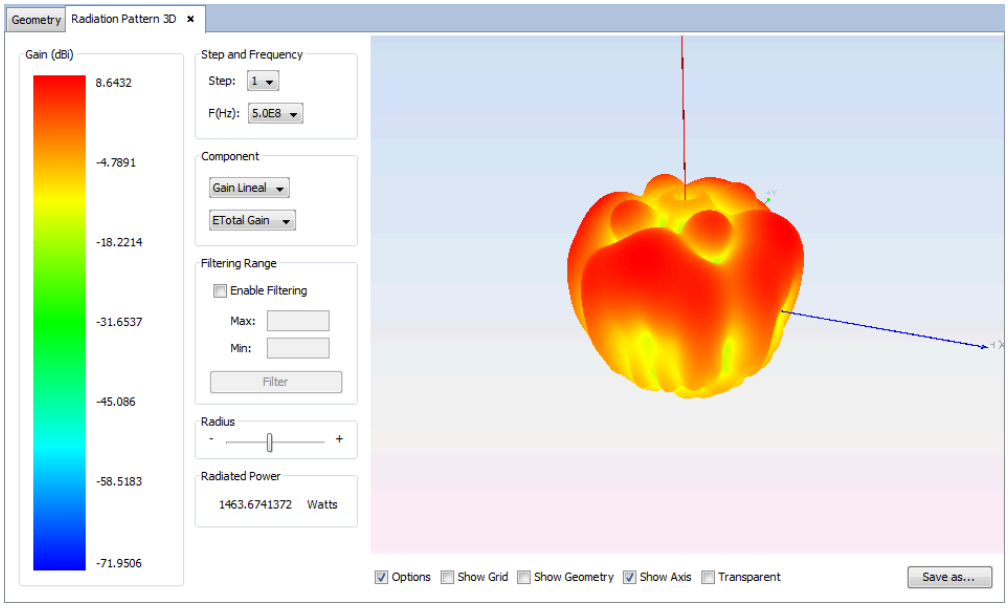


Figure 1281: Radiation Pattern 3D for frequency 0.5 GHz

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

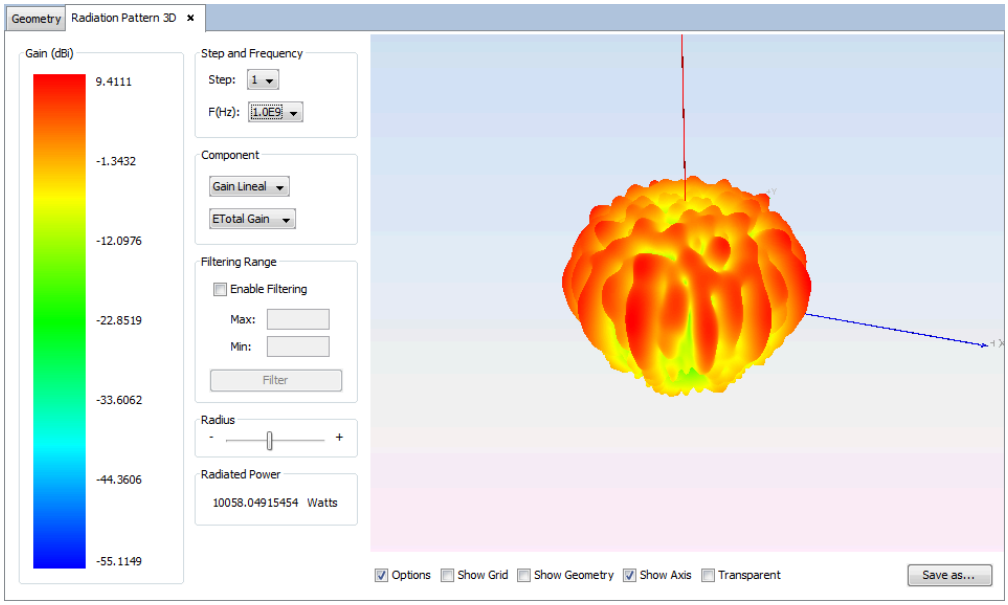


Figure 1282: Radiation Pattern 3D for frequency 1.0 GHz

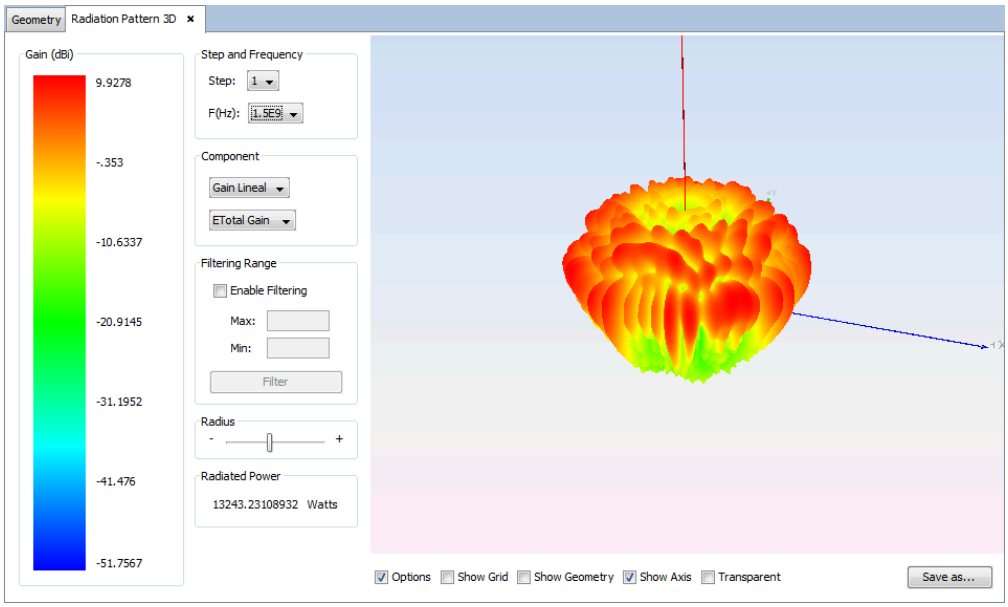


Figure 1283: Radiation Pattern 3D for frequency 1.5 GHz

Select 'Show Results --> View Currents' option to show the current density.

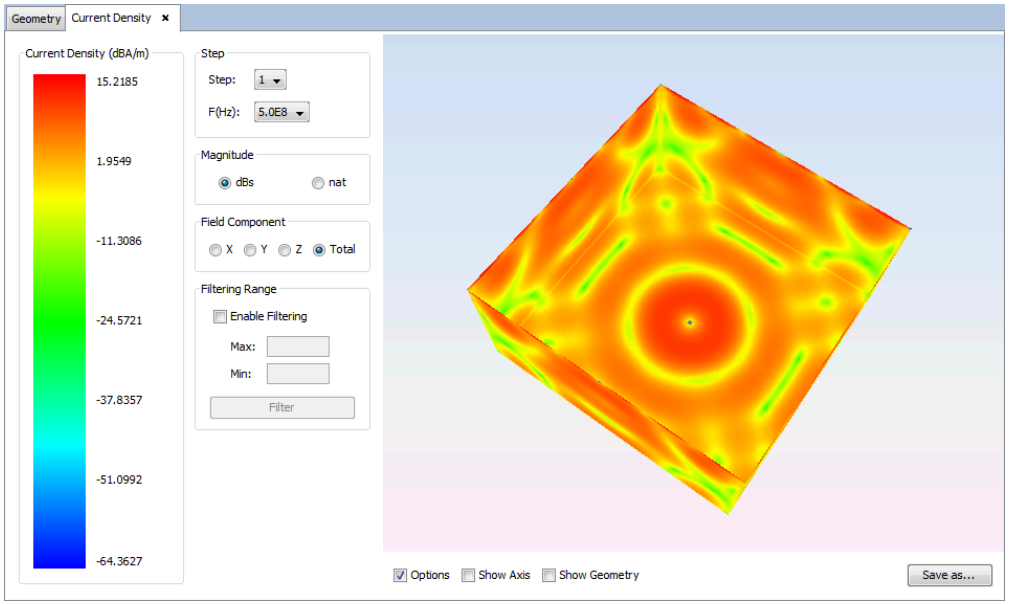


Figure 1284: Current Density for frequency 0.5 GHz

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

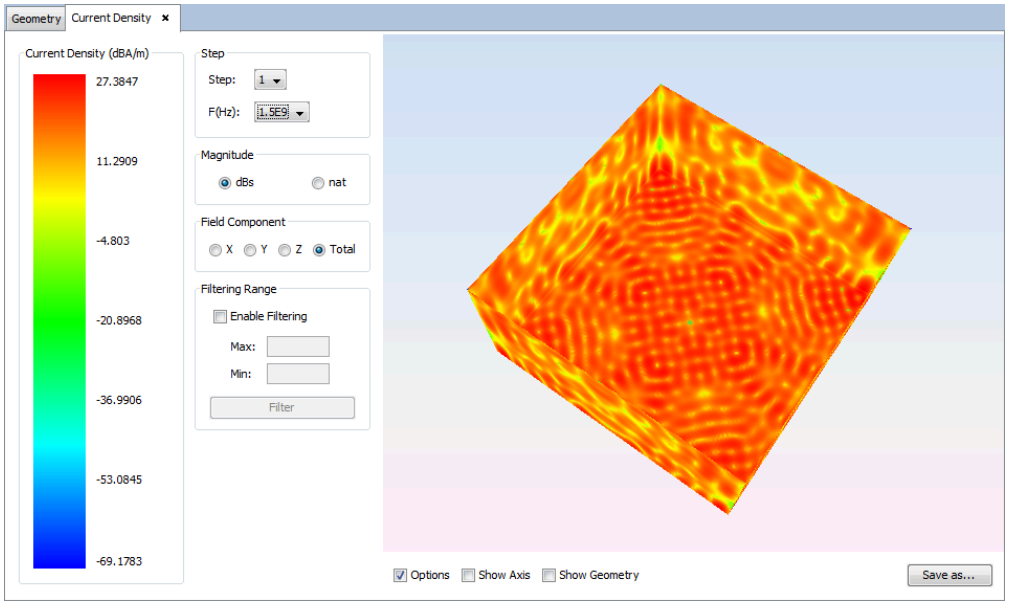


Figure 1285: Current Density for frequency 1.5 GHz

Select 'Show Results --> View Charges' option to show the charge density.

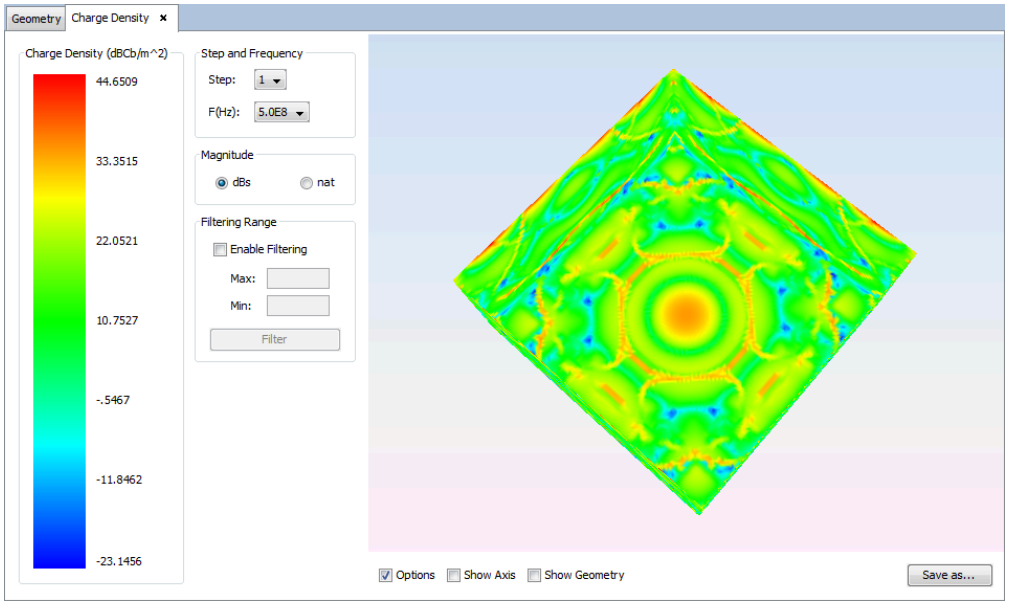


Figure 1286: Charge Density for frequency 0.5 GHz

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

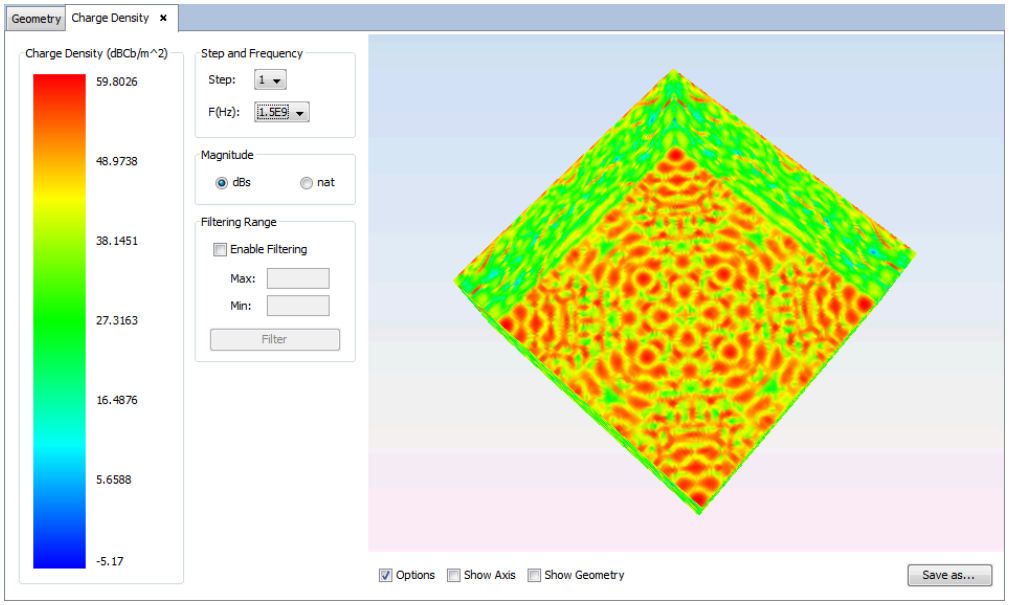


Figure 1287: Charge Density for frequency 0.5 GHz

Select 'Show Results --> View Delay Process' option to show the time delay. Set the parameters as the next figure shows:

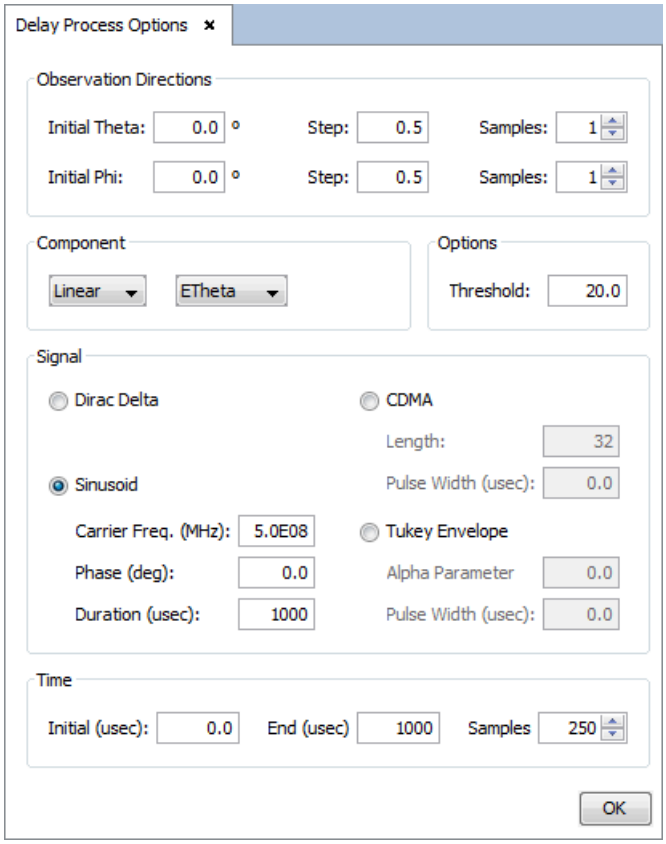


Figure 1288: Delay Process parameters panel

Then, click on 'OK' Button the delay process cut appears:

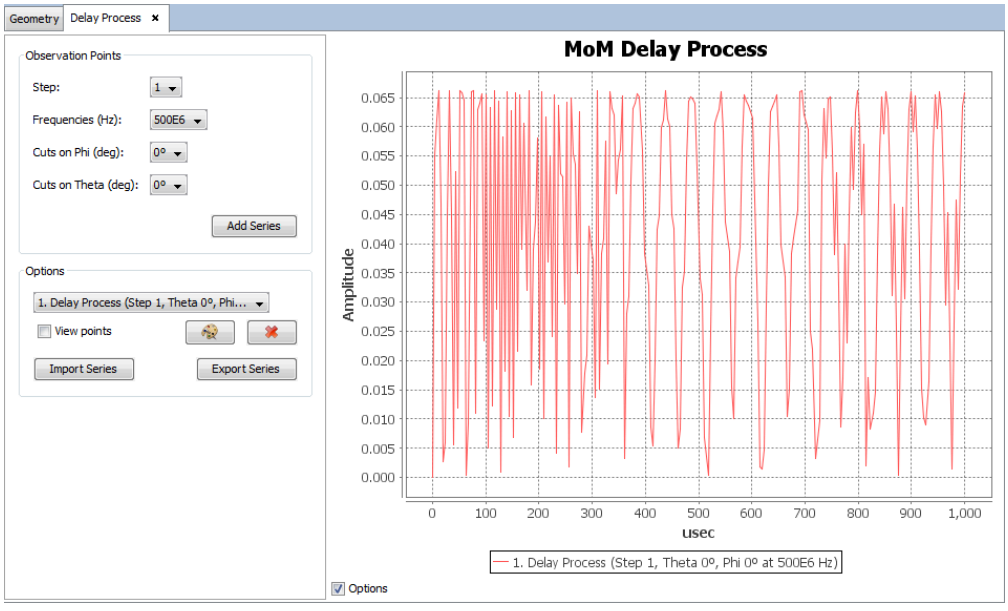


Figure 1289: Delay Process cuts

Changing values for step, frequency, theta cut or phi cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters.

11.15.5 Example 5: Multipole Examples

Multipole Simulation

This case explains how to calculate far-field, radiation pattern, current density, charge density and save a multipole file. Then use the multipole file to calculate the near field not calculated in the first simulation.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

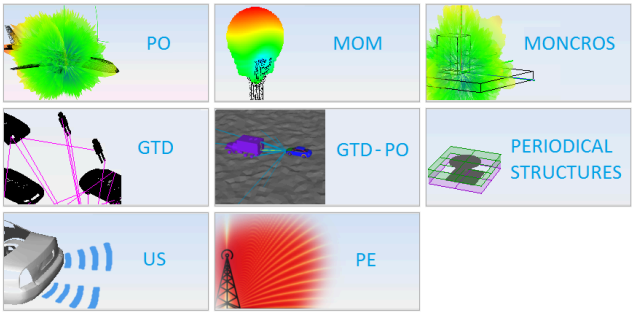


Figure 1290: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To obtain more information about geometries generation see Parameters.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

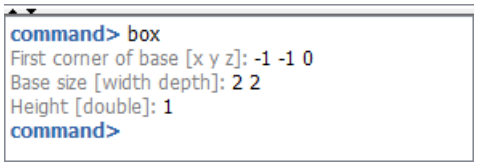


Figure 1291: Box parameters

Select box geometry on the main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

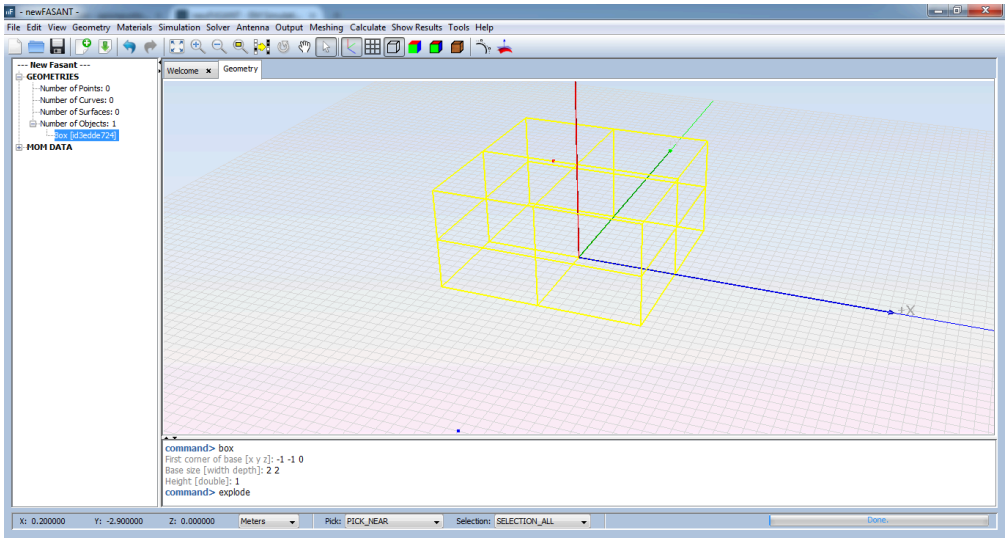


Figure 1292: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

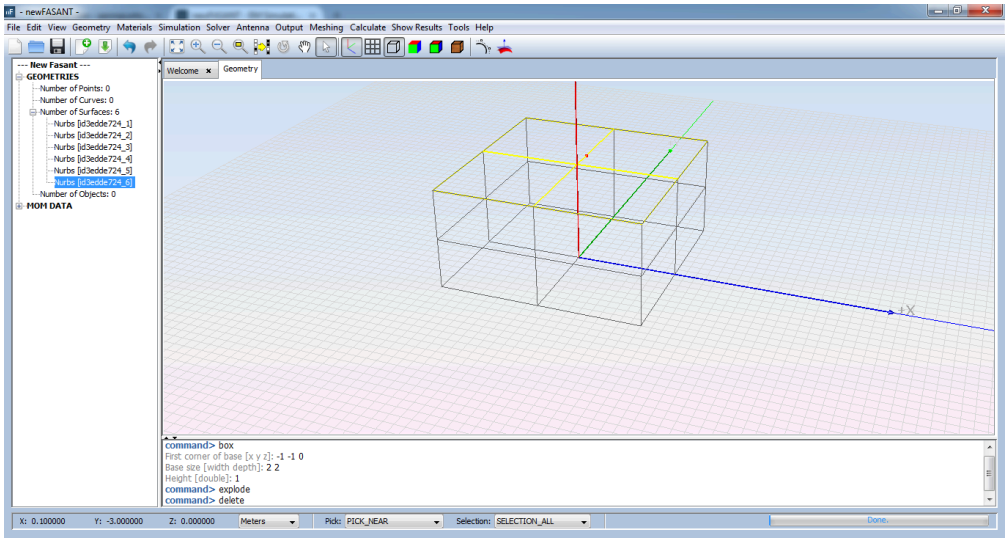


Figure 1293: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

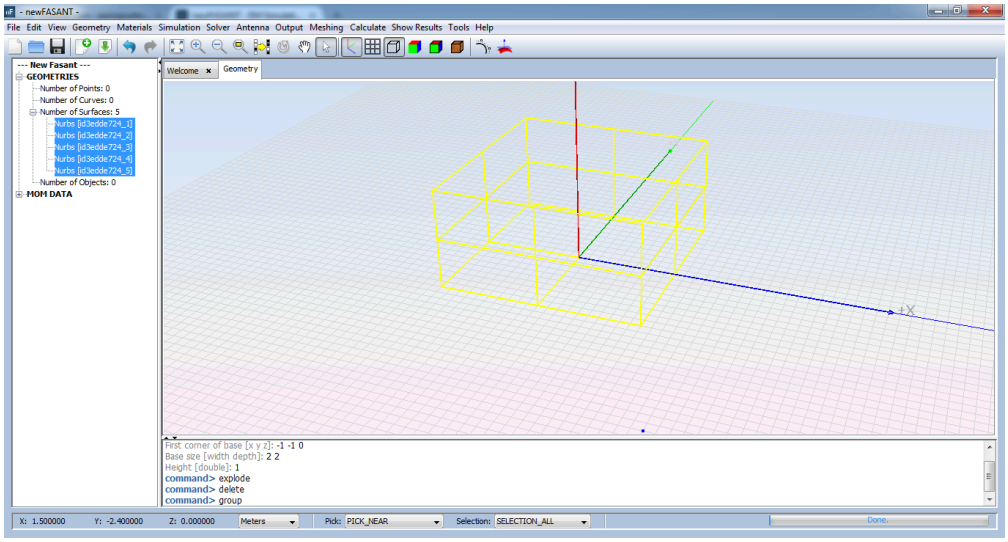


Figure 1294: Surfaces selection and 'group' command

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

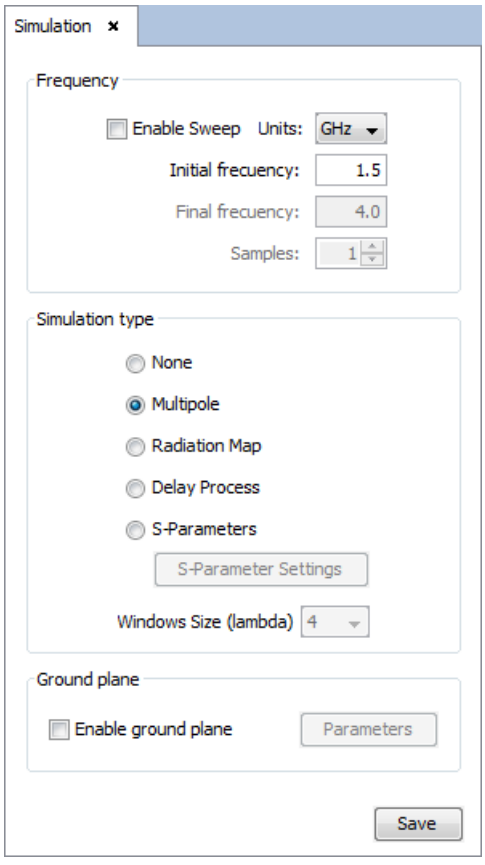


Figure 1295: Simulation Parameters panel

Step 4: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Antenna --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

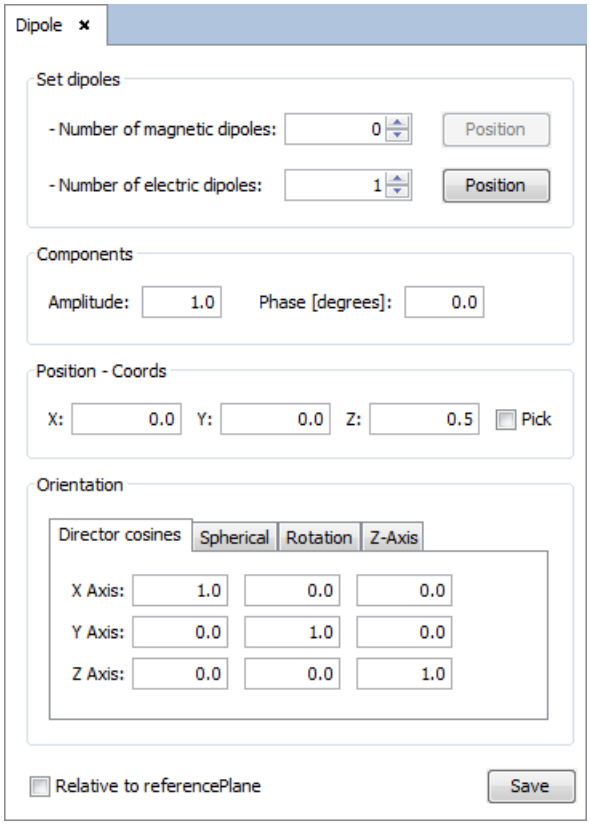


Figure 1296: Dipole Antenna panel

Step 5: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

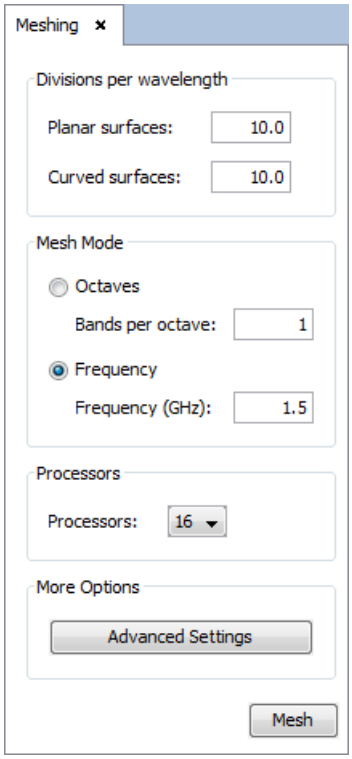


Figure 1297: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

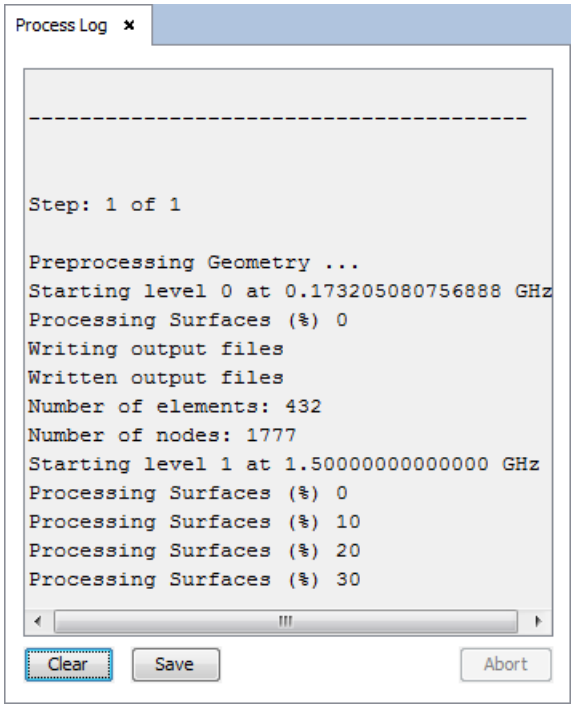


Figure 1298: Meshing process log

Step 6: Execute the simulation. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

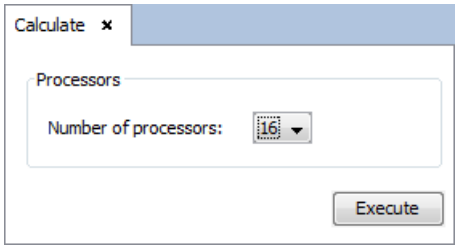


Figure 1299: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

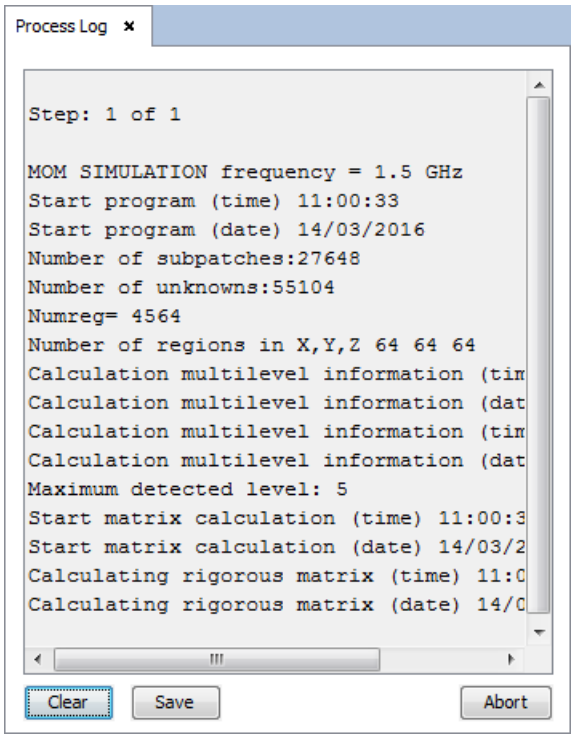


Figure 1300: Execute process log

Step 7: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

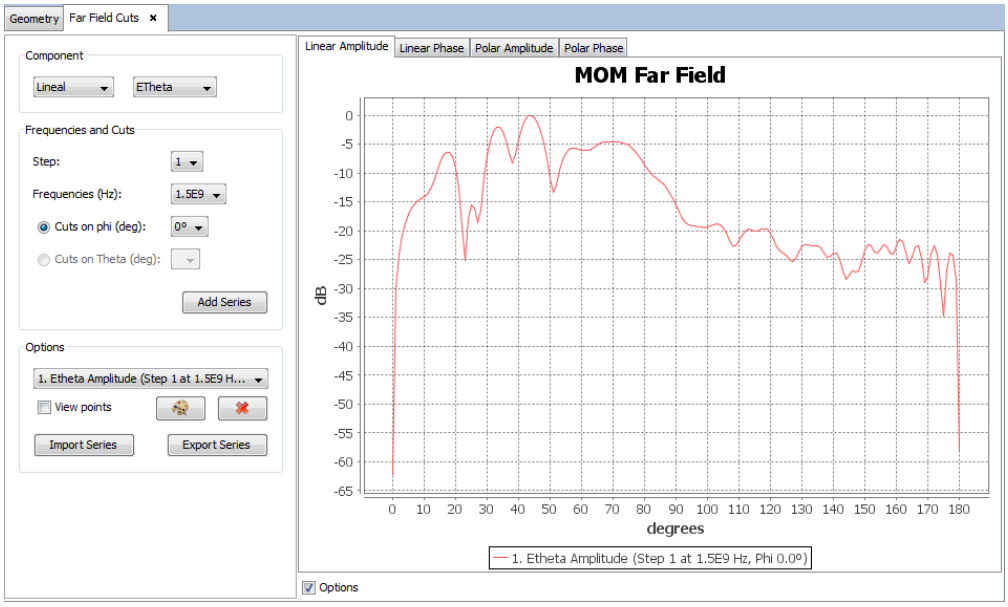


Figure 1301: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

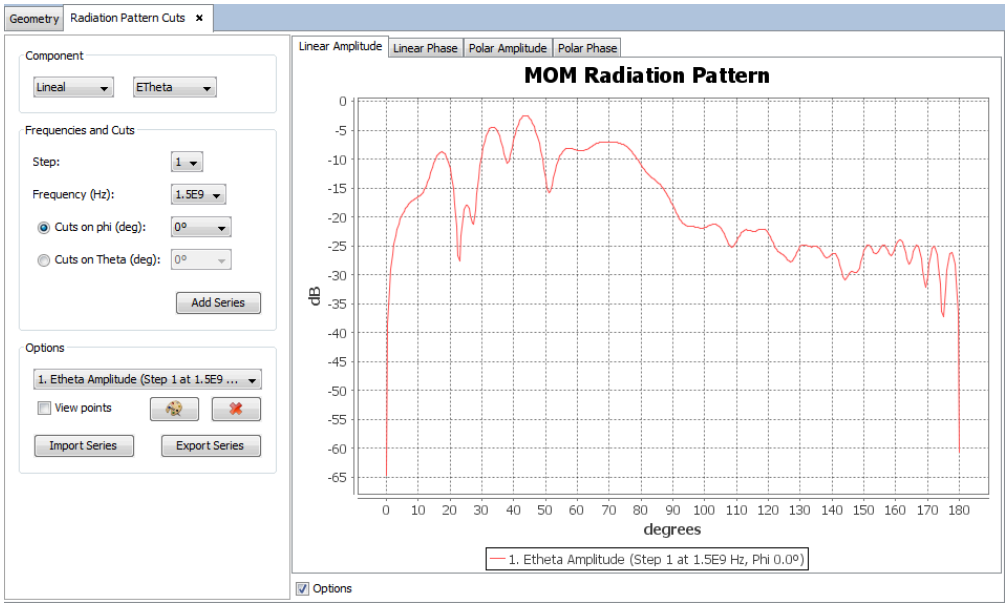


Figure 1302: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Radiation

Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

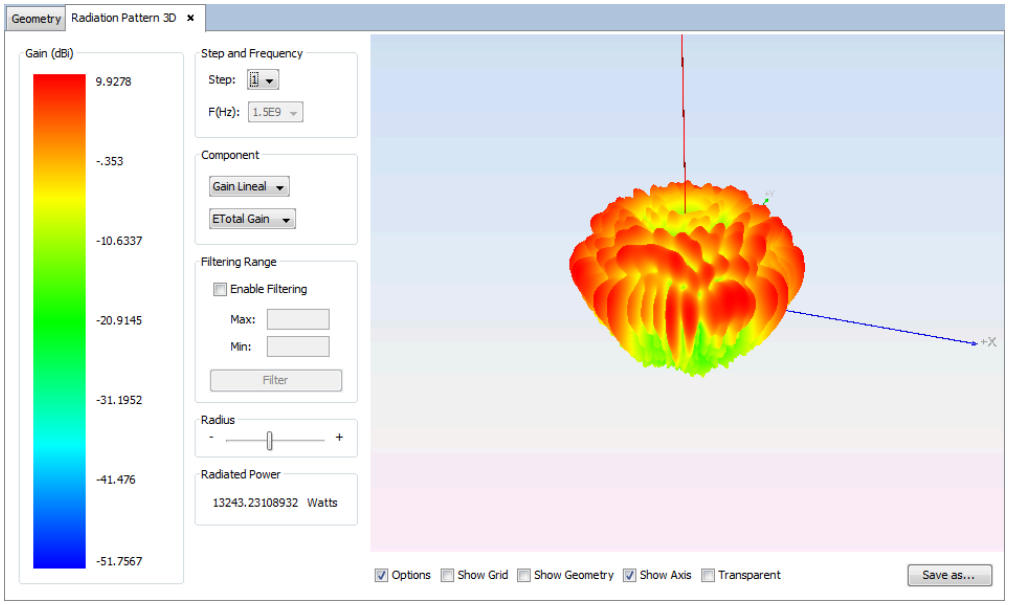


Figure 1303: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

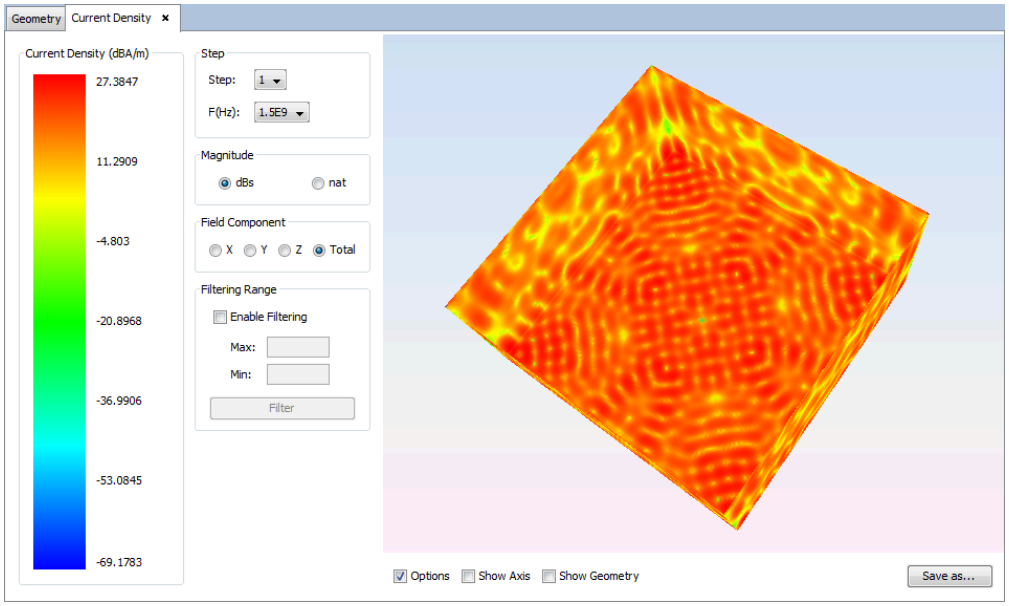


Figure 1304: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

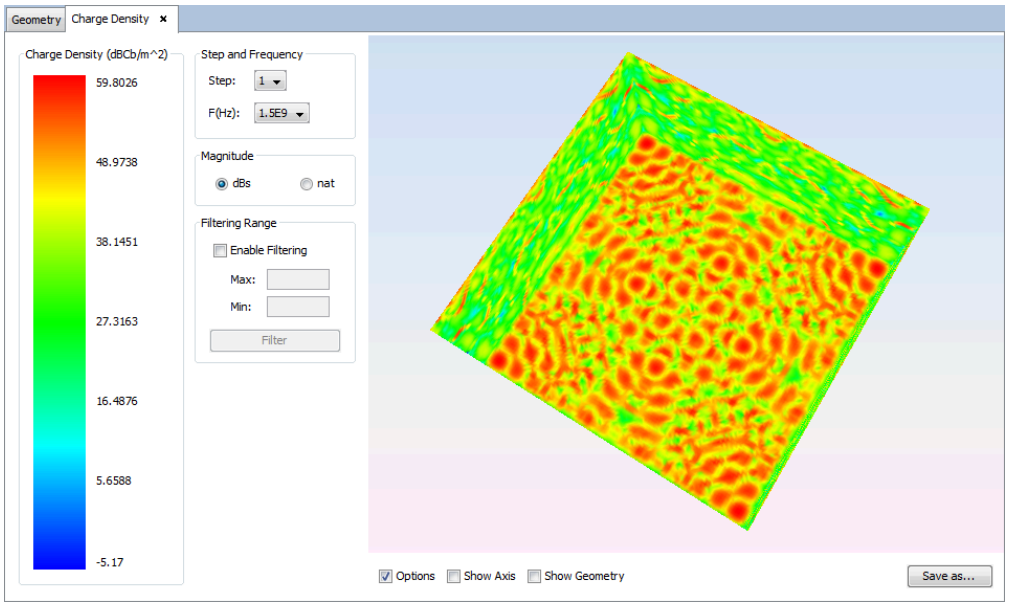


Figure 1305: Charge Density

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> Export Multipole File' option to save the multipole file on a selected path for use in other simulations. Previous, select the step and frequency for the multipole file.

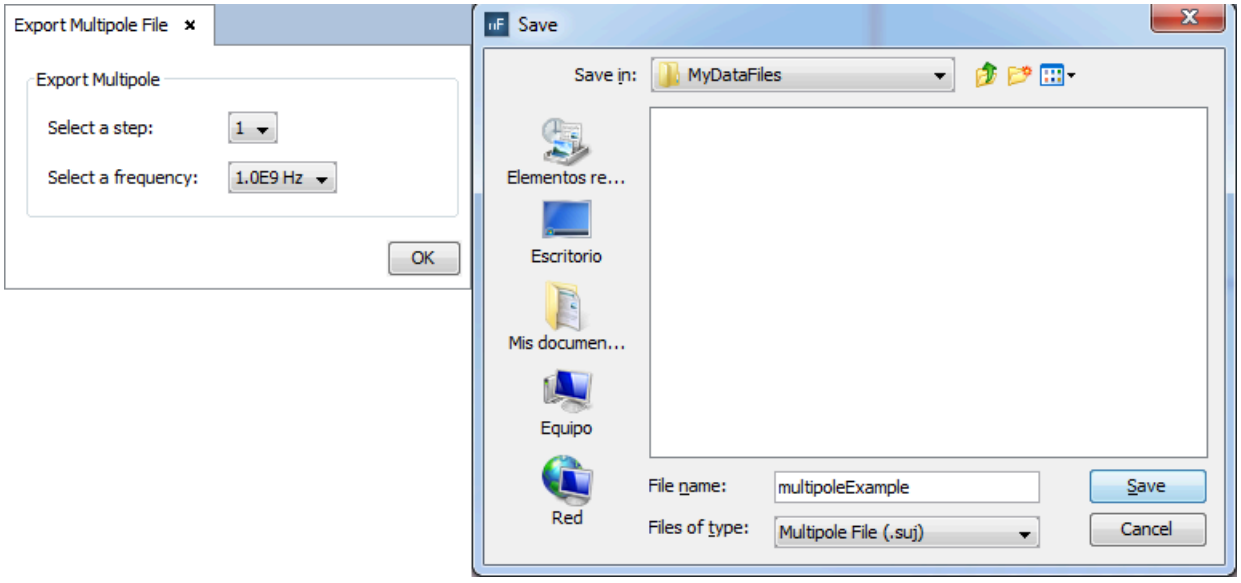


Figure 1306: Export Multipole

Step 8: Delete the geometry model and the dipole antenna and set the multipole file saved as an antenna. To obtain more information about sources and antennas see Antennas.

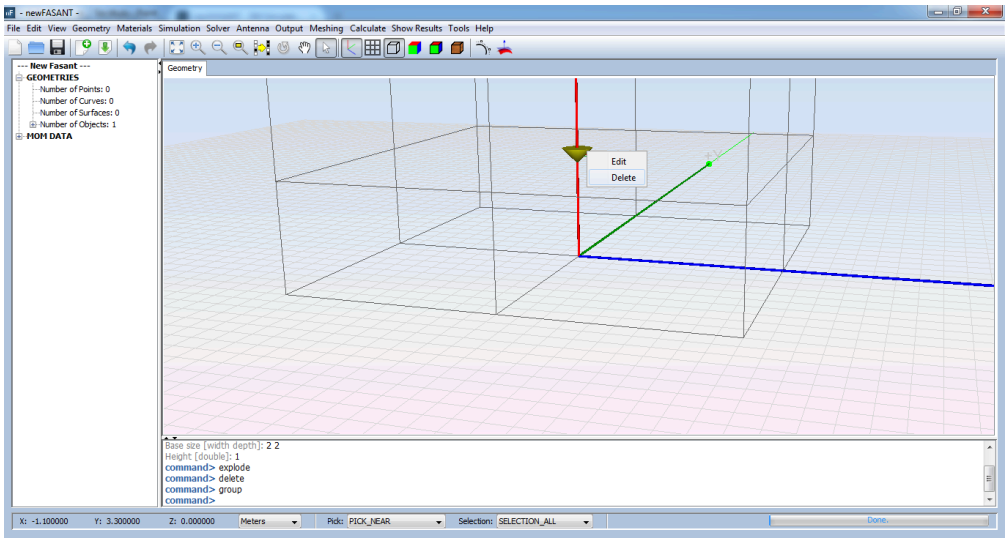


Figure 1307: Delete Antenna

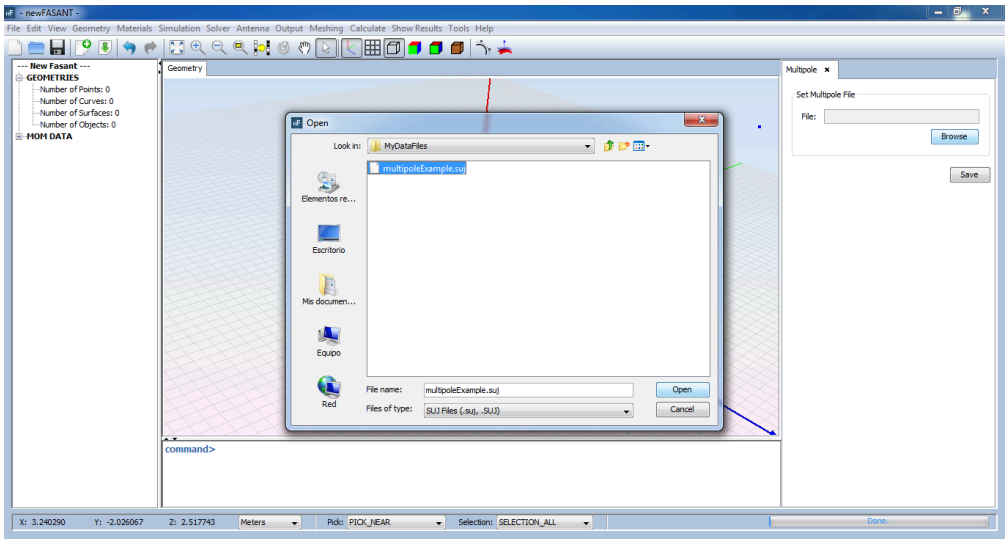


Figure 1308: Import Multipole File as Antenna

Step 9: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

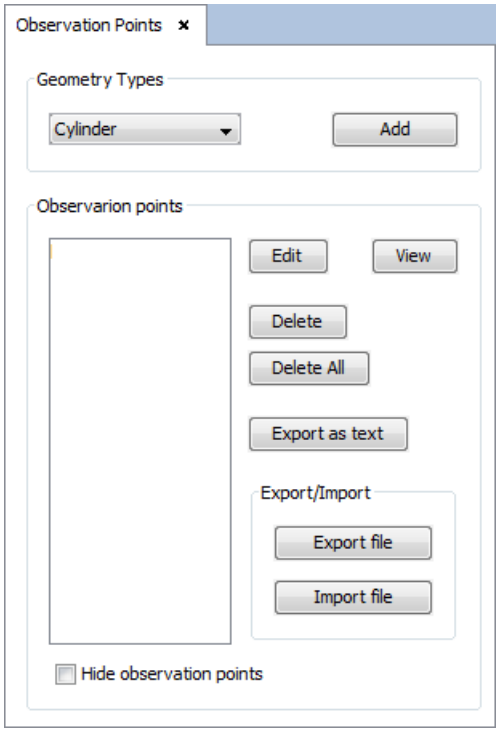


Figure 1309: Observation Points panel

To add a line visualization, select 'line' on the selector of 'Geometry Types' section and click on 'Add' button. The line parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

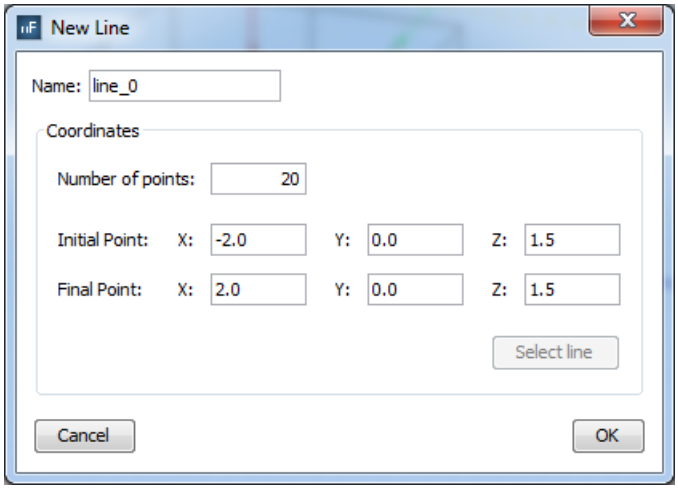


Figure 1310: Observation Line panel

The observation line will appear as a dashed line on the position configured.

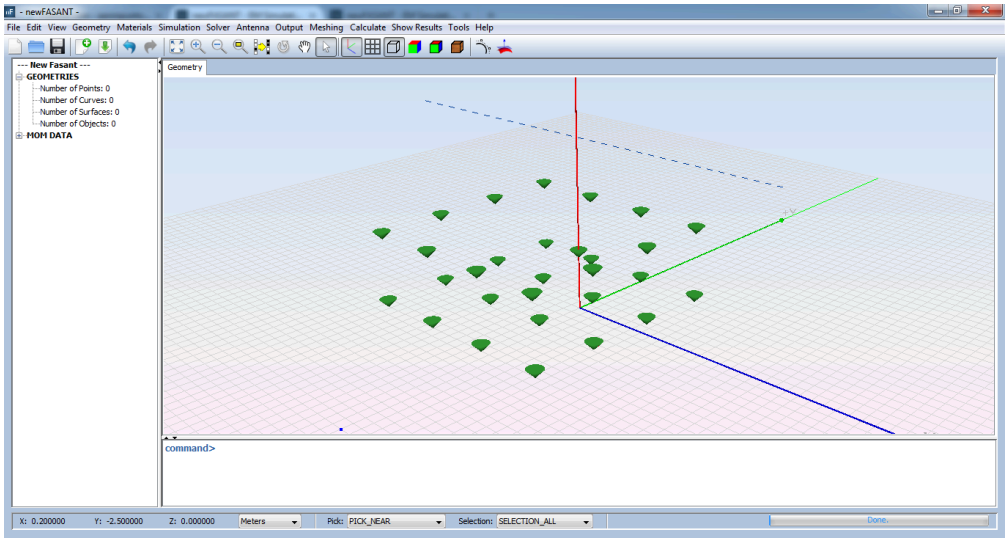


Figure 1311: Observation Line visualization

Step 10: Meshing and Execute as in steps 5 and 6.

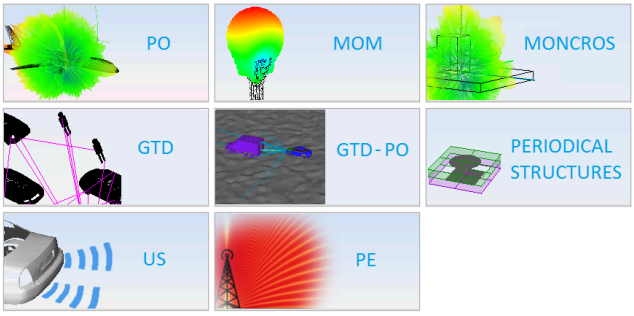
Step 11: Show Results again to ensure that results are similar to the original simulation.

Multipole Array Simulation

This example shows how to simulate an array of dipoles and how to create the multipole file and how to use it in another project.

Step 1: Create a new MOM Project.

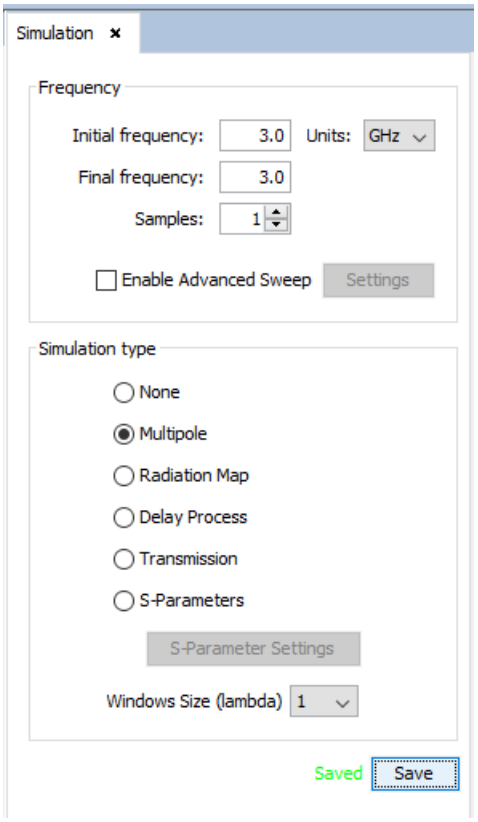
Open newFASANT and select 'File --> New' option.



Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Set Simulation Parameters

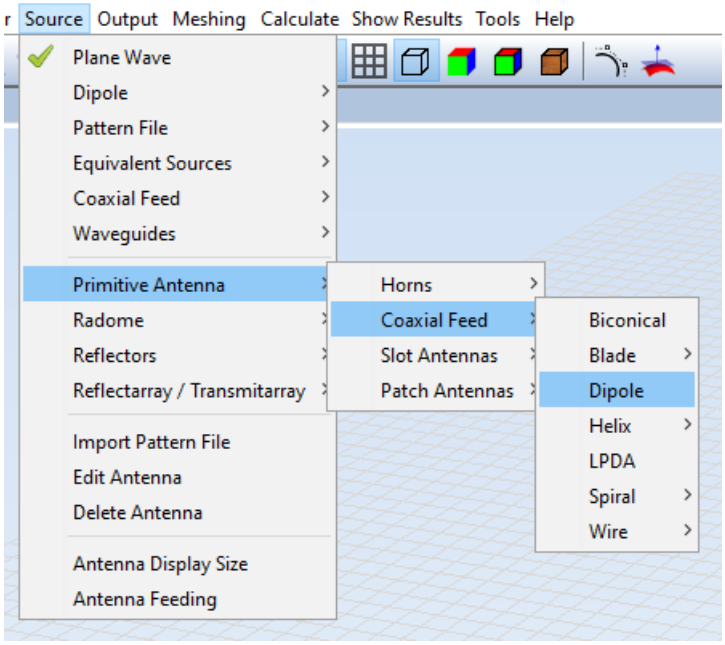
Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set up the frequency at 3 GHz and the multipole option, as the next figure shows and save it.

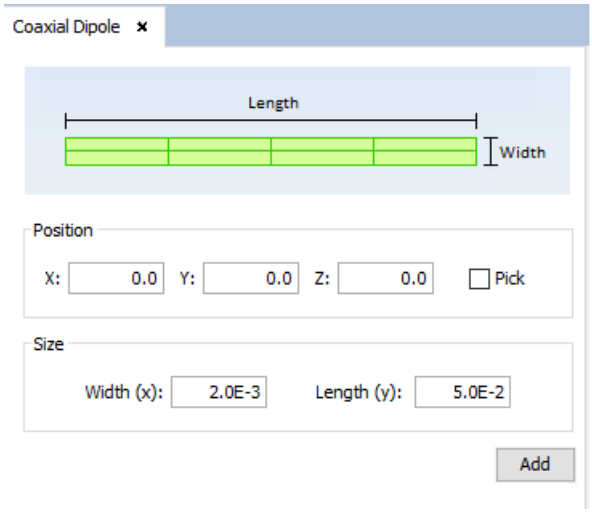


Step 3: Set the source parameters. To obtain more information about sources and antennas see Antennas.

In this example, a physical dipole (coaxial feed) is set up.

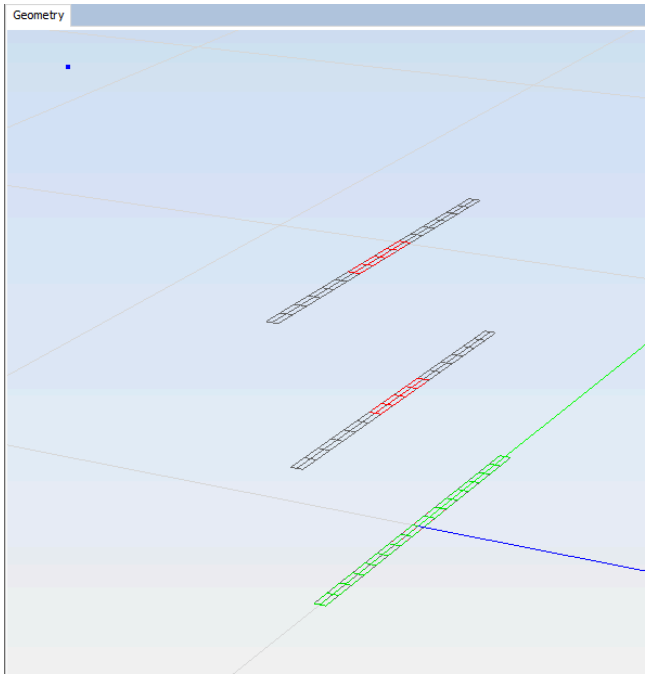
Select 'Antenna --> ‘Primitive Antenna’ --> ‘Coaxial Feed’ --> ‘Dipole' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.





Step 4: Using the array command, a small array is created.

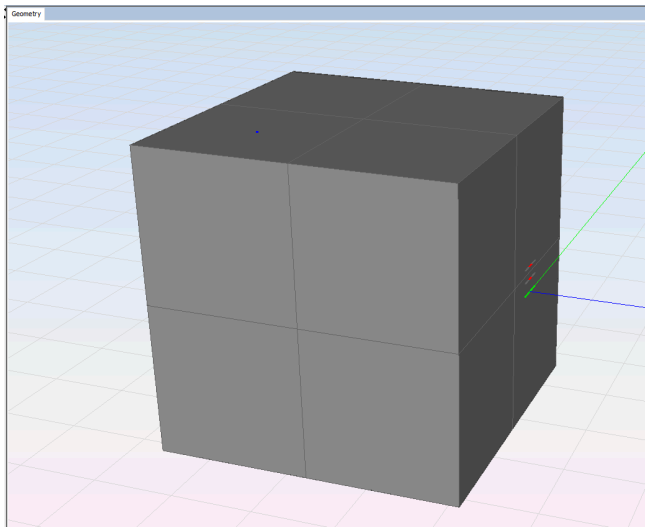
```
command> array
Select objects on screen (Press enter when done):
Number of objects in X axis (default=1): 1
Number of objects in Y axis (default=1): 1
Number of objects in Z axis (default=1): 3
Select start point [x y z] (default=(0,0,0)):
Select end point [x y z]}: 0 0 0.022
command>
```



Step 5: Create the geometry model. To obtain more information about geometries generation see Geometry Menu.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

```
command> box
First corner of base [x y z]: -0.525 -0.25 -0.25
Base size [width depth]: 0.5 0.5
Height [double]: 0.5
command>
```



Note if you are doing this example to obtain the multipole file (arraySmallBoxMultipole4lambda.suj) to be used on (GTD-Example) then, to get a faster simulation, the dimensions of the box are:

First point -0.225, -0.1, -0.1

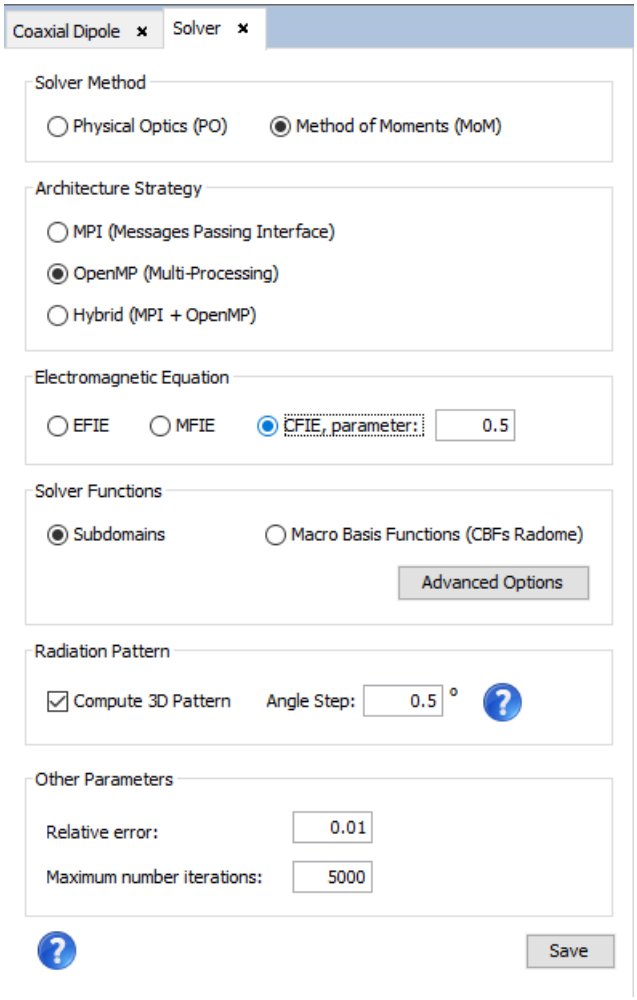
Width 0.2

Depth 0.2

Height 0.2

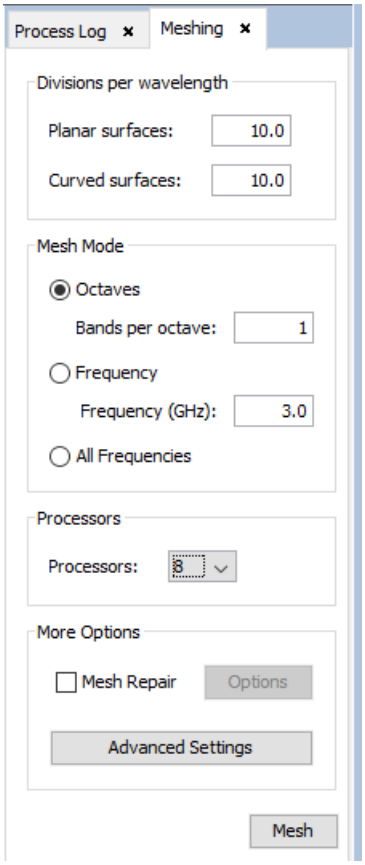
Step 6: Solver parameters menu.

As you can see in the below figure, in this example, CFIE is setting up.



Step 7: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing, it is recommended to run the process of meshing with the number of physical processors available to the machine.

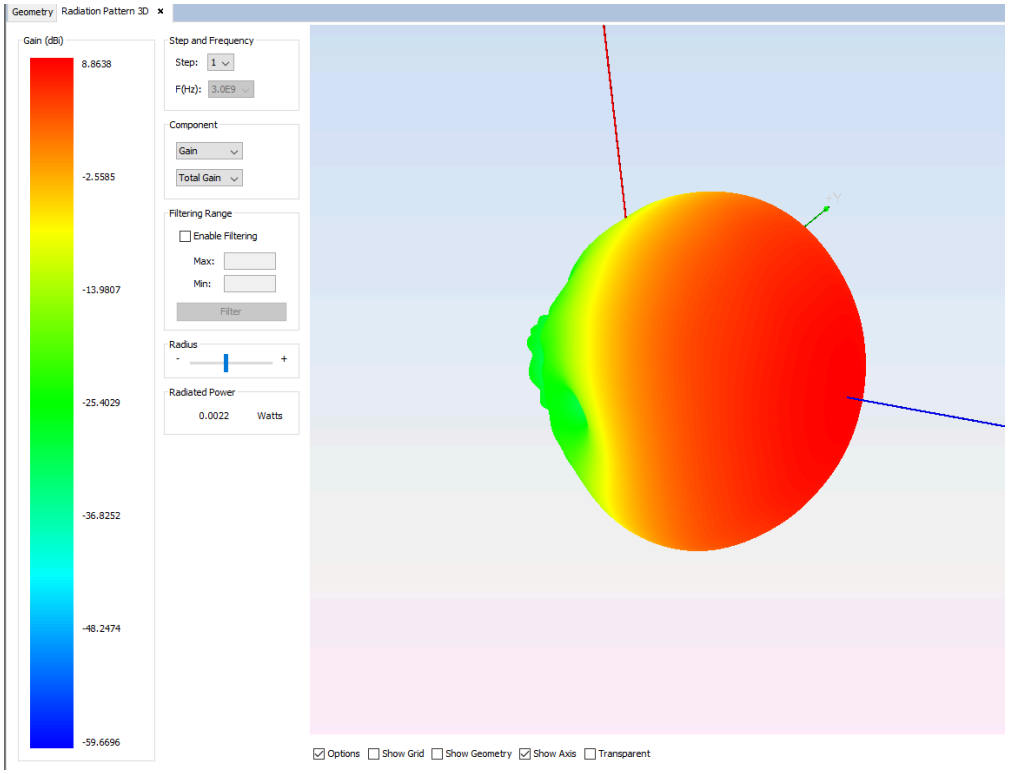


Step 8: Execute the simulation. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters.

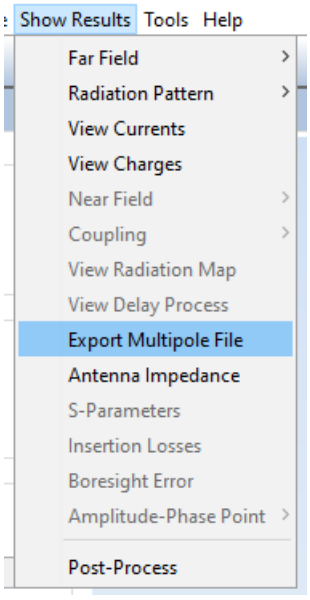
Step 9: Show Results. To get more information about the graphics panel advanced options (clicking on the right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.



Step 10: Export Multipole File.

Select 'Show Results --> Export Multipole File' option to save the multipole file on a selected path for use in other simulations. Previous, select the step and frequency for the multipole file.



Step 11: Close the project and open a new MoM Project.

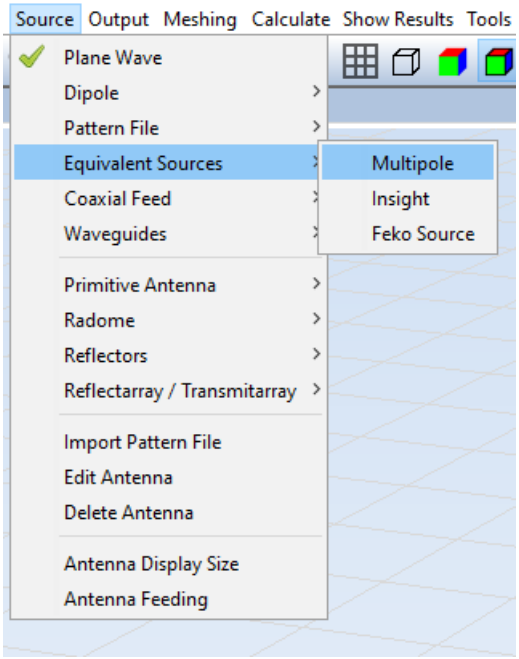
In the new project, we import the previously created multipole. Because the geometry in the previous project is very big we have many multipole, each multipole model the radiation of the current in

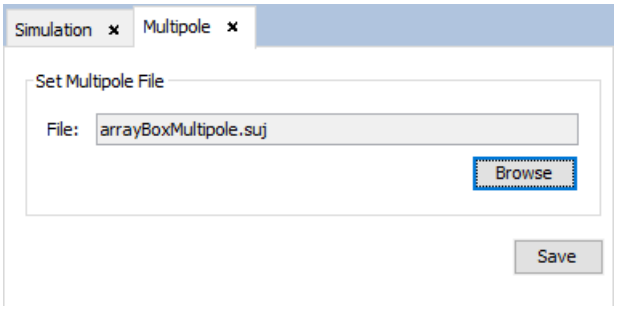
each cube of one wavelength of side. We can use this multipole to compute the fields at a distance greater than 2*(D**2)/lambda, where D is the size of the cubes. In this case this distance is given by 2*(lambda**2)/lambda = 2*lambda= 20 cm.

The process to create this project is similar to the previous one, the main changes are on the next steps:

Step 1: Set the source parameters. To obtain more information about sources and antennas see Antennas.

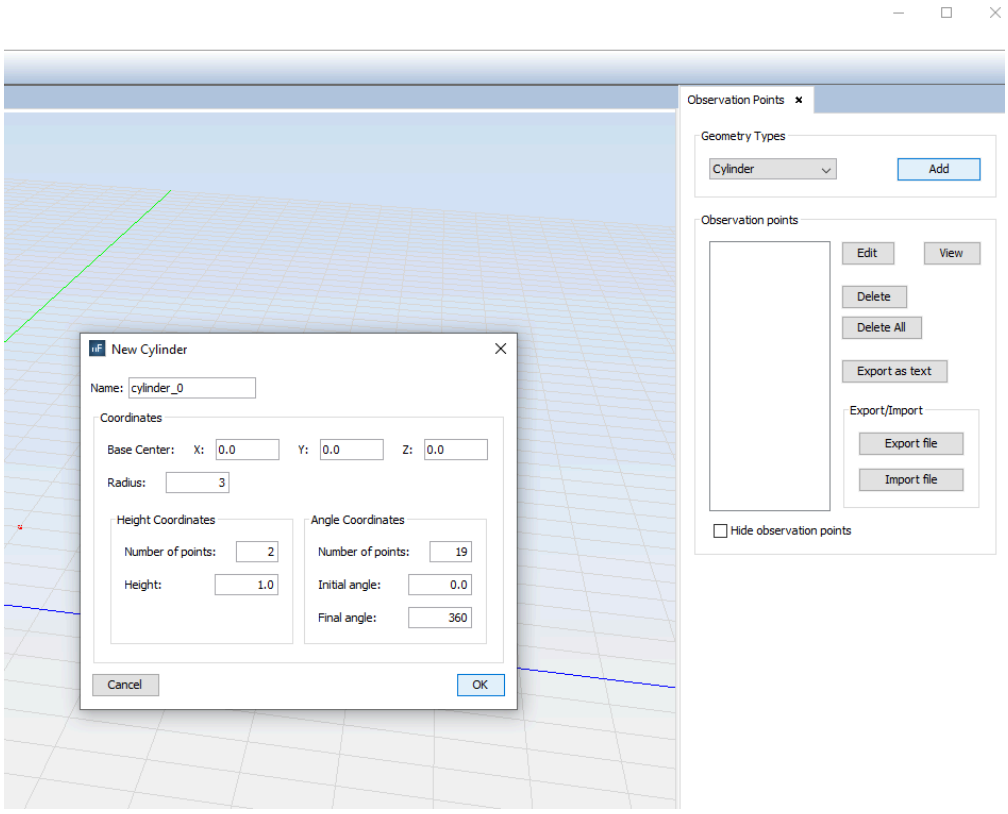
Select 'Antenna --> 'Primitive Antenna' --> 'Equivalent Sources' --> 'Multipole' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.





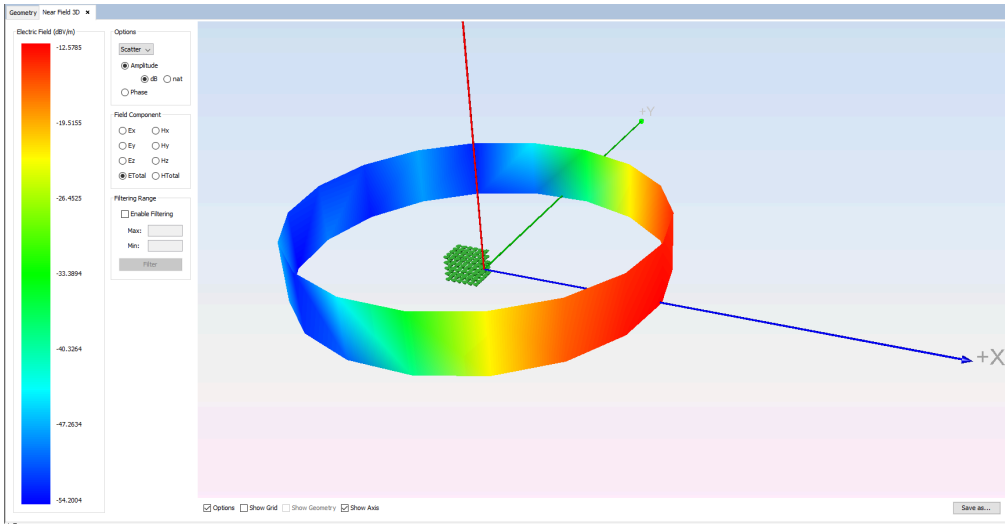
Step 2: Set Near Field Parameters

Select 'Output' --> 'Observation Points' option. Select 'cylinder' on the selector 'Geometry Types' section and click on 'Add' button. The line parameters panel will appear, then set up the values as the next figure show and accept it clicking on 'Ok' button.



Step 3: Show Results.

Select 'Show Results --> Near Field --> View Near Field Diagram' option to show the observation points diagram. Previously, select the observation to visualize and the step and frequency on the next figure.



11.15.6 Example 6: Common Simulation with a Coaxial Feed Source

This case explains how to calculate far-field, radiation pattern, current density, charge density, near field and impedance of an open box with a coaxial feed simple blade insides the box.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

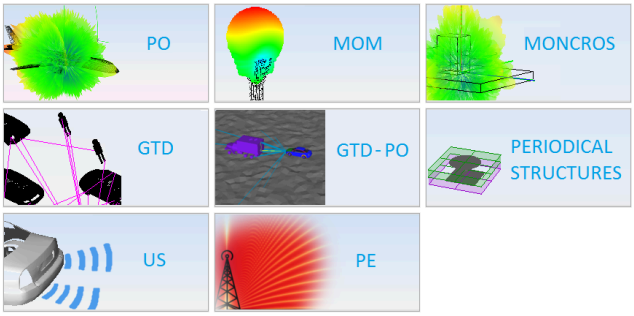


Figure 1312: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To obtain more information about geometries generation see Parameters.

Execute 'box' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

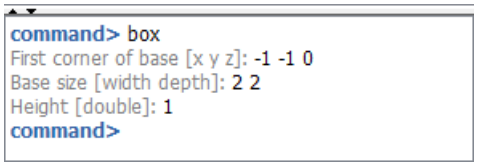


Figure 1313: Box parameters

Select box geometry on main panel and execute 'explode' command writing it on the command line. Then the box will be transformed into 6 surfaces.

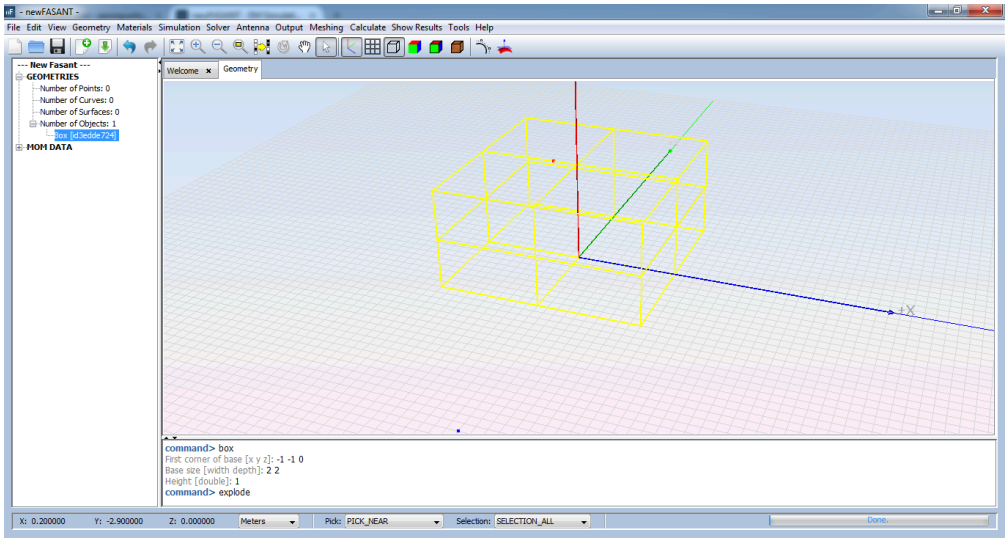


Figure 1314: Box selection and 'explode' command

Select the top surface and execute 'delete' command. Then the surface will be removed from the geometry.

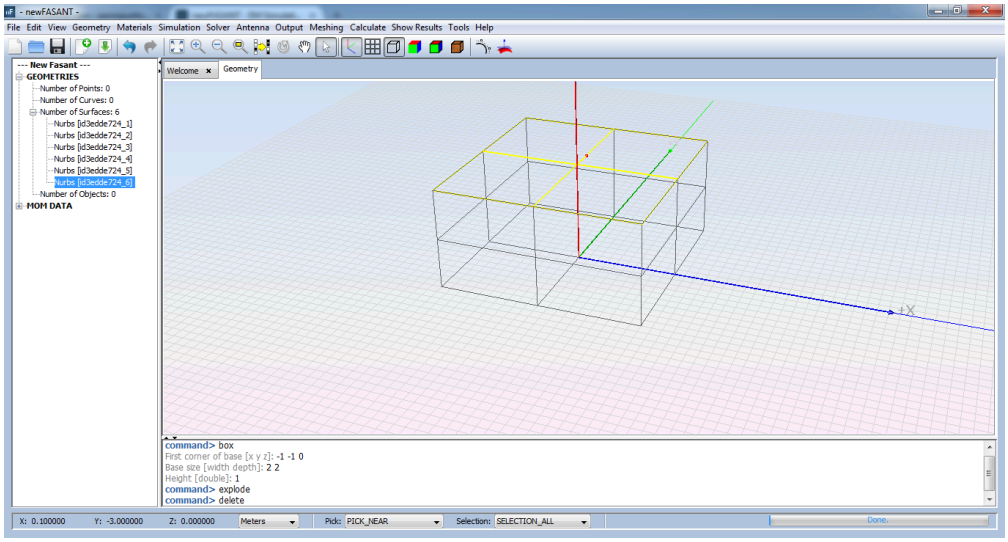


Figure 1315: Surface selection and 'delete' command

Select all remaining surfaces and execute 'group' command.

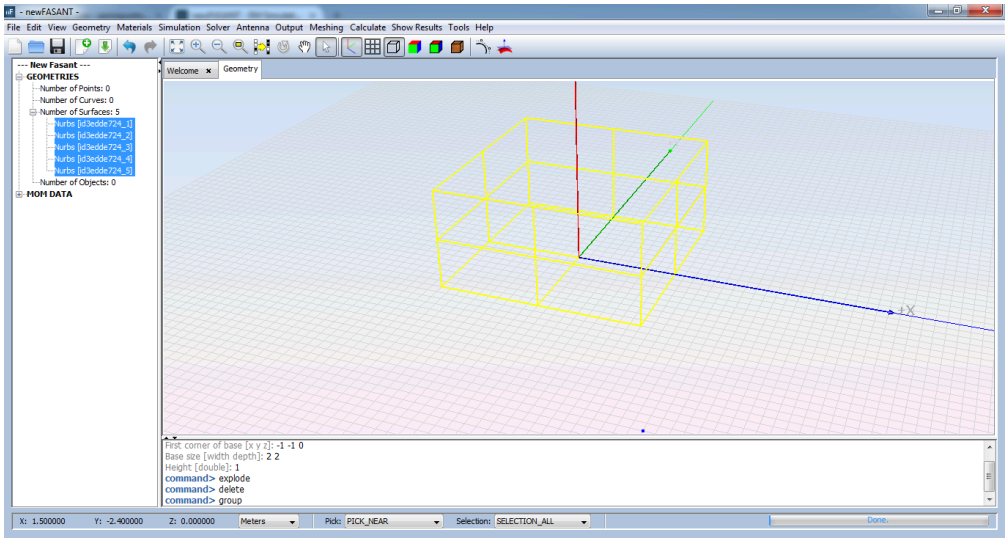


Figure 1316: Surfaces selection and 'group' command

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

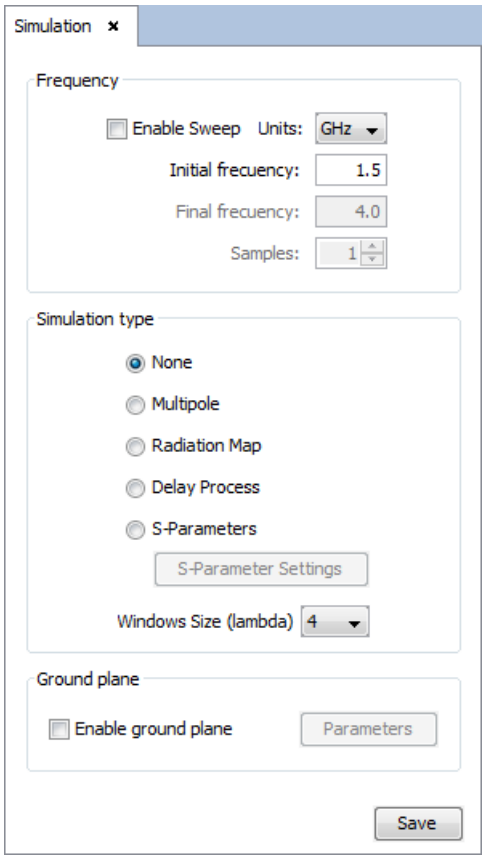


Figure 1317: Simulation Parameters panel

Step 4: Set the source parameters. To obtain more information about sources and antennas see Antennas.

Select 'Source --> Primitive Antenna --> Coaxial Feed --> Blade --> Simple Blade' option and set the parameters as show the next figure. Then save the parameters and the antenna appears.

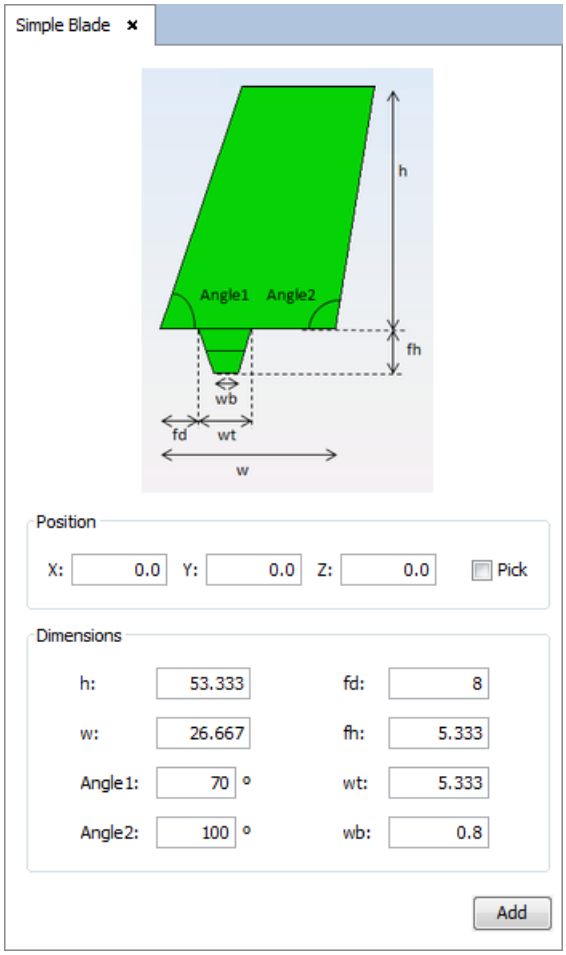


Figure 1318: Simple Blade Antenna panel

Step 5: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

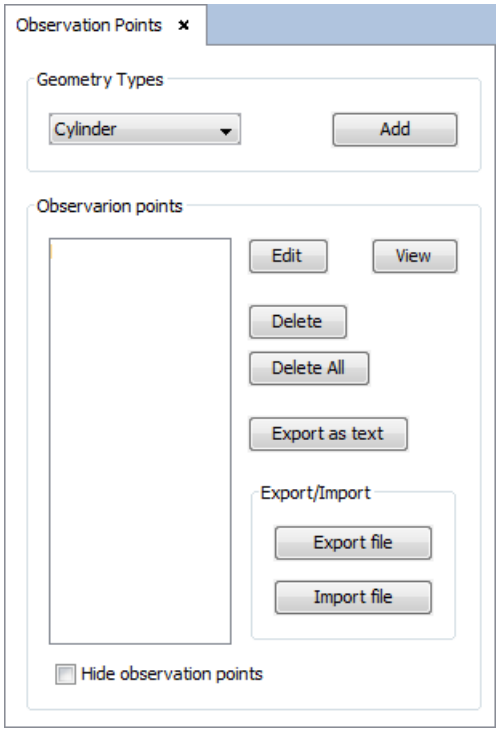


Figure 1319: Observation Points panel

To add a line visualization, select 'line' on the selector of 'Geometry Types' section and click on 'Add' button. The line parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

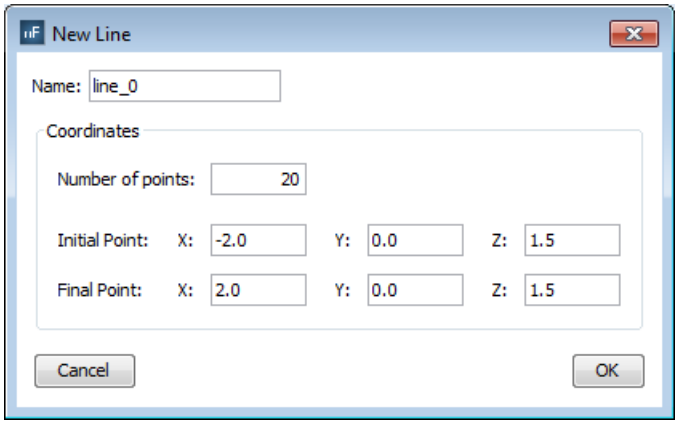


Figure 1320: Observation Line panel

The observation line will appear as a dashed line on the position configured.

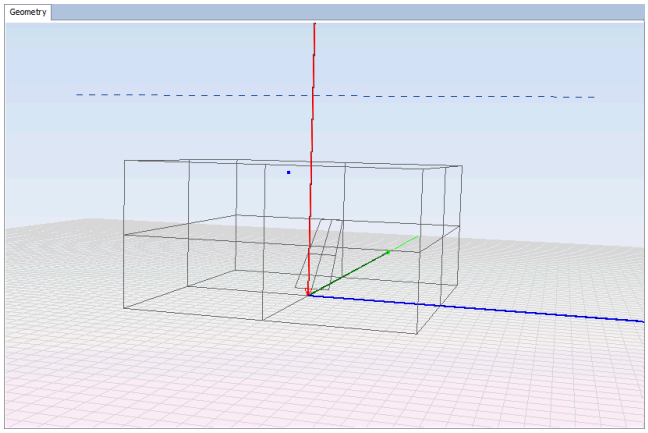


Figure 1321: Observation Line visualization

Step 6: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

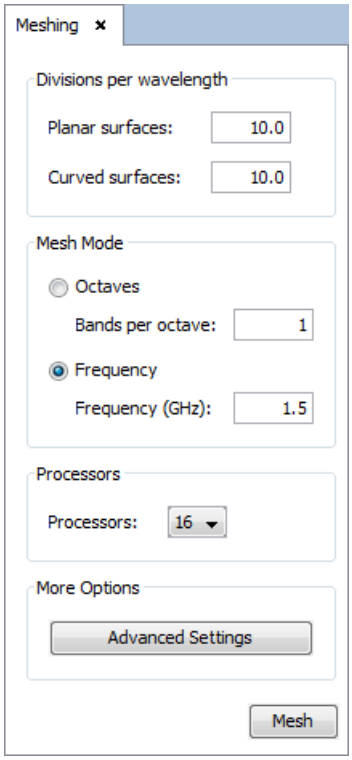


Figure 1322: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

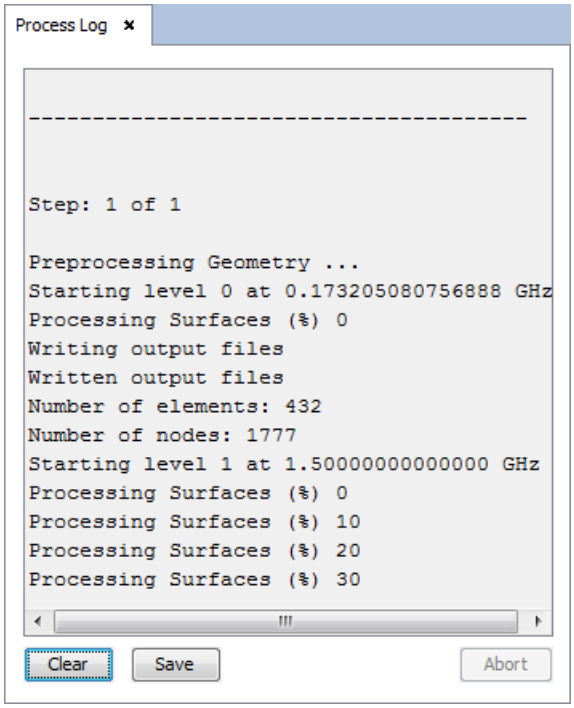


Figure 1323: Meshing process log

Step 7: Execute the simulation. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

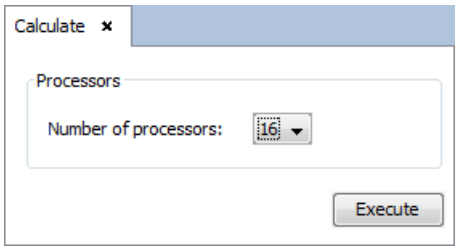


Figure 1324: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

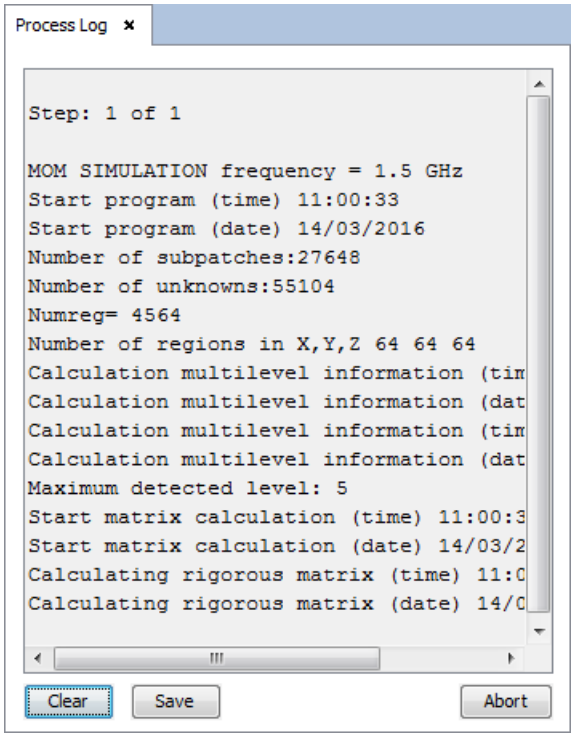


Figure 1325: Execute process log

Step 8: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

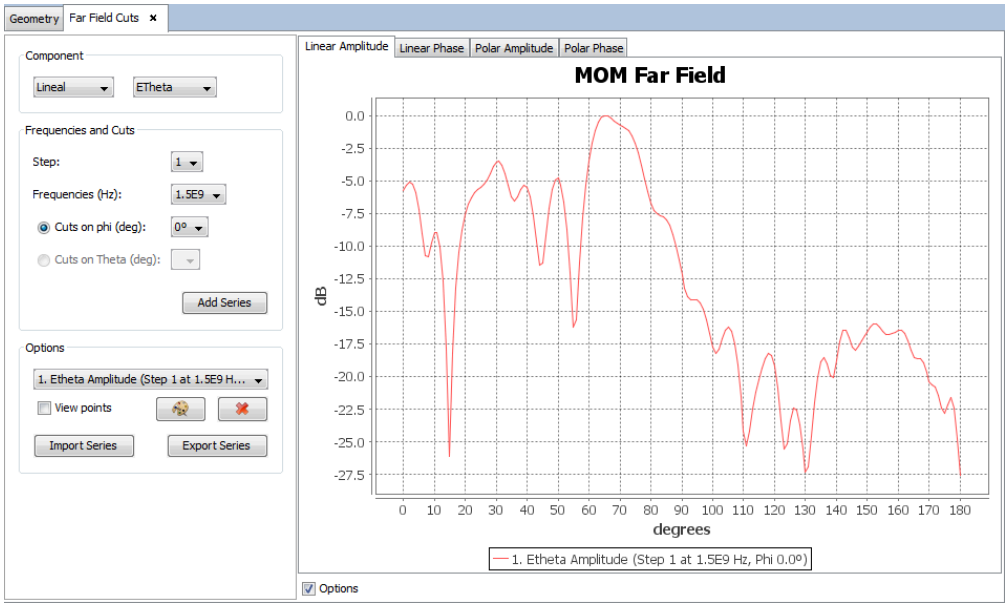


Figure 1326: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

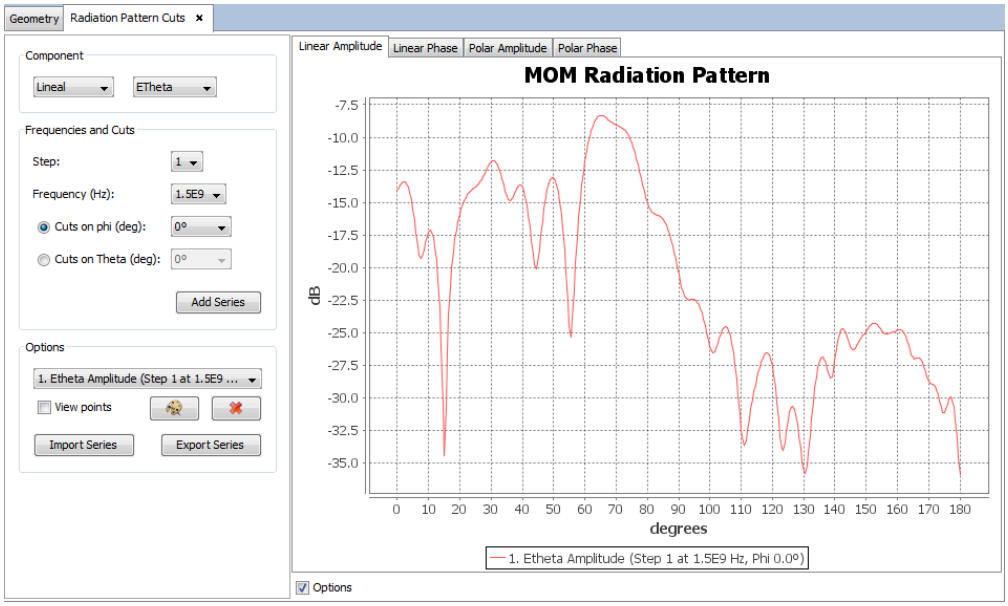


Figure 1327: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

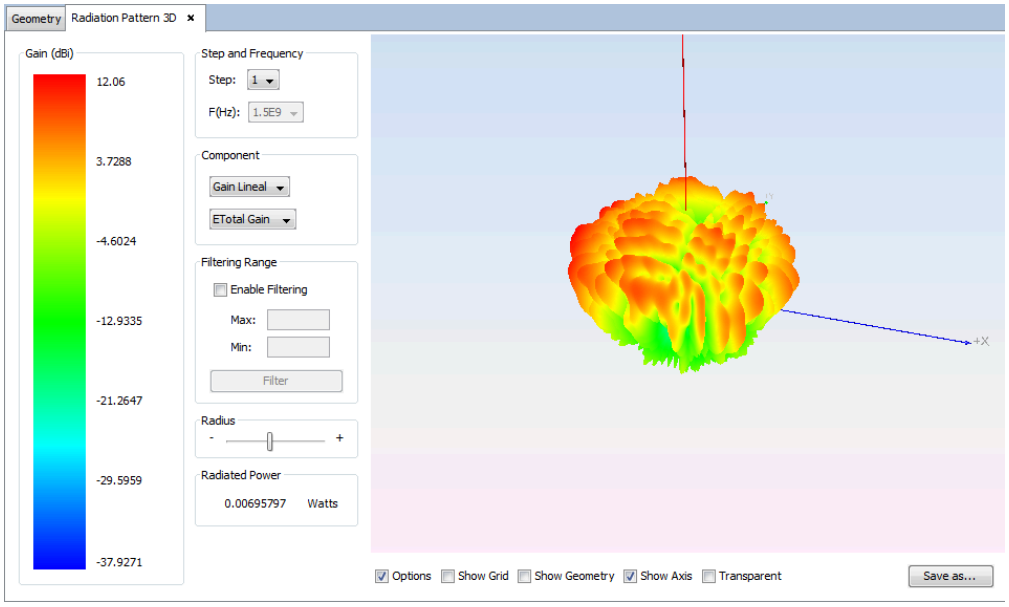


Figure 1328: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

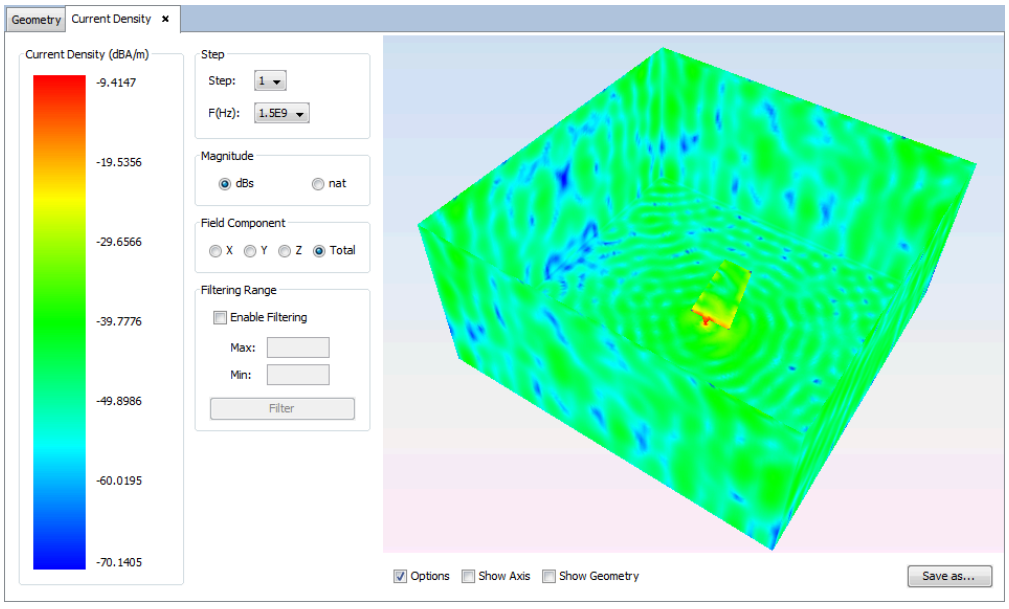


Figure 1329: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

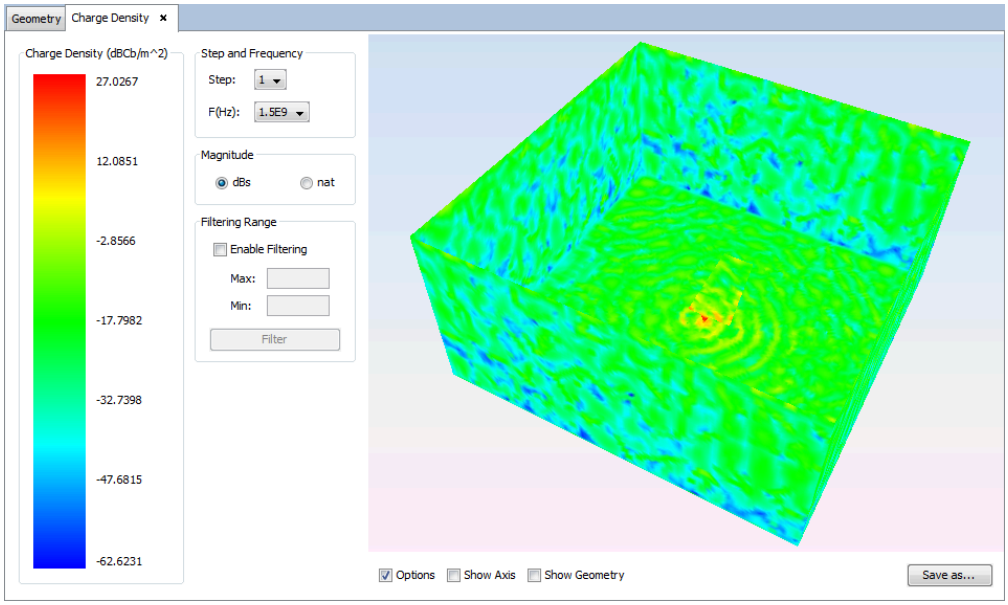


Figure 1330: Charge Density

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> Near Field --> View Near Field Diagram' option to show the observation points diagram. Previously, select the observation to visualize and the step and frequency on the next figure.

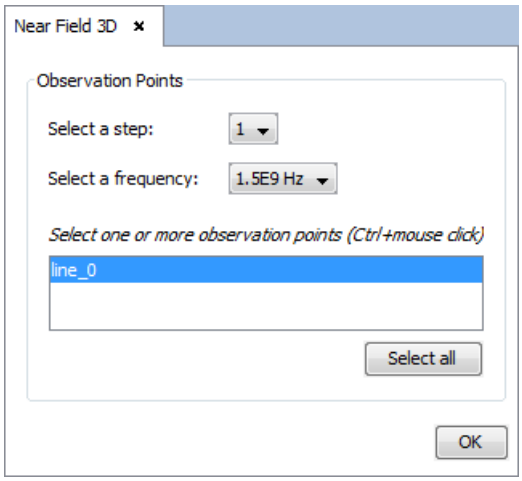


Figure 1331: Near Field diagram selection

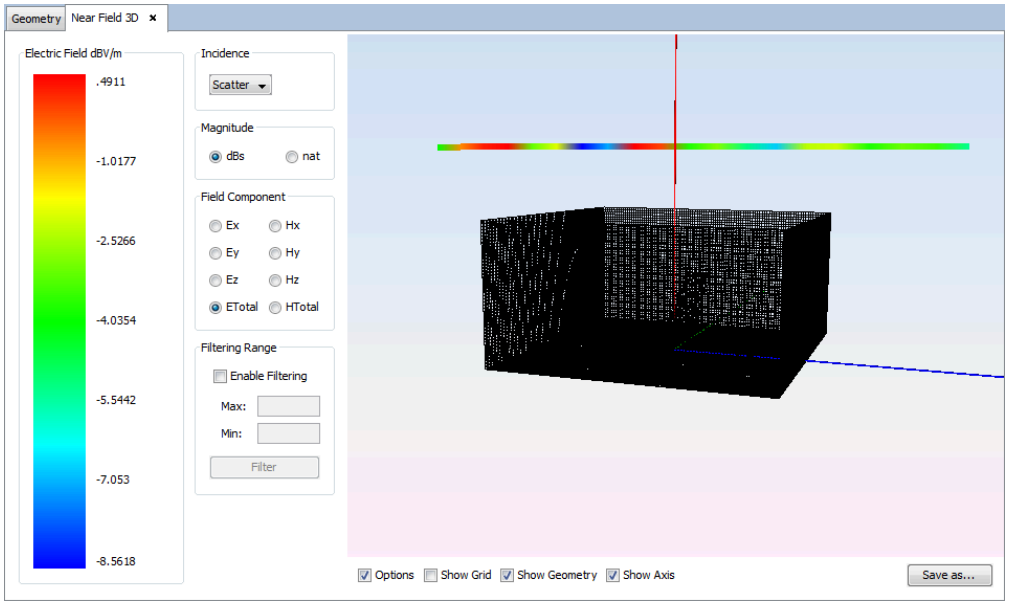


Figure 1332: Near Field diagram

Changing values for incidence, magnitude, field or component parameters the visualization for the new parameters will be shown.

Select 'Show Results --> Near Field --> View Observation Points' option to show the observation points chart.

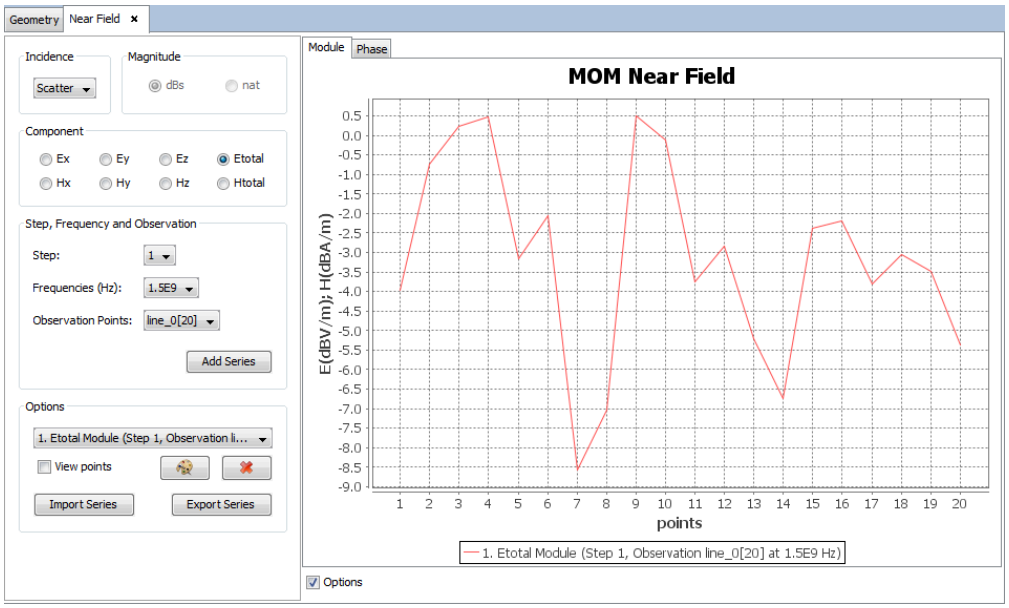


Figure 1333: Near Field Observation Points cuts

Selecting other values for incidence, component, step, frequency or observation parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Near Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> View Impedance' option to show the impedance values for the simulated coaxial feed.

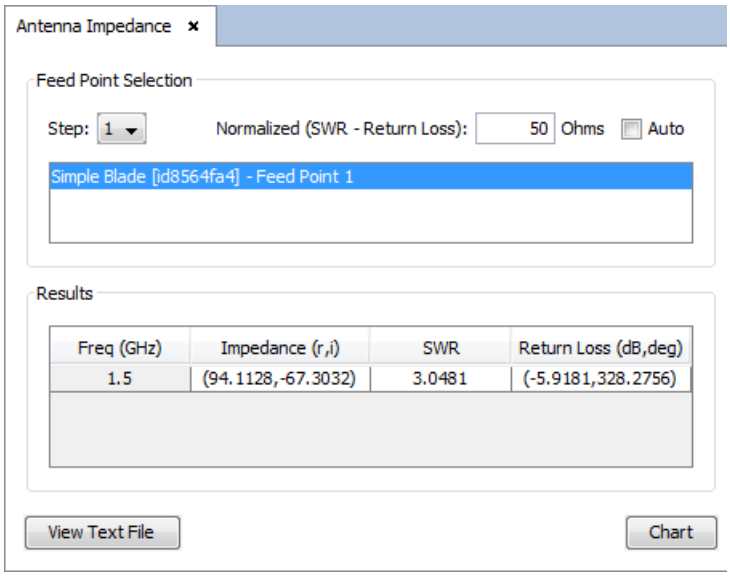


Figure 1334: Impedance panel

Selecting other values for the step or return loss parameters new values will be displayed for the selected parameters.

Clicking on 'Chart' button when any columns selected on 'Results' table, a plot with the values will be opened.

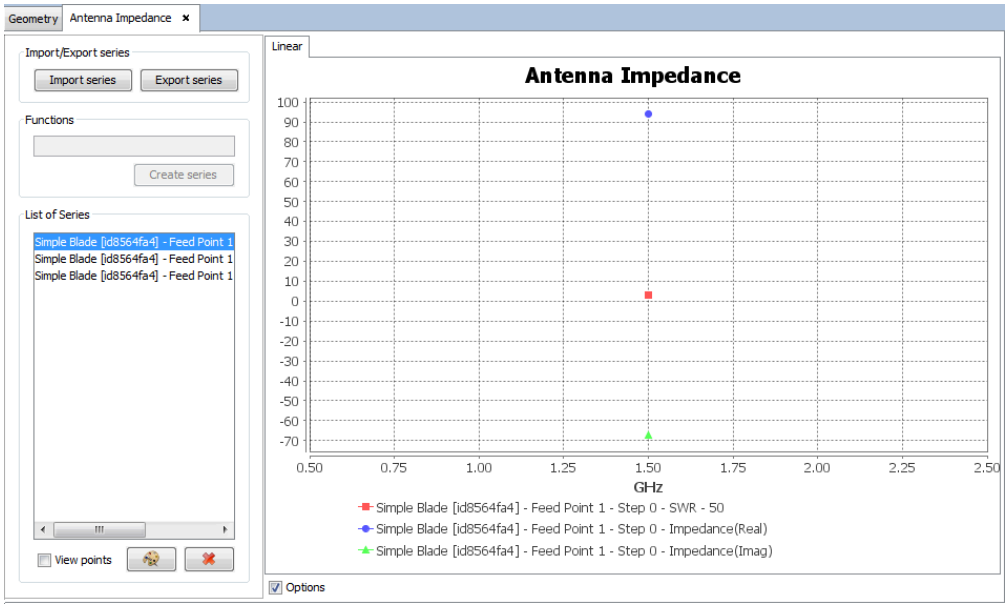


Figure 1335: Impedance chart

11.15.7 Example 7: Basic Reflectarray Simulation

This case explains how to analyze a reflectarray layout. The database used in step 4 to create the reflectarray layout is the database created on section Annex 1: Creating a Reflectarray Database.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

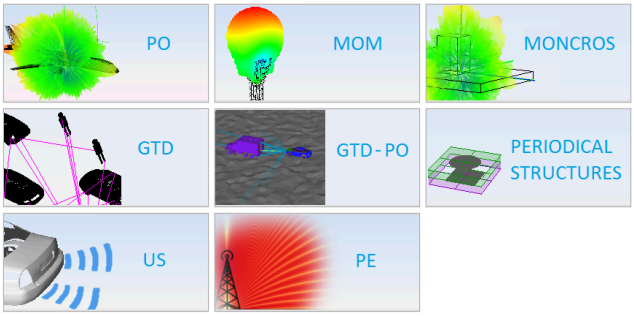


Figure 1336: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model. To get the complete information about geometries edition see 4. Geometry Menu.

First, select 'centimeters' on units list on the bar at the bottom of the main window.

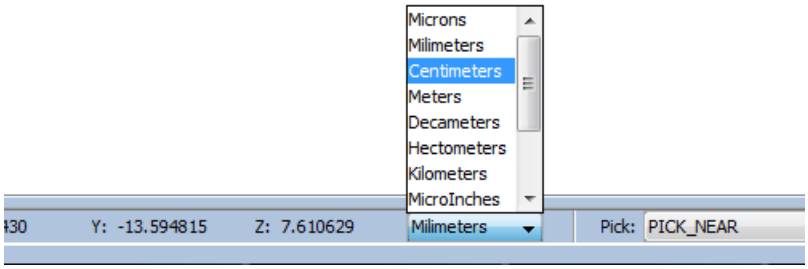


Figure 1337: Units selection

Execute 'plane' command writing it on the command line and sets the parameters as the next figure shows when command line ask for it.

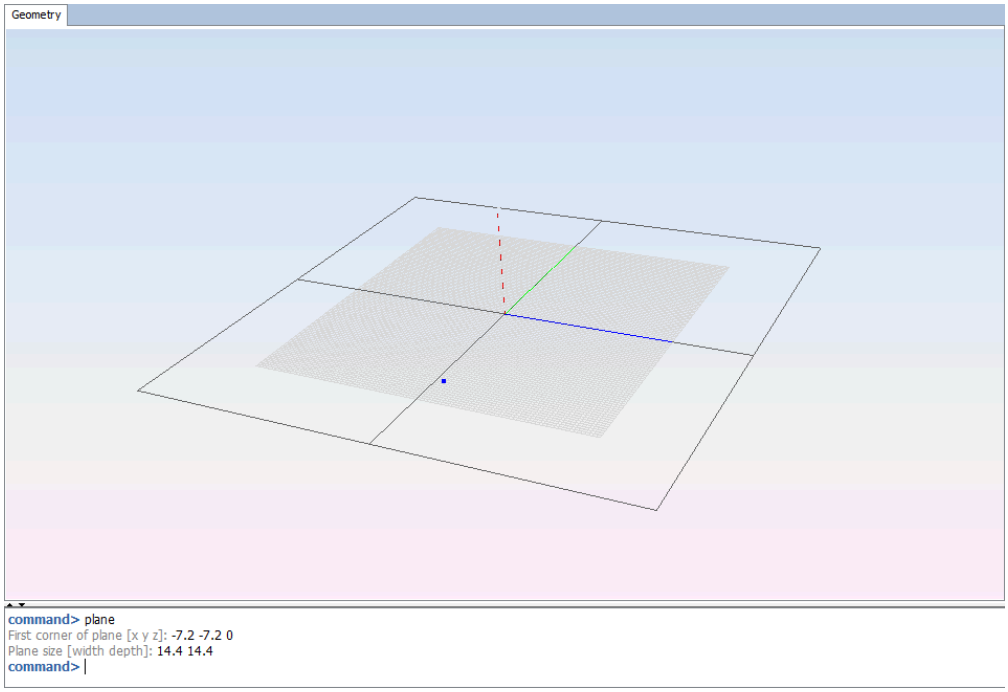


Figure 1338: Plane parameters

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

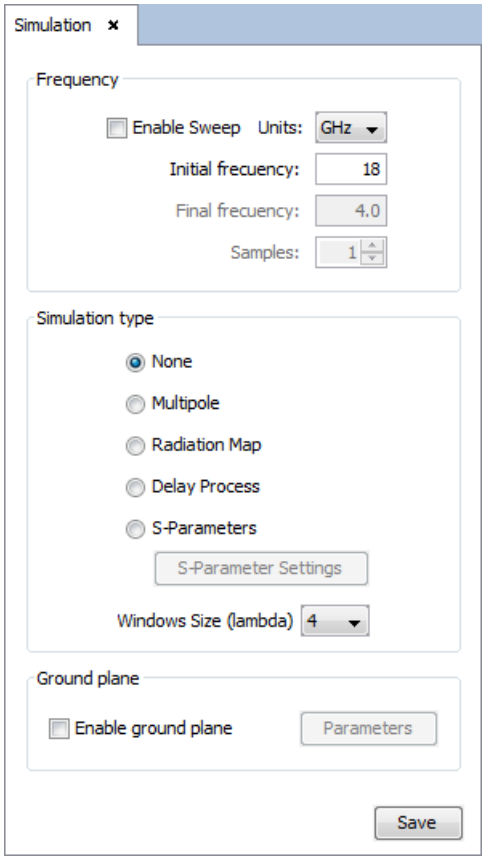


Figure 1339: Simulation Parameters panel

Step 4: Create a reflectarray layout.

Select 'Source --> Reflectarray --> New Layout' option. The following panel will appear. First load de database previously created with Periodical Structures module, using the icon with a green plus sign and selecting the database on the path where it is stored. Then select the parameters as the next figure shows:

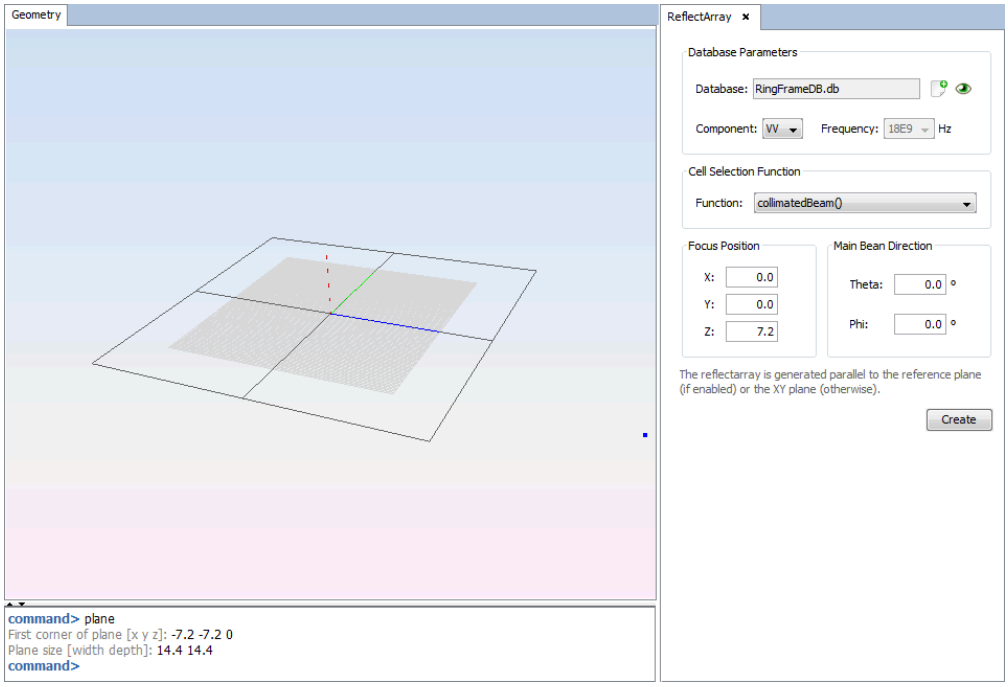


Figure 1340: Reflectarray New Layout panel

Clicking on the 'eye' icon, the parameters of the database will be shown on a new tab.

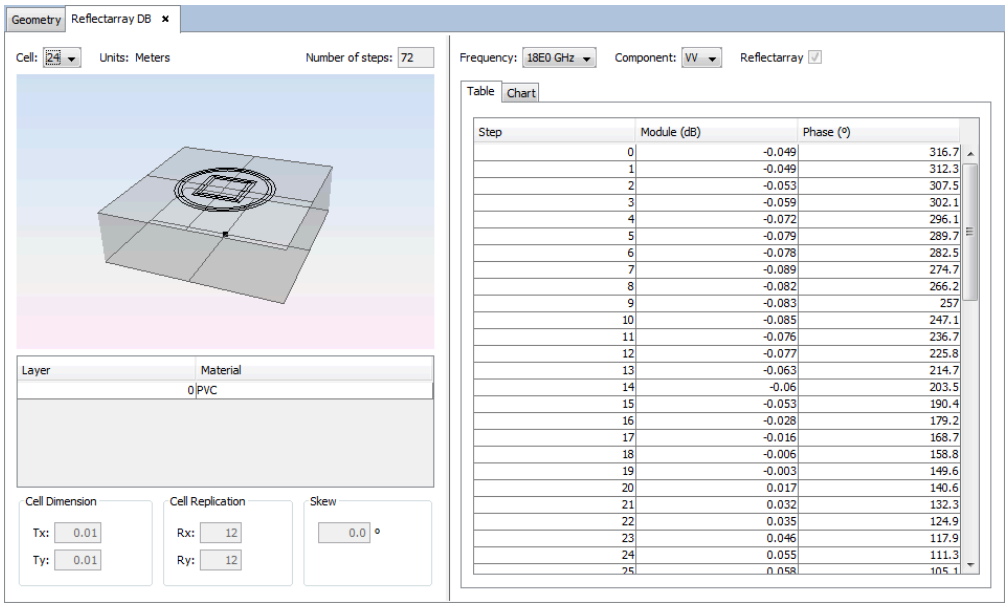


Figure 1341: Reflectarray Database Information panel

Select the plane previously created to generate the reflectarray over it. The reflectarray layout is generated placing in each cell the structure of geometries with the phase value closest to the phase value calculated by the user-selected function for this purpose. Then, click on 'Create' button and wait until the reflectarray layout is created.

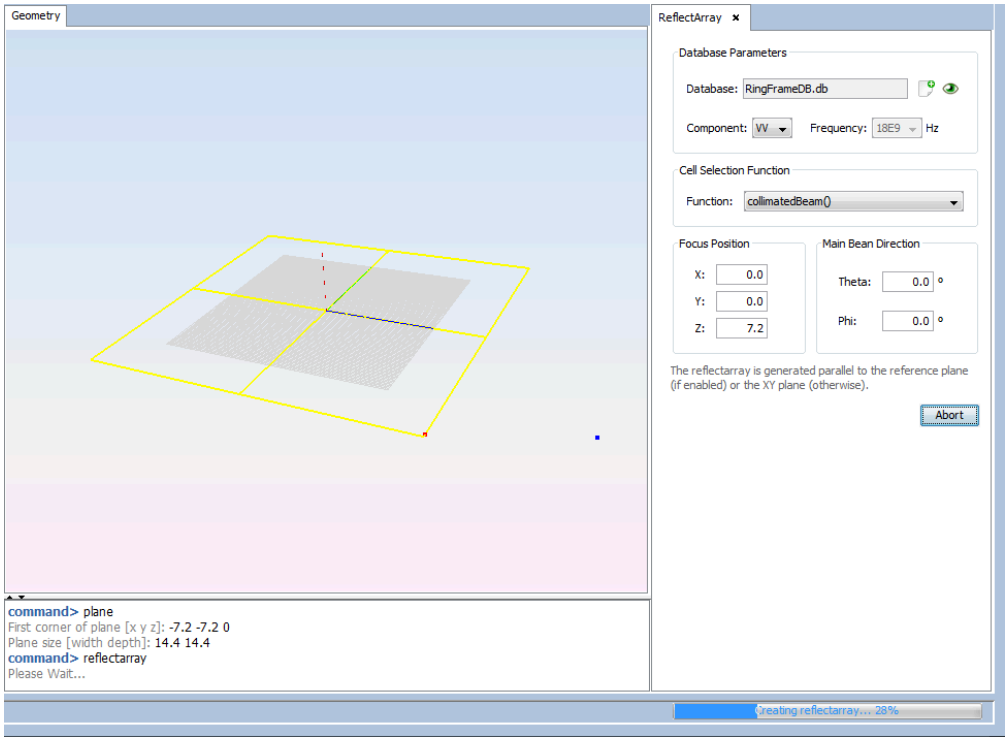


Figure 1342: Reflectarray creation process

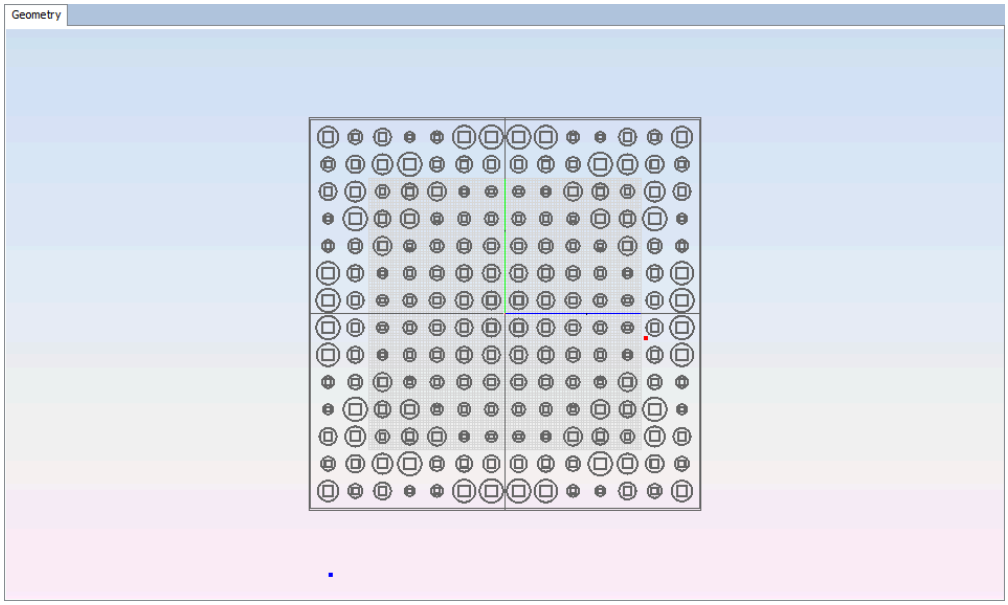


Figure 1343: Reflectarray Layout

Step 5: Set the source parameters. To obtain more information about sources and antennas see Antennas.

First, select 'View--> Reference Plane' and set the parameters as the next figure shows, then click on 'Update' button to put the reference plane as the figure:

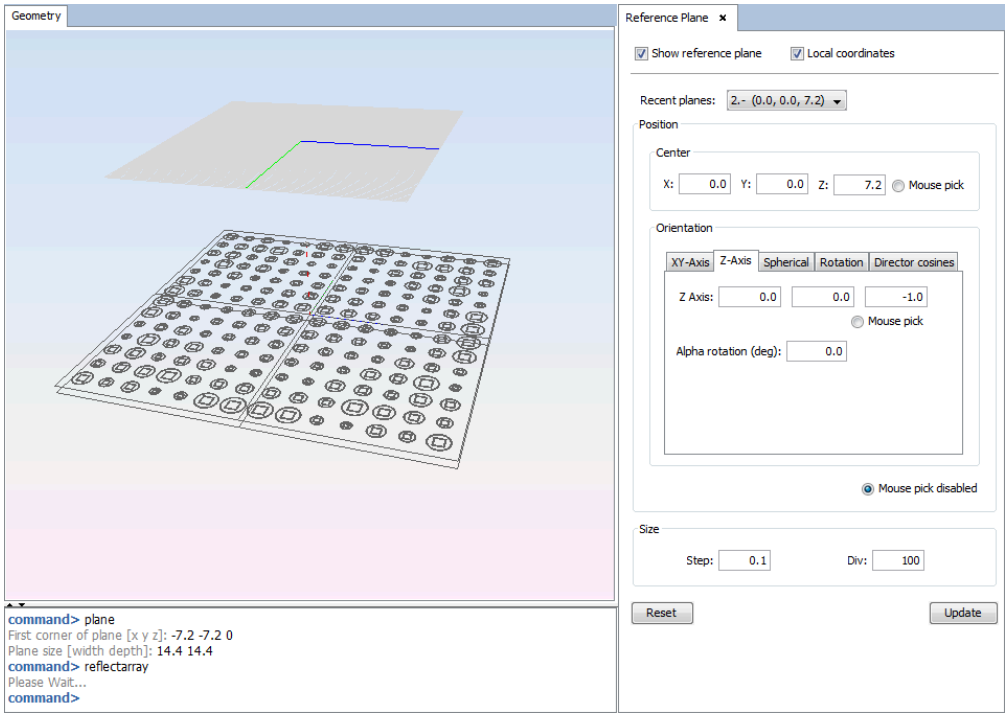


Figure 1344: Reference Plane panel

Select 'Source --> Import Pattern File' option and select `BOC0677.dia` file from 'myDataFiles' folder on the installation path of newFASANT software.

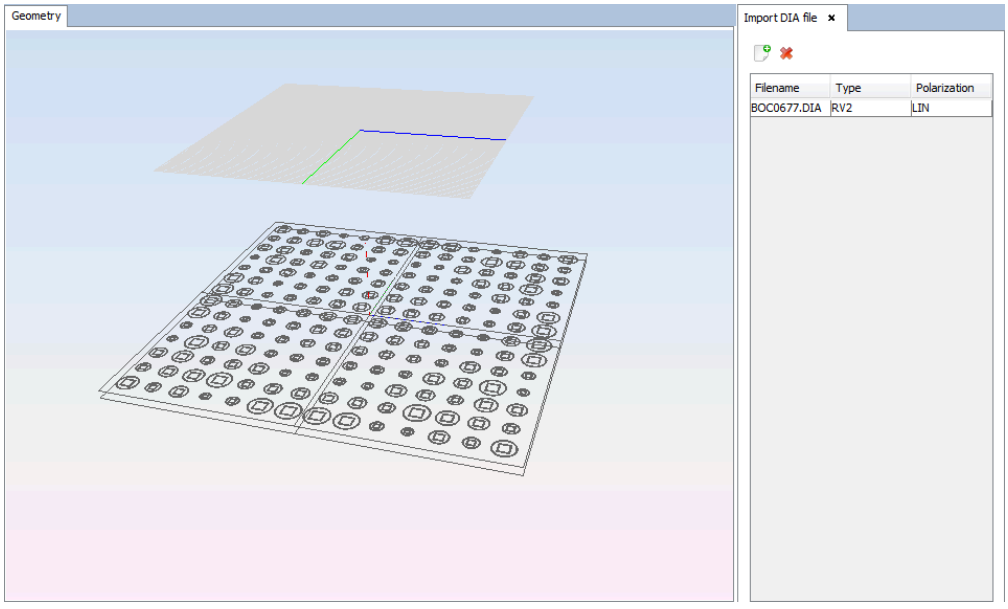


Figure 1345: Pattern File Import panel

Select 'Source --> Pattern File --> Pattern File Antenna' option and set the parameters as show the next figure. Then save the parameters and the antenna appears.

Important select the check box "Relative to reference plane" to set the position and orientation relative to the local reference plane.

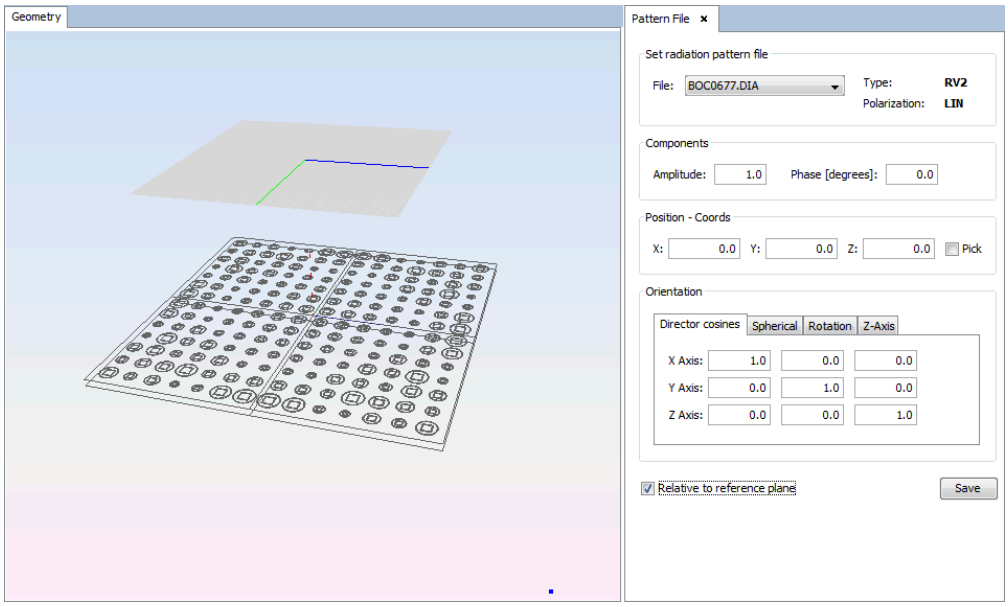


Figure 1346: Pattern File Antenna panel

Step 6: Meshing the geometry model. In order to obtain the shortest possible time for meshing,it is recommended to run the process of meshing with the number of physical processors available to the machine.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure.

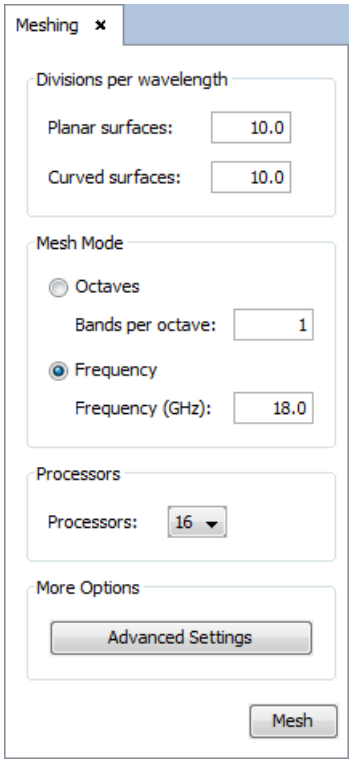


Figure 1347: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

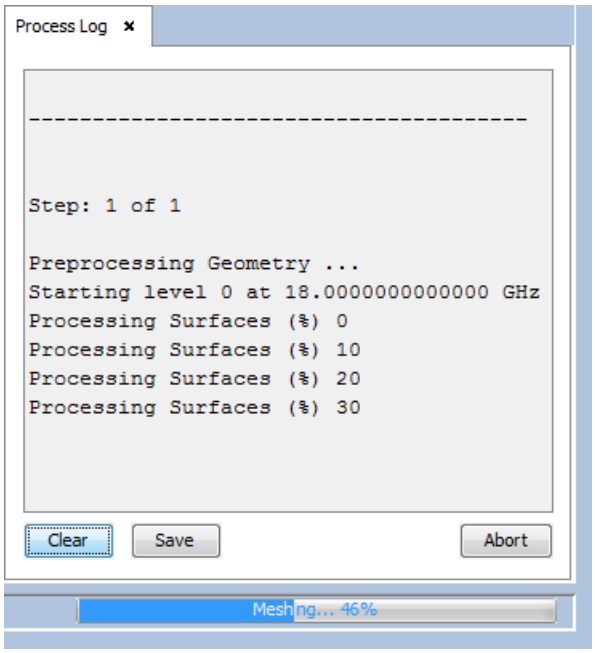


Figure 1348: Meshing process log

Step 7: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results,it is recommended to run the process with the number of physical processors available to the machine.

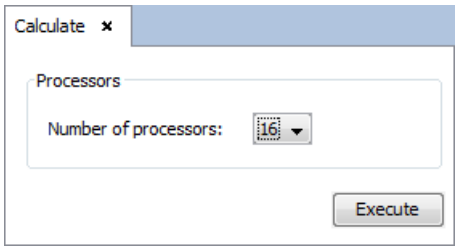


Figure 1349: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

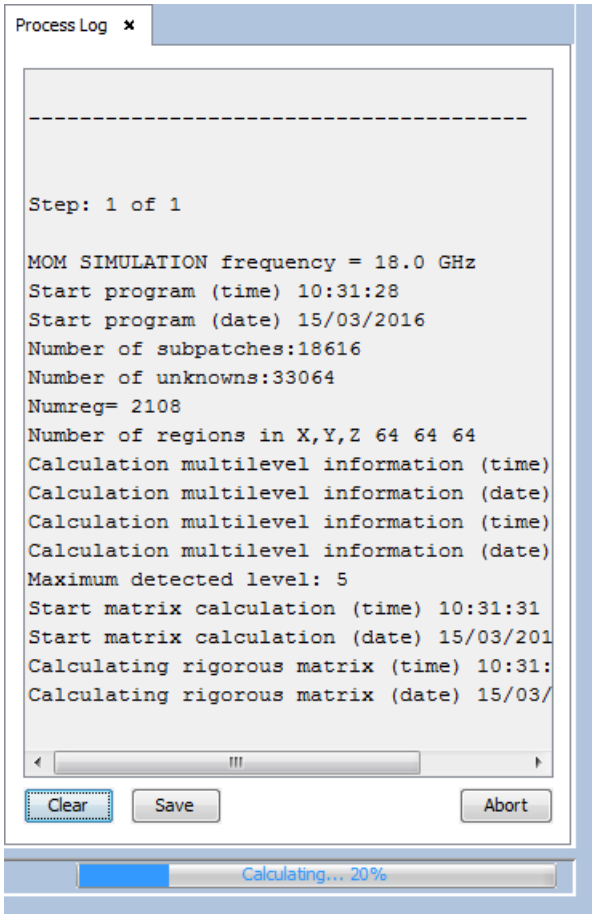


Figure 1350: Execute process log

Step 8: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

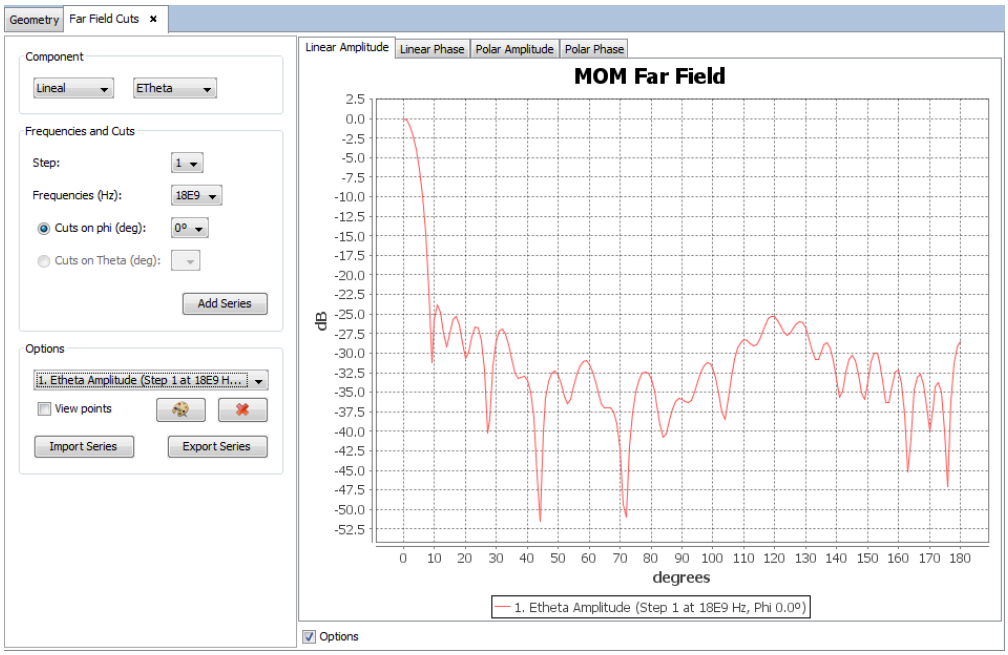


Figure 1351: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

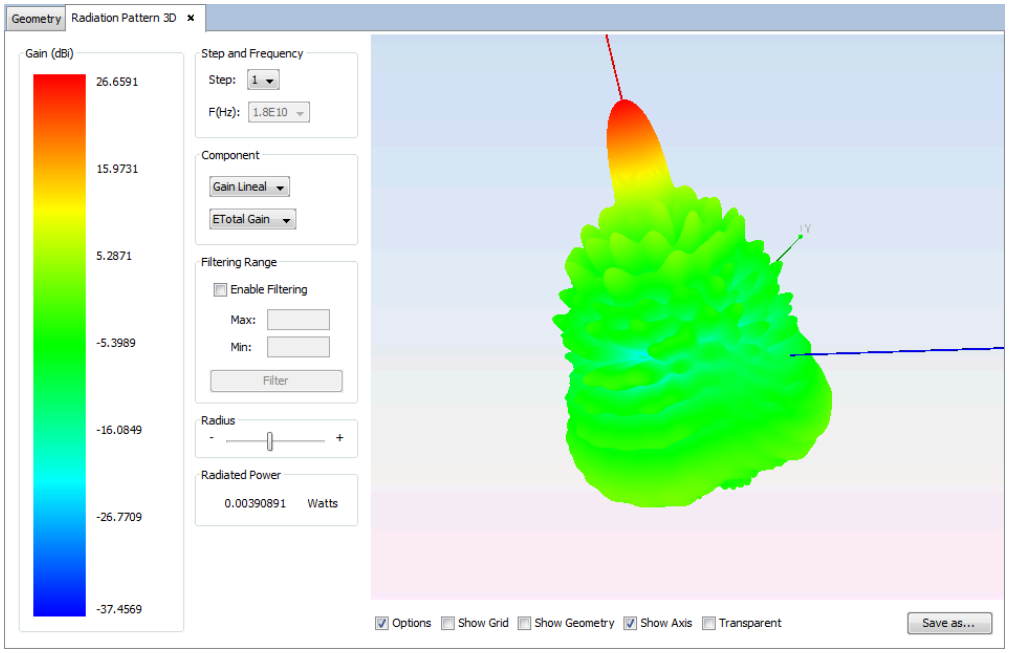


Figure 1352: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

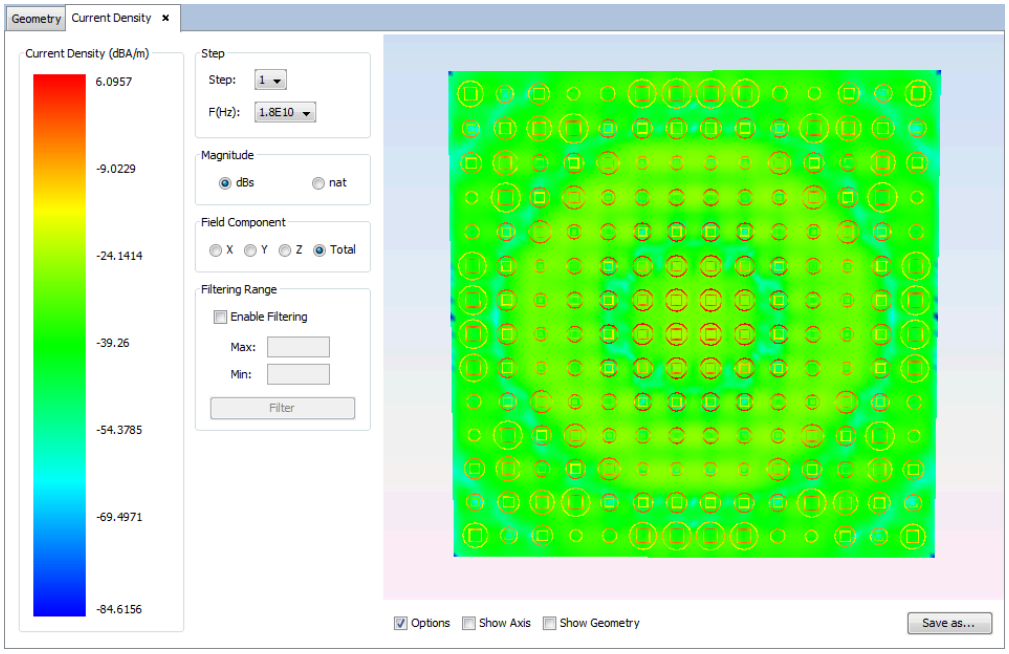


Figure 1353: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

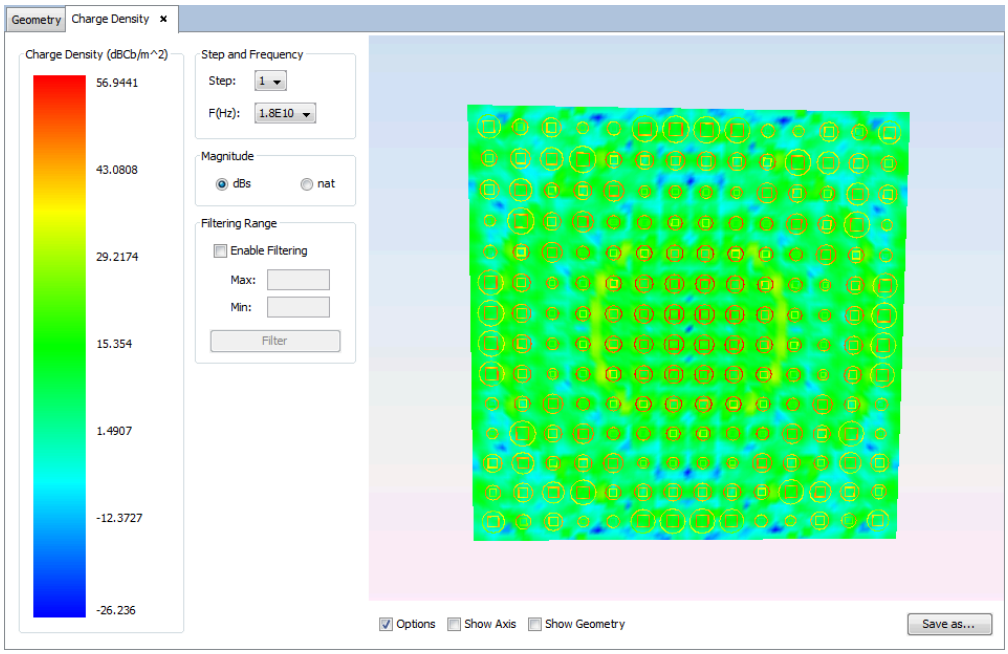


Figure 1354: Charge Density

11.15.8 Example 8: S-Parameters

This case explains how to calculate the S-parameters of an bifilar helix antenna placed on a plane.

Step 1 Create a new MOM Project

Open newFASANT and select **File - New** option. Select **MOM** on the module selection.

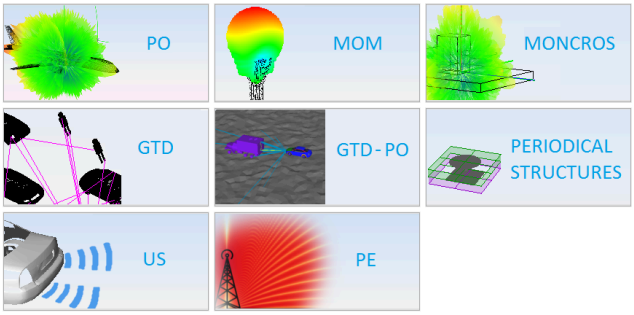


Figure 1355: New Project panel

Step 2 Create the bifilar antenna with its ground plane

Open the **Source** menu and navigate to click on **Primitive Antenna - Coaxial Feed - Helix - Multi-filar Helix** option to select an antenna from the primitives.

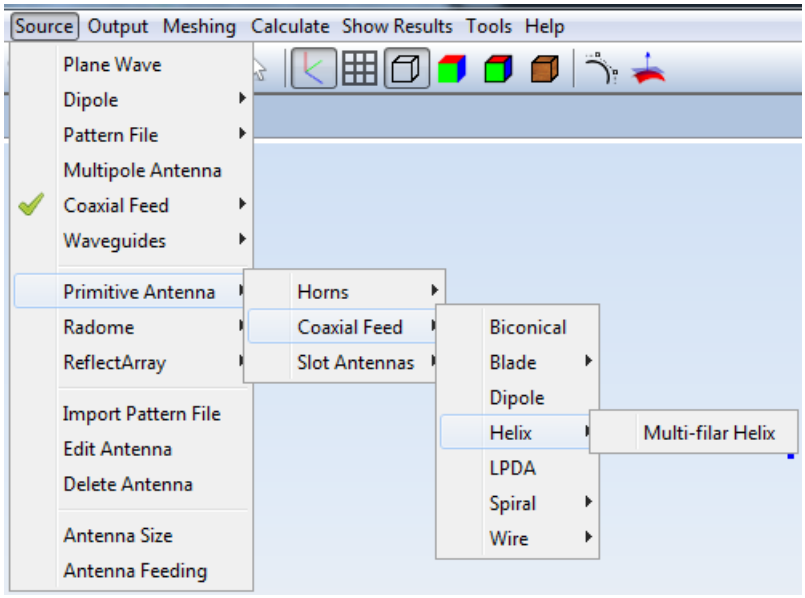


Figure 1356: Primitive Antenna menu

The Multi-filar Helix panel is open on right side. Set all parameters by default except the number of **Helixes**, which is set to 2. Then, this primitive provide two ports defined one in each Helix. Confirm the antenna definition by clicking on **Add** button. After that, close the **Multi-filar Helix** panel.

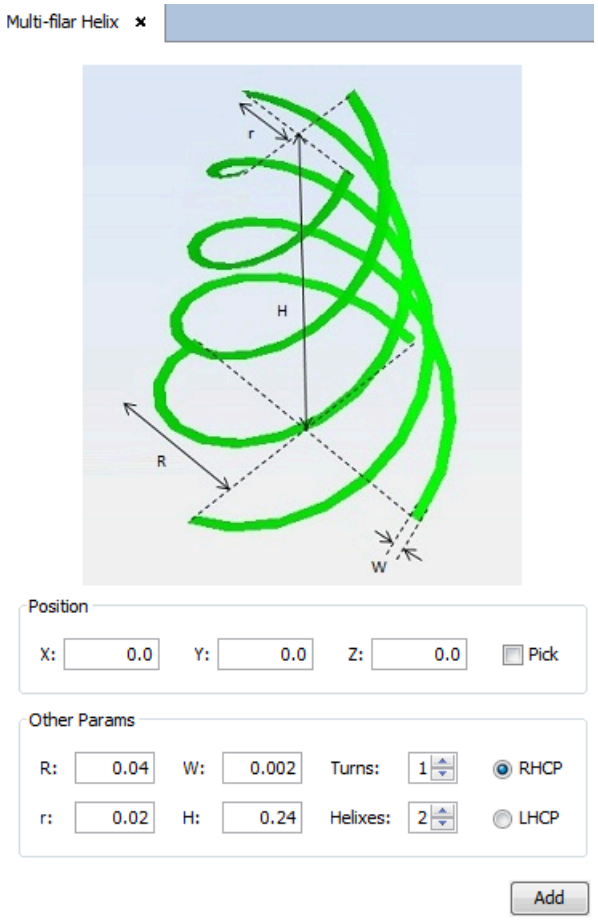


Figure 1357: Bi-filar helix definition

This antenna must be placed on a planar structure which acts as reference plane. Use the plane command to insert a plane under the helix antenna with the following parameters.

```
command> plane
First corner of plane [x y z]:  -0.1 -0.1 0.0
Plane size [width depth]:  0.2 0.2
command>
```

Figure 1358: Plane parameters

The resulting geometry is represented in next figure.

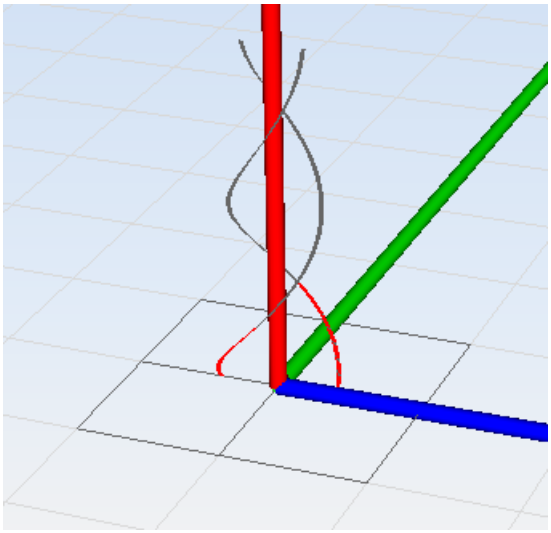


Figure 1359: Final geometric design

Step 3 Set Simulation Parameters

Click on **Simulation - Parameters** option on the menu bar. Set a **Frequency Sweep** and the **S-Parameters** as shown in the figure below. Remember clicking on **Save** button to confirm the changes.

Note that a large number of frequencies has been set in this example, so the simulation time may requires several minutes. You may reduce the number of **Samples** to speed-up the simulation time and generate the example with less resolution.

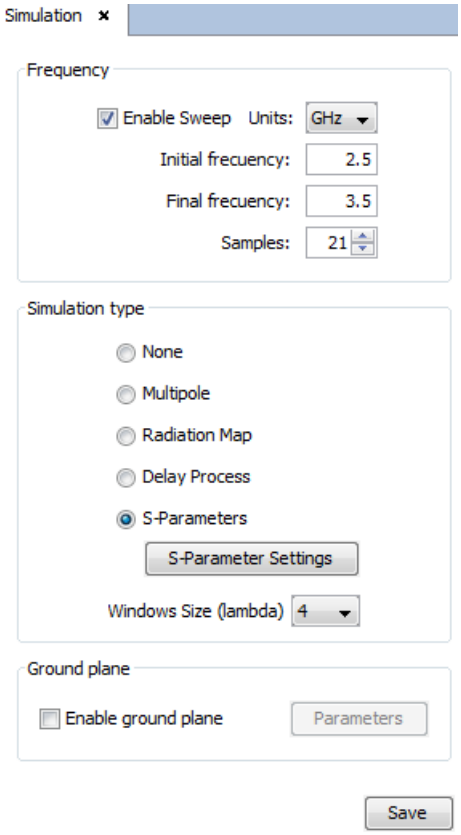


Figure 1360: Simulation Parameters panel

Step 4 Set Solver Parameters

Click on **Solver - Parameters** option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on **Save** button before going to next step.

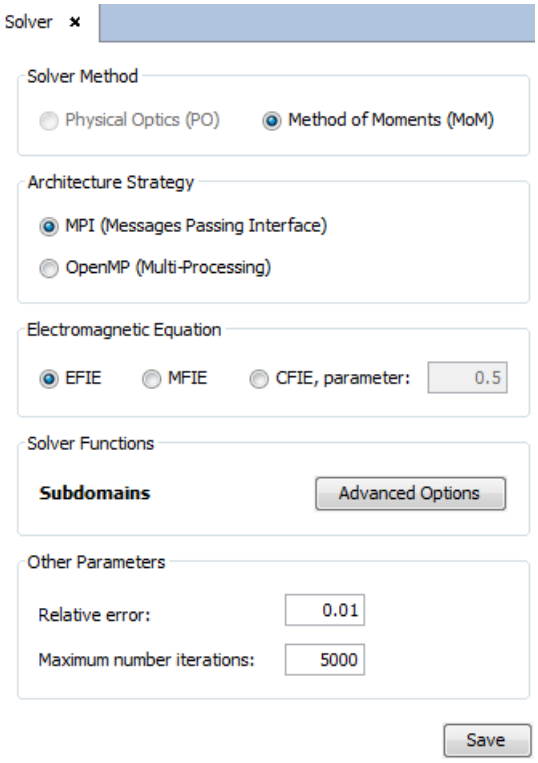


Figure 1361: Solver Parameters panel

Step 5 Meshing

Click on **Meshing - Create Mesh** option on the menu bar. To use the same mesh for all the frequency sweep, select the **Frequency**and specify the upper frequency of the range, **3.5** GHz. Select the available number of **Processors**, and then launch the meshing process by clicking on **Mesh** button.

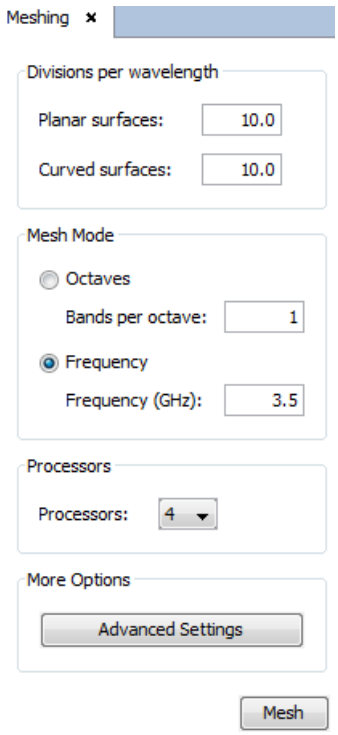


Figure 1362: Meshing parameters

After the end of the meshing process, the meshes may be visualized by clicking on **Meshing - Visualize Existing Mesh** and select the mesh within any frequency folder. Then, you can verify that the mesh show the electrical continuity between the helix filaments and the ground plane, as shown in next figure.

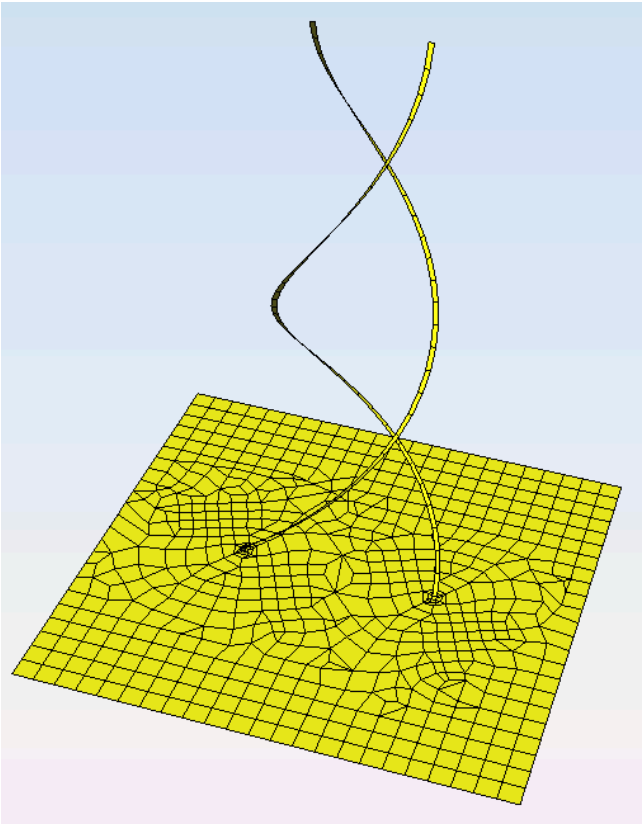


Figure 1363: Generated mesh

Step 6 Execute

Click on **Calculate - Execute** menu and launch the simulation with the available processors. Launch the simulation by clicking on **Calculate** button.

Step 7 Results

When the simulation process has finished, the S-Parameters are enabled within the results menu. Click on **Show Results** menu and **S-Parameters** option to visualize this parameters. The panel with the available options to visualize the S-Parameters is open on right side.

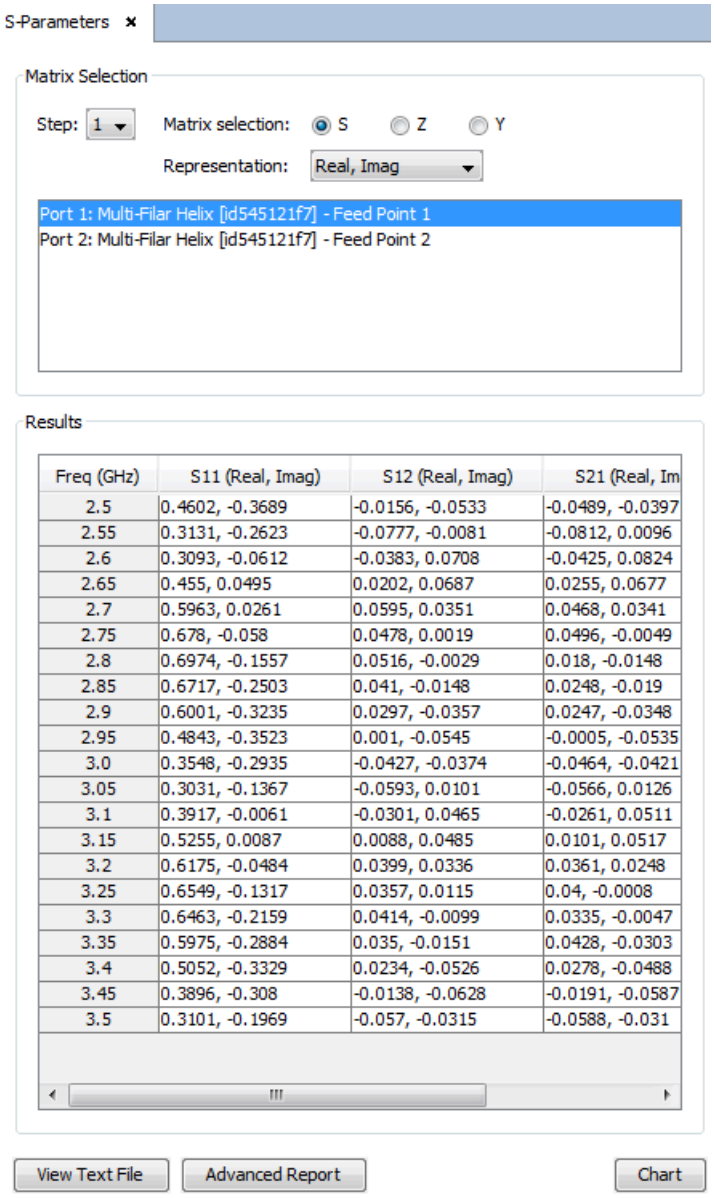


Figure 1364: S-Parameters parameters

Select the **S Matrix Representation** and the option **db, Phase** in **Representation** section. Then, select the column to represent in the **Results** table and then click on **Chart** button to add them to plot them.

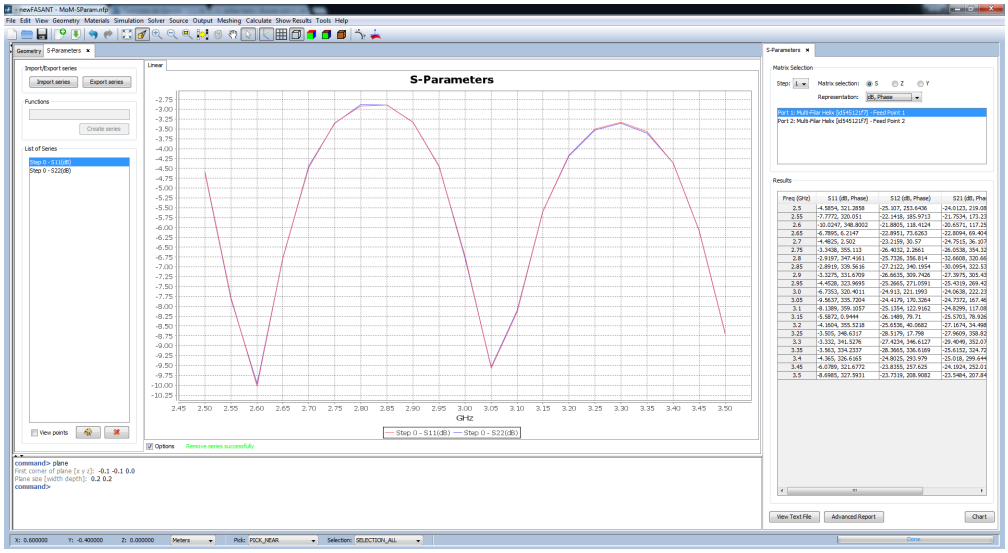


Figure 1365: S11 and S22 parameters plot

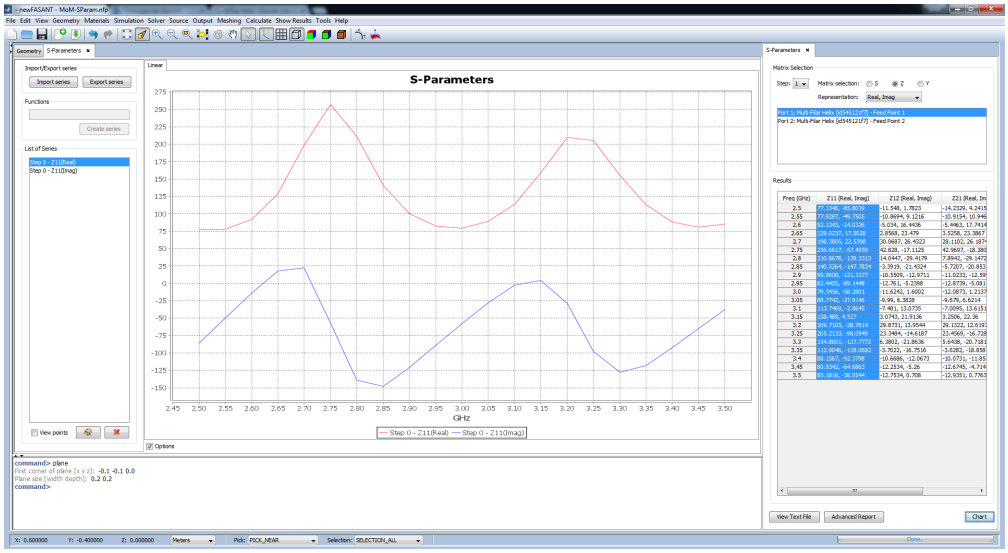


Figure 1366: Z11 parameters plot

Click on **Advanced Report** button, and select all the fields to be exported into a **html** file, which may be selected in the new **Advanced Report** window shown in next figure.

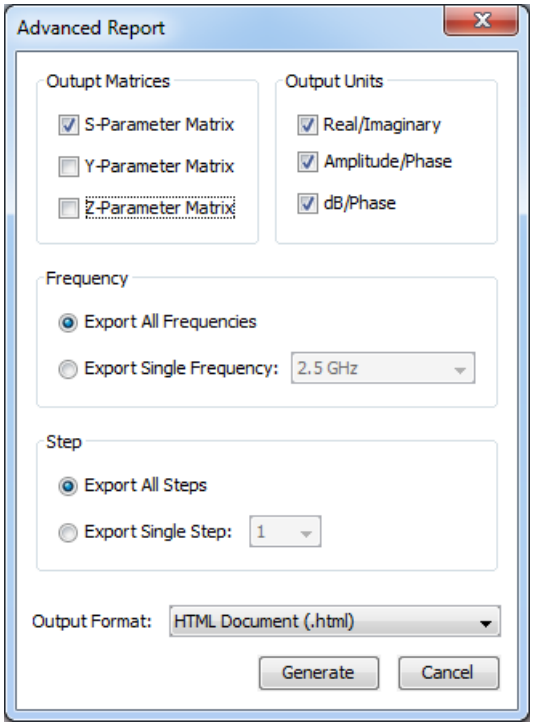


Figure 1367: Advanced Report window

Advanced Scattering Parameters Report

Step 1

S-Parameters

Frequency: 2.5E9

	Real/Imag	Amplitude/Phase	dB/Phase
S11	R = 0.4602 I = -0.3689	A = 0.5898 P = 321.2858	dB = -4.5854 P = 321.2858
S12	R = -0.0156 I = -0.0533	A = 0.0555 P = 253.6436	dB = -25.107 P = 253.6436
S21	R = -0.0489 I = -0.0397	A = 0.063 P = 219.0859	dB = -24.0123 P = 219.0859
S22	R = 0.4562 I = -0.3709	A = 0.5879 P = 320.889	dB = -4.6141 P = 320.889

Figure 1368: Advanced Report results for the first frequency

11.15.9 Example 9: Parameters, Frequency Sweep, Far and Near Field

This case explains how to generate a full example with most of options available in the tool geometric parameters, frequency sweep, primitive antennas, solver parameters, 3D radiation pattern, and observation entities for near field.

Step 1 Create a new MOM Project

Open newFASANT and select **File - New** option. Select **MOM** on the module selection.

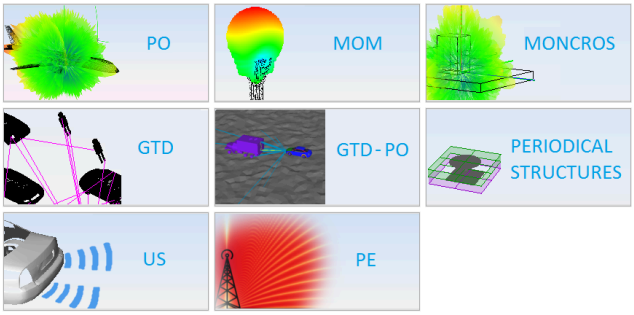


Figure 1369: New Project panel

Step 2 Create the geometry

A quadri-filar helix antenna on a box which is moving according to a parameter will be generated.

Create the geometric parameter to be used in the box later. Click on **Geometry - Parameters** menu and select the **Define Parameters** option. Then, click on **Add parameter** button and the new parameter added it is named **X** and its values ranges from **-0.25 to 0.25** with **11** equispaced samples, as shown in next figure. Remember clicking on **Save** button before closing the parameters panel.

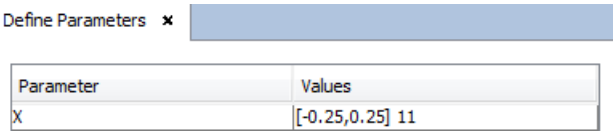


Figure 1370: Parameter X defined

Now, create a box which is moving along the X axis according to the defined parameter. Tip **box** in the command line and press **Enter** key or click on **Geometry - Solid** menu and select the **Box** option. Define the box with the same arguments than shown below:

```
command> box
First corner of base [x y z]: X-0.5 -0.25 -0.1
Base size [width depth]: 1 0.5
Height [double]: 0.2
command>
```

Figure 1371: Box parameters

As the box definition contains an operation over the **X** parameter, a new auxiliary parameter has been added, so it is possible to use again this operation by repeating it or by using the name of this auxiliary parameter (in this example, it is named $1 but it may change in other cases). Verify that the auxiliary parameter has been correctly added within **Geometry - Parameters - Define Parameters** panel.

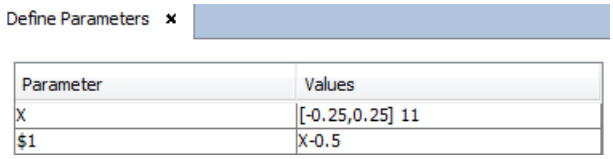


Figure 1372: Geometric parameters with an auxiliary parameter

Now, an antenna will be placed on the top of the box in a fixed position. That means that only the box is moving. It may be useful to make a parametric study to find which is the optimal ground plane position for the antenna behaviour. Click on **Source - Primitive Antenna - Coaxial Feed - Helix - Multi-filar Helix** menu, as shown in the following figure.

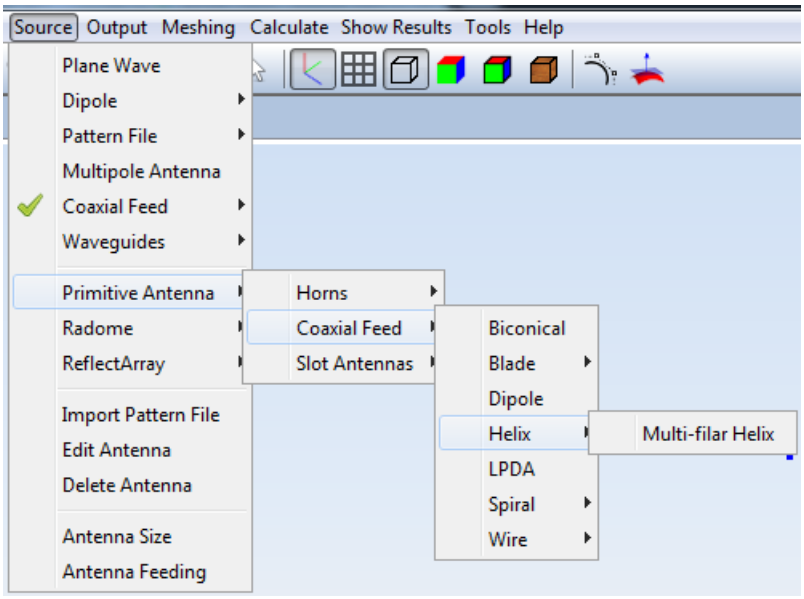


Figure 1373: Primitive Antenna menu

The Multi-filar Helix panel is open on right side. Set all parameters by default **Z** position, which is set to 0.1. Then, this primitive provide four ports defined one in each Helix. Confirm the antenna definition by clicking on **Add** button. After that, close the **Multi-filar Helix** panel.

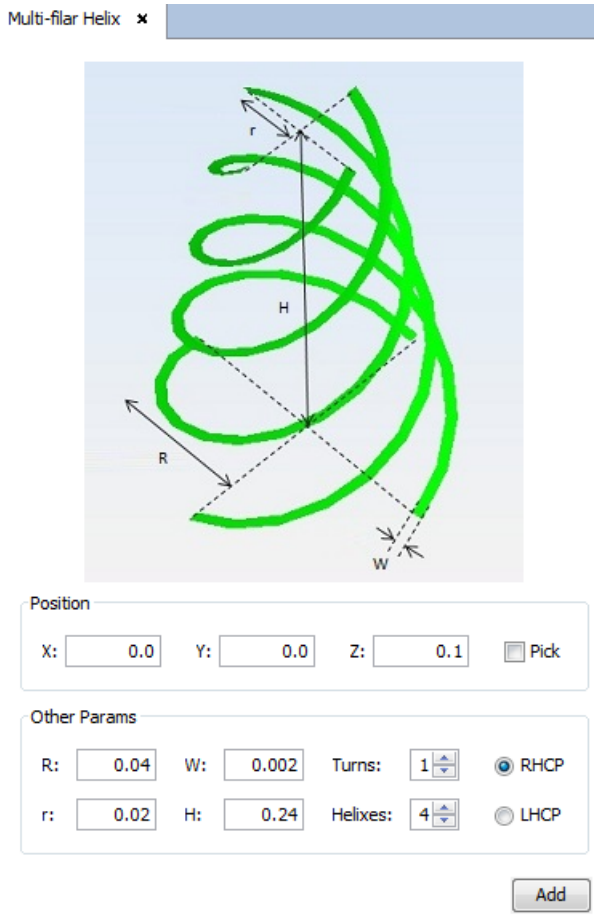


Figure 1374: Quadri-filar helix definition

This antenna must be placed on a planar structure which acts as reference plane, so the box performs this function. The resulting geometry is represented in next figure.

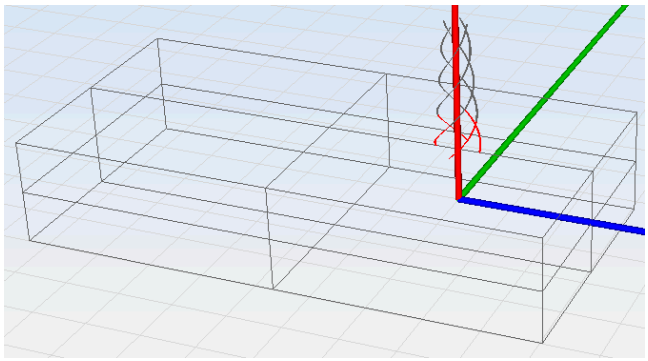


Figure 1375: Final geometric design

Step 3 Set Simulation Parameters

Click on **Simulation - Parameters** option on the menu bar. Set a **Frequency Sweep** and the **None** as shown in the figure below. Remember clicking on **Save** button to confirm the changes.

Note that a large number of frequencies has been set in this example, so the simulation time may requires several minutes due to the combination with the parametric steps. You may reduce the number of **Samples** to speed-up the simulation time and generate the example with less resolution.

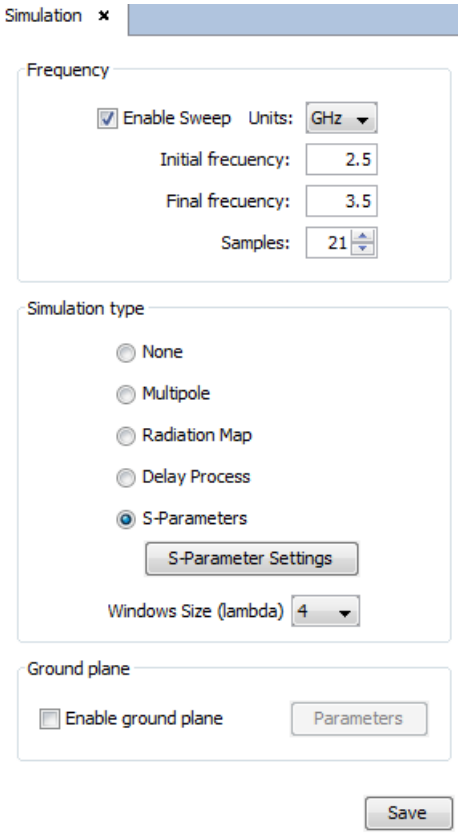


Figure 1376: Simulation Parameters panel

Step 4 Set Solver Parameters

Click on **Solver - Parameters** option on the menu bar. Set the OpenMP Architecture Strategy, and then verify that all the parameters are defined by default, as shown in figure below.

Note that **EFIE** is set even having a box which is closed and could be analyzed with **CFIE**. As it is not an electrically large case and the convergence is good, we can solve it directly with **EFIE**. However, if **CFIE** is selected, the user must assign **EFIE** to the helix antenna within the **Solver - Advanced Options - Specify**.

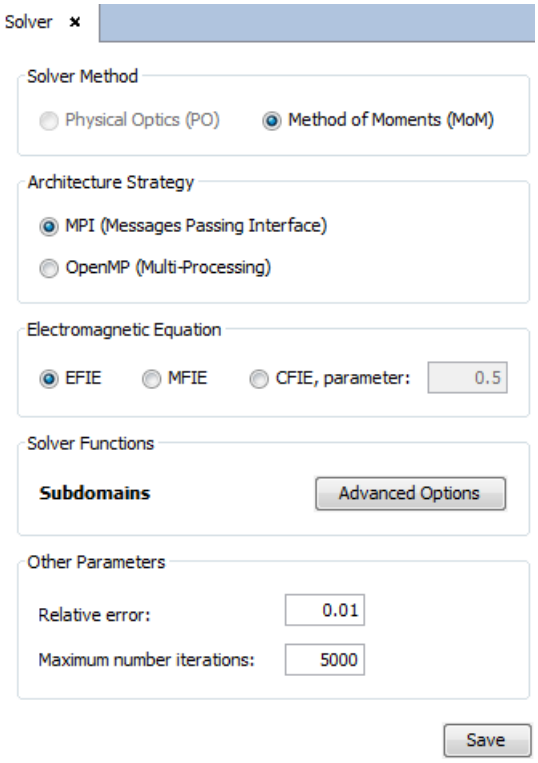


Figure 1377: Solver Parameters panel

This example will be solved with **Direct Solver**. To do that, click on **Advanced Options**, and set the **Max. Number of unknowns for Direct Solver** with a value which is higher than the number of unknowns of the problem. As it is a small case, a value of 10000 is high enough for considering the **Direct Solver**. Note that the **Compute 3D Pattern** option is enabled, as we want to generate the 3D radiation pattern in the simulation process. Click on **Save** button to close the **Solver Advanced Options** window by confirming the changes.

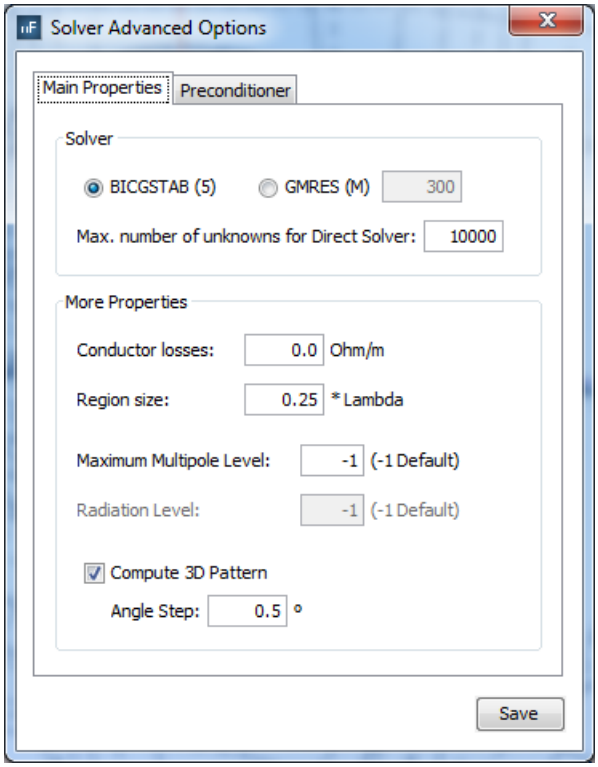


Figure 1378: Solver Advanced Options

Click on **Save** button before going to next step, and the **Solver** panel may be already closed.

Step 5 Output

Both **Far** field and **Near** field results are considered in this example.

Regarding the **Far** field, click on **Output** menu and select the **Observation Directions** option. Set the default parameters in the **Observation Directions** panel, and click on **Save** button before closing it.

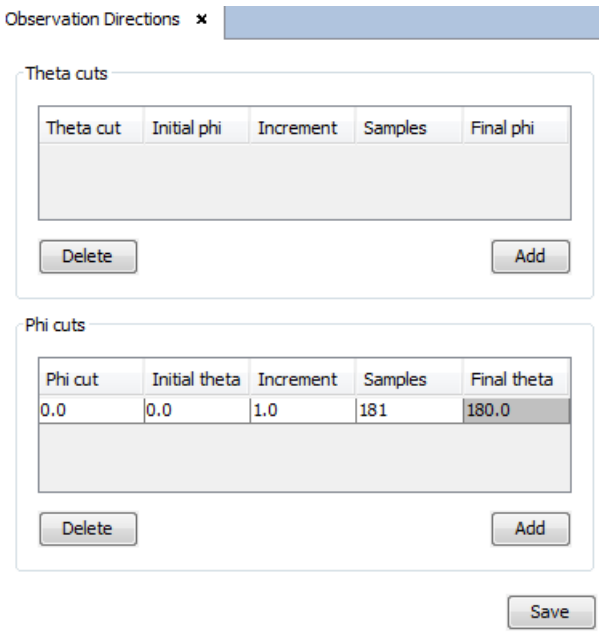


Figure 1379: Observation directions

On the other hand, a plane and a progressive radial sphere will be generated as observation points. Click on **Output** menu and select the **Observation Points** option to open the **Observation Points** panel.

First, within the **Geometry Types** section select the **Plane** on the list and click on **Add** button to open the **New Plane** window. Set the values as shown in figure below and click on **OK** button to add the observation plane.

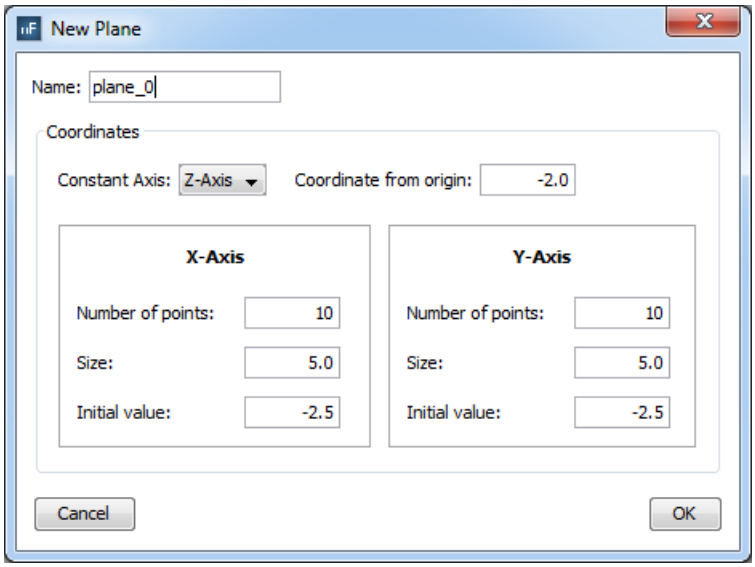


Figure 1380: Observation plane

Next, within the **Geometry Types** section select the **Sphere** on the list and click on **Add**button to open the **New Sphere** window. Set the values as shown in figure below and click on **OK**button to add

the observation sphere. Note that several **Radius Samples** has been set, so we will generate multiple spheres in the same entity and the **Radial Points** results will be enabled after the simulation process.

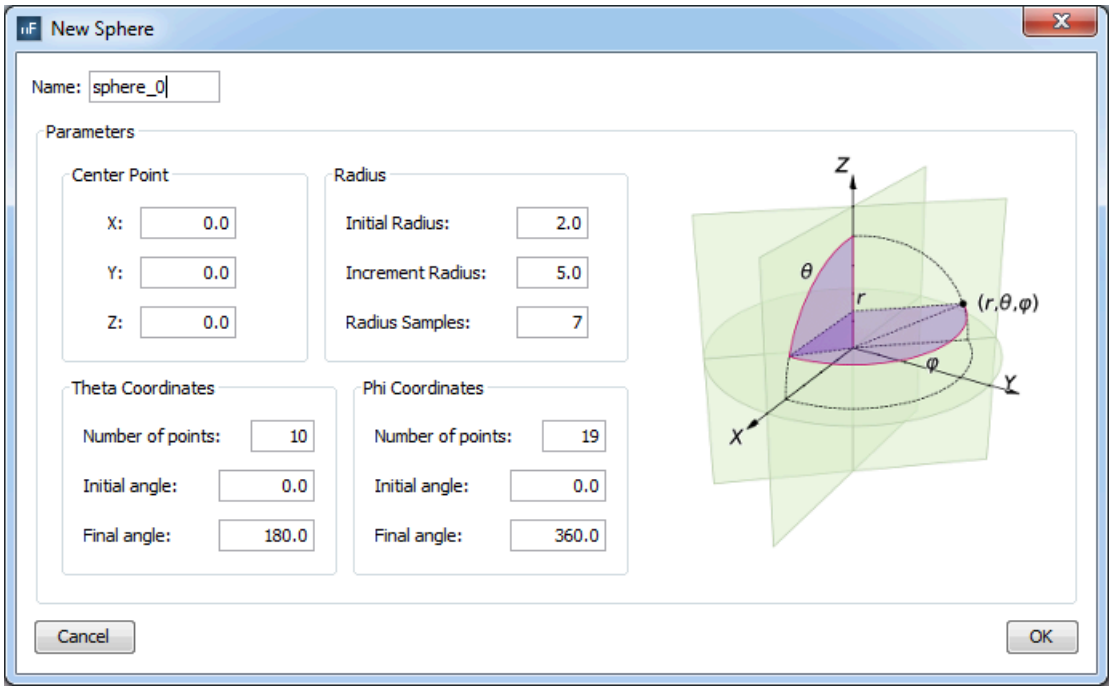


Figure 1381: Observation sphere

Having inserted the plane and the sphere as observation points entities, they are shown together the geometry in the **Geometry** panel. Then, you can close the Observation Points panel.

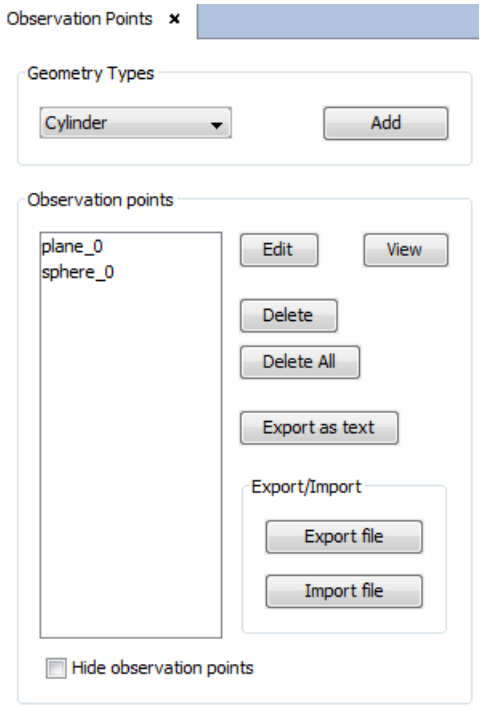


Figure 1382: Observation points panel

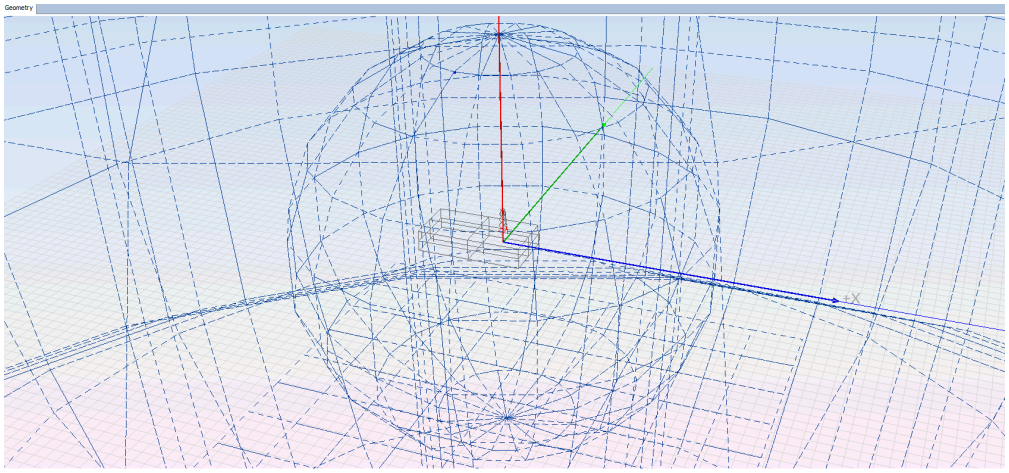


Figure 1383: Geometry together the observation points entities

Step 6 Meshing

Click on **Meshing - Create Mesh** option on the menu bar. To use the same mesh for all the frequency sweep, select the **Frequency**and specify the upper frequency of the range, **0.9** GHz. Select the available number of **Processors**, and then launch the meshing process by clicking on **Mesh** button.

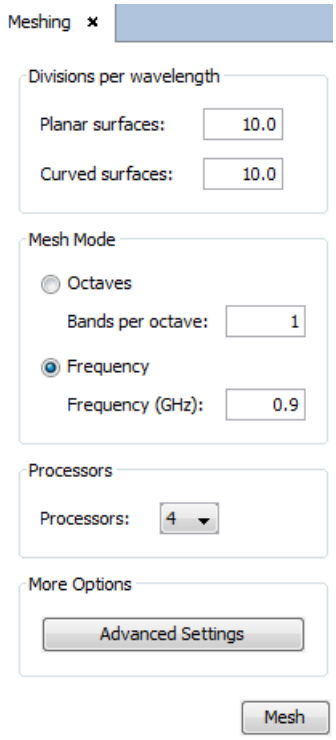


Figure 1384: Meshing parameters

After the end of the meshing process, the meshes may be visualized by clicking on **Meshing - Visualize Existing Mesh** and select the mesh in any frequency folder within the desired step folder. Then, you can verify that the mesh show the electrical continuity between the helix filaments and the box, as shown in next figure.

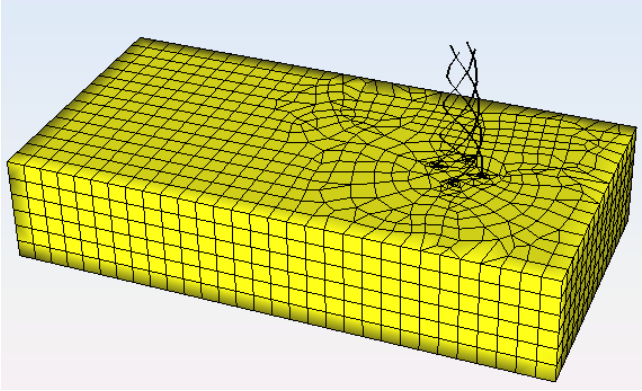


Figure 1385: Generated mesh (step 0)

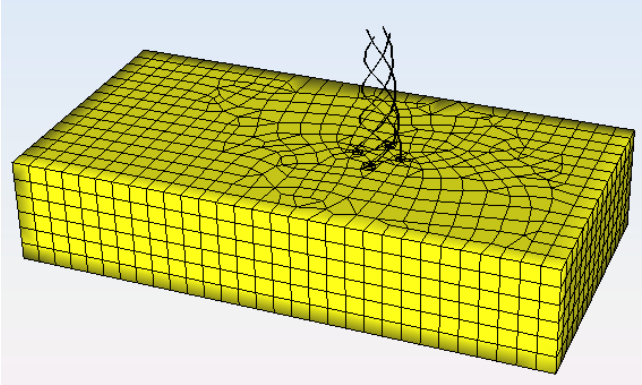


Figure 1386: Generated mesh (step 3)

Step 7 Execute

Click on **Calculate - Execute** menu and launch the simulation with the available processors. Launch the simulation by clicking on **Calculate** button.

Step 8 Results

When the simulation process has finished, most of options are enabled within the results menu.

Click on **Show Results - Far Field** menu and **View Cuts** option. In the figure below, we compare the **Total Gain** of the only considered far field cut versus all the geometric steps. To do that, select the **Gain Lineal - ETotal Gain**, and then click on **Add Series** button after selecting every step.

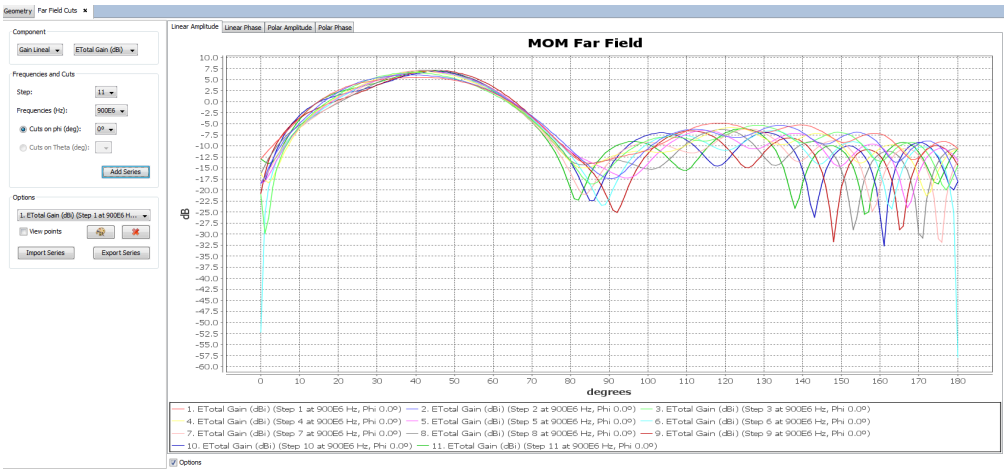


Figure 1387: Far field cuts for every parametric step

Click on **Show Results - Radiation Pattern** menu and **View Cuts By Step** option. In the next figure, a comparison of a far field direction (obtained from the 3D radiation pattern) versus the parametric step is shown. To do that, select the **Circular**, and then click on **Add Series** button after selecting each component.

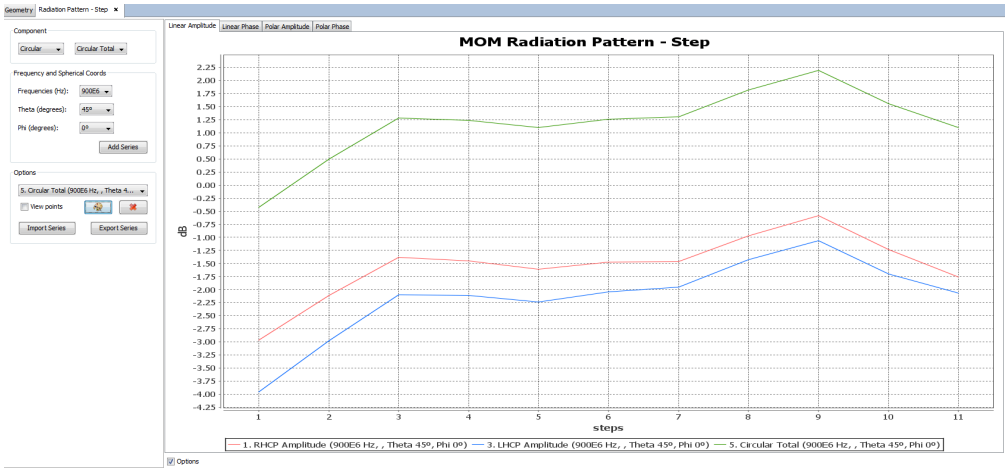


Figure 1388: Far field cuts for every parametric step

To visualize the 3D radiation pattern, click on **Show Results - Radiation Pattern** menu and **View 3D Pattern** option. The diagram of the first parametric step at 800 MHz is depicted in next figure.

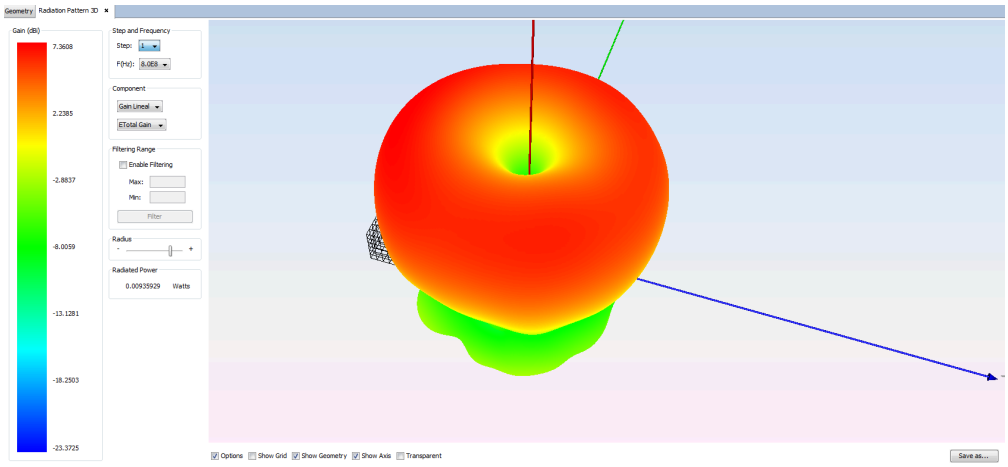


Figure 1389: Far field 3D pattern

The next figure shows the current distribution on the structure at the second parametric step. To visualize it, click on **Show Results - View Currents** menu.

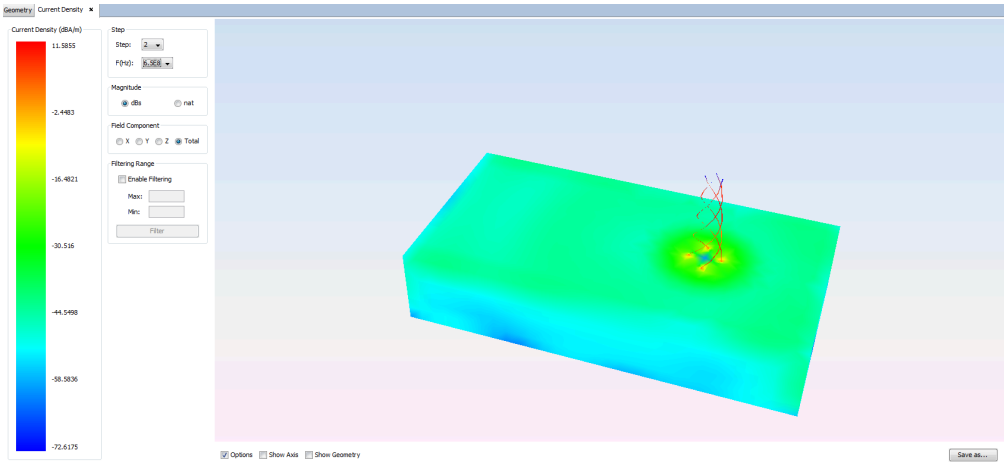


Figure 1390: Current density distribution

Far field 3D pattern

The charges density on the structure is represented in next figure. To visualize it, click on **Show Results - View Charges** menu. Note that the **Enable Filtering** has been enabled.

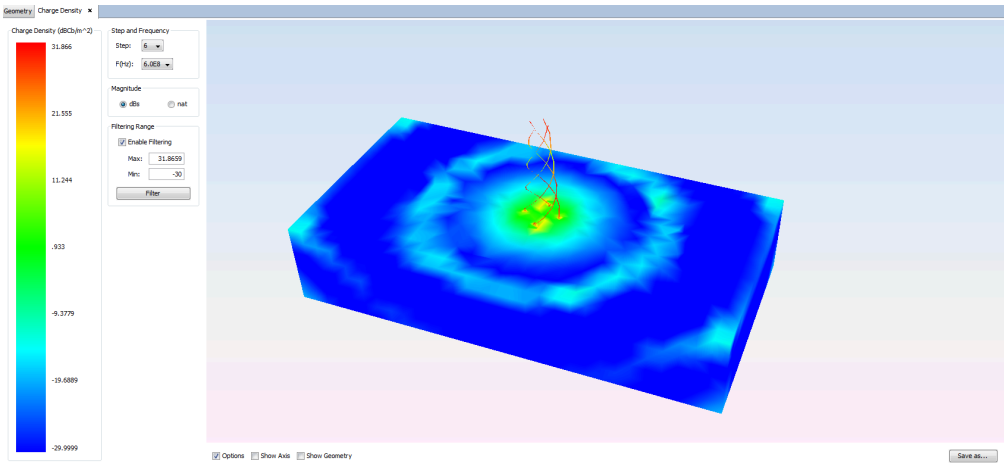


Figure 1391: Charge density distribution

The next figure shows the near field distribution on the observation plane. To visualize it, click on **Show Results - Near Field** menu and select the first option, **View Near Field Diagram**. Then, select the **Step**, **Frequency** and **Observation entity** in the **Near Field 3D** panel and click on **OK** button to open the results.

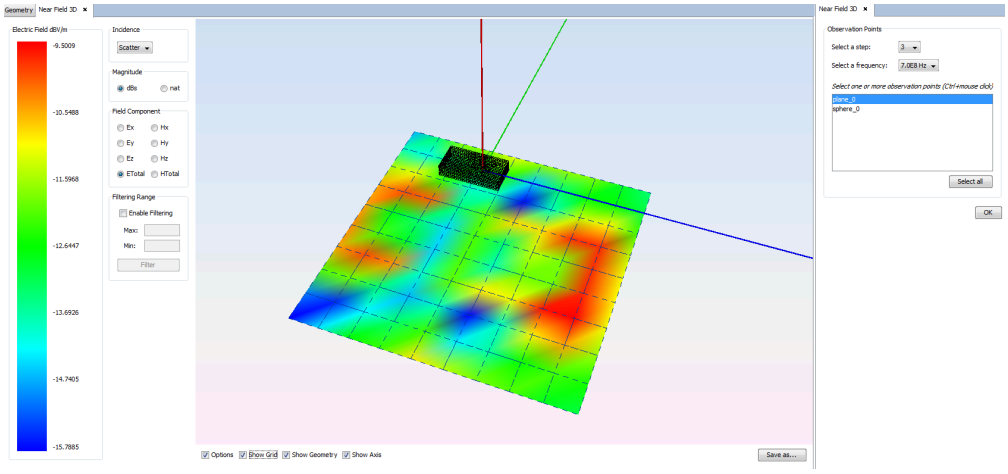


Figure 1392: Near field distribution on the observation plane

The last results represented in this example is the **Near field** obtained from the observation radial spheres in a near field direction at all the simulated frequencies. To do that, Click on **Show Results - Near Field** menu and select the **View Radial Points** option. To do that, select the **Theta** and **Phi** directions, and then click on **Add Series** button after selecting frequency step.

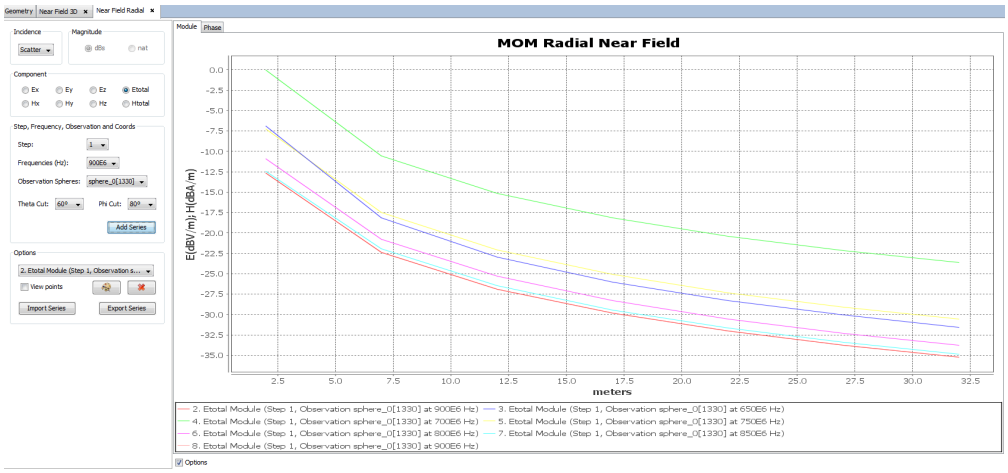


Figure 1393: Radial near field comparison at every frequency

11.15.10 Examples with Different Pointing Algorithms

Example 1: 16x16 Short Dipole Array with Uniform Algorithm

This case explains how to use the uniform algorithms to calculates the pointing parameters in a bidimensional short dipole array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

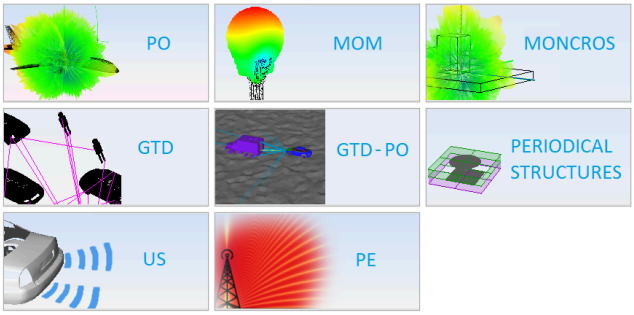


Figure 1394: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1395: Simulation panel

Step 3 Create the array.

Click on **Source - Dipole - Dipole Array** to create an array of **16x16** electrical short dipoles, with a spacing of **0.04 m**, that are oriented following the **y-axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz**.

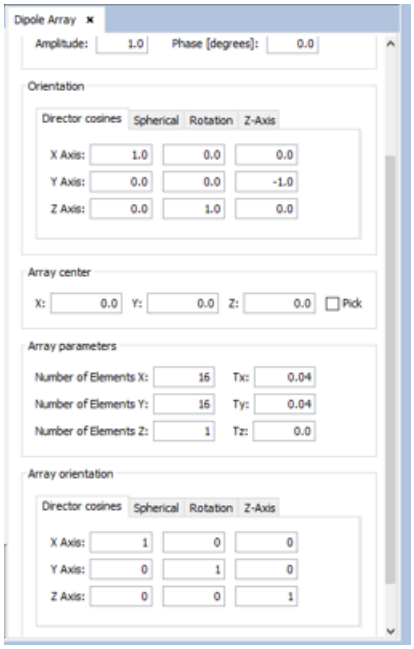


Figure 1396: Dipole Array panel

Step 4 Feed the array.

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

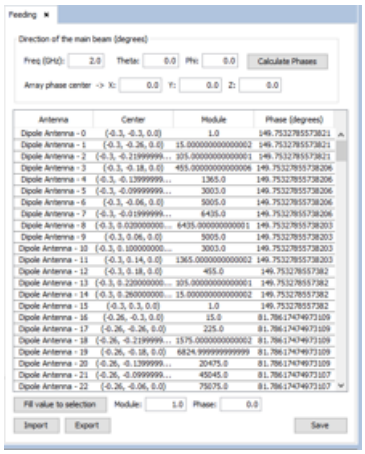


Figure 1397: Antenna Feeding panel

This is the default setting. To use the **uniform algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE** To use the bidimensional uniform function it is needed to download both the bi-dimensional and the unidimensional functions.

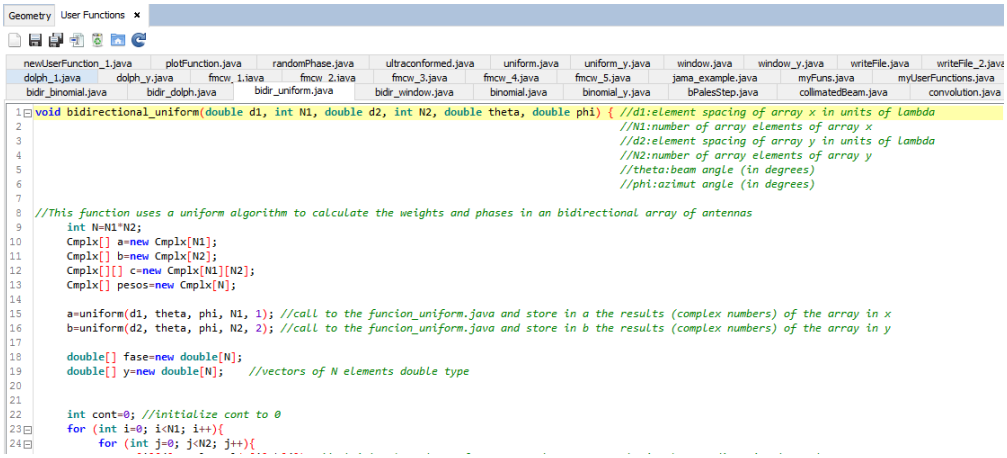


Figure 1398: Bi-dimensional Uniform function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the newFASANT directory.

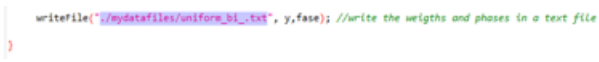


Figure 1399: Bidimensional Uniform function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

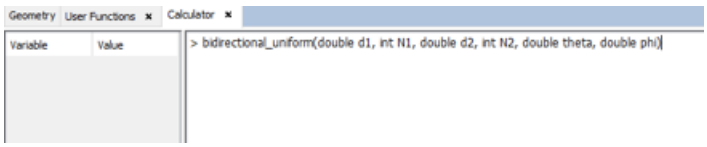


Figure 1400: Calculator panel

The parameters to set are:

- **d1** element spacing of the array in the x-axis in units of lambda
- **N1** number of array elements of the array in the x-axis
- **d2** element spacing of the array in the y-axis in units of lambda
- **N2** number of array elements of the array in the y-axis
- **theta** beam angle, in degrees
- **phi** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=45º** and **phi=0º** are selected as an example.

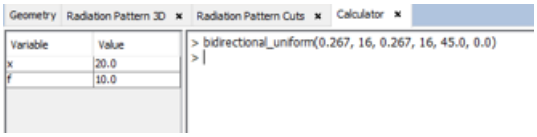
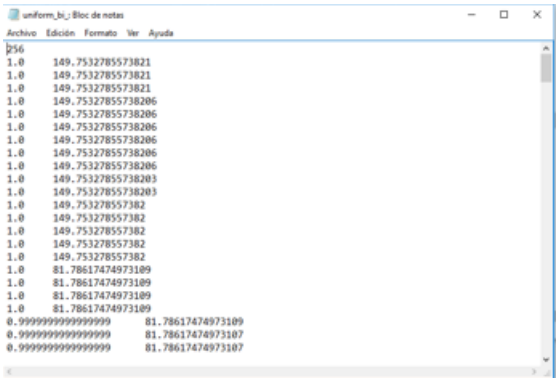


Figure 1401: Calculator panel

The text file will be automatically generated in the **mydatafiles** folder.


uniform_bi_: Bloc de notas
Archivo Edición Formato Ver Ayuda
256
1.0 149.7532785573821
1.0 149.7532785573821
1.0 149.7532785573821
1.0 149.75327855738206
1.0 149.75327855738206
1.0 149.75327855738206
1.0 149.75327855738206
1.0 149.75327855738206
1.0 149.75327855738206
1.0 149.75327855738203
1.0 149.75327855738203
1.0 149.75327855738203

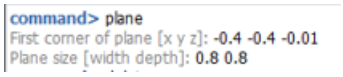


Figure 1404: Plane parameters

View of the dipole array.

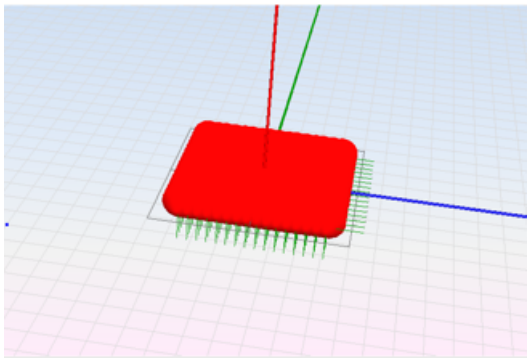


Figure 1405: Array view

Step 6 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

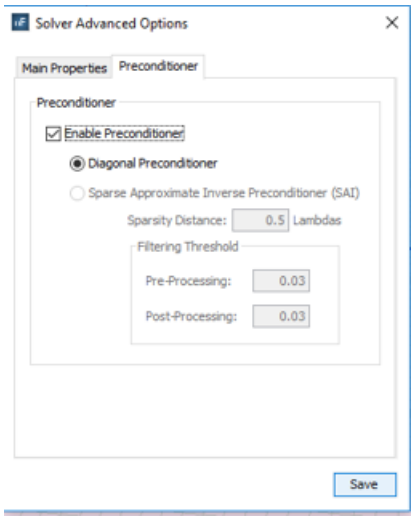


Figure 1406: Solver Advanced Options panel

Step 7 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

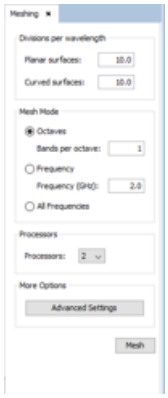


Figure 1407: Meshing panel

Click on **Mesh**.

Step 8 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

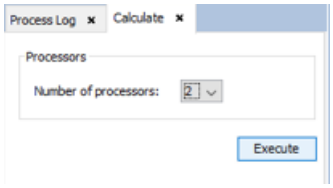


Figure 1408: Calculate panel

Step 9 Show Results.

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

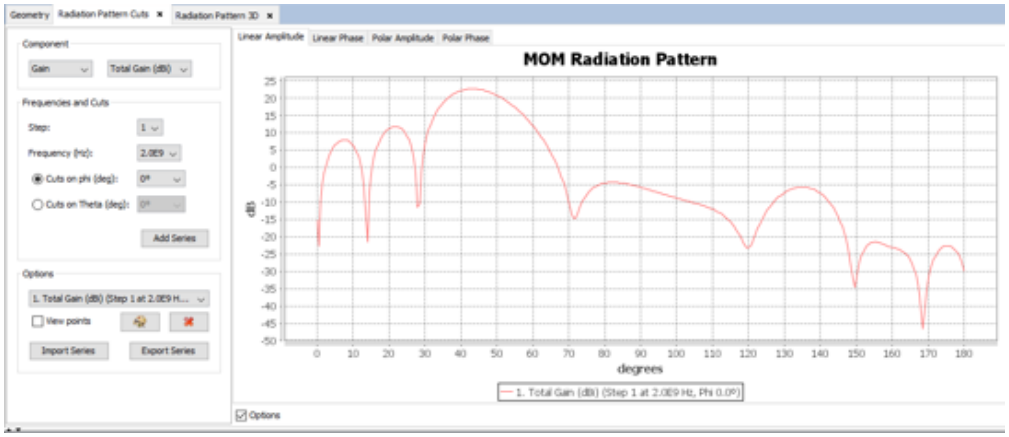


Figure 1409: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

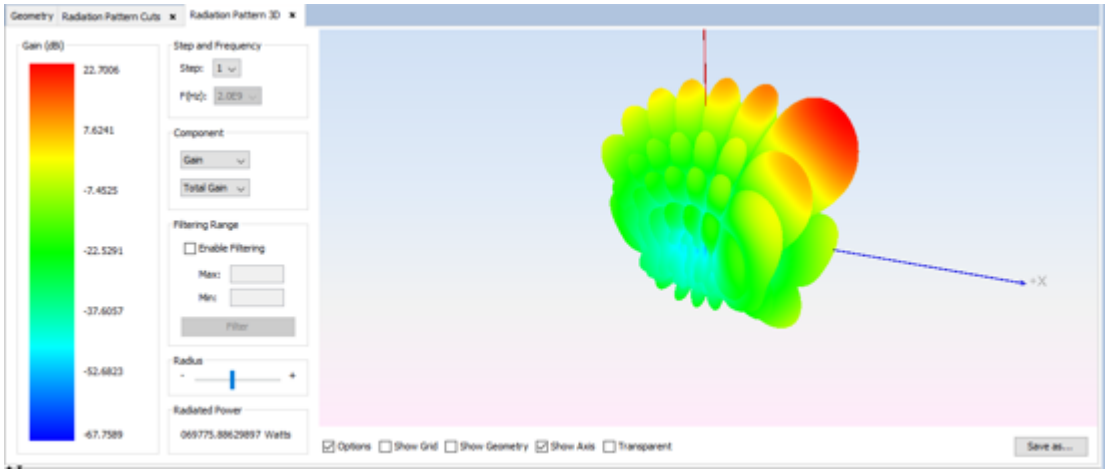


Figure 1410: Radiation Pattern 3D

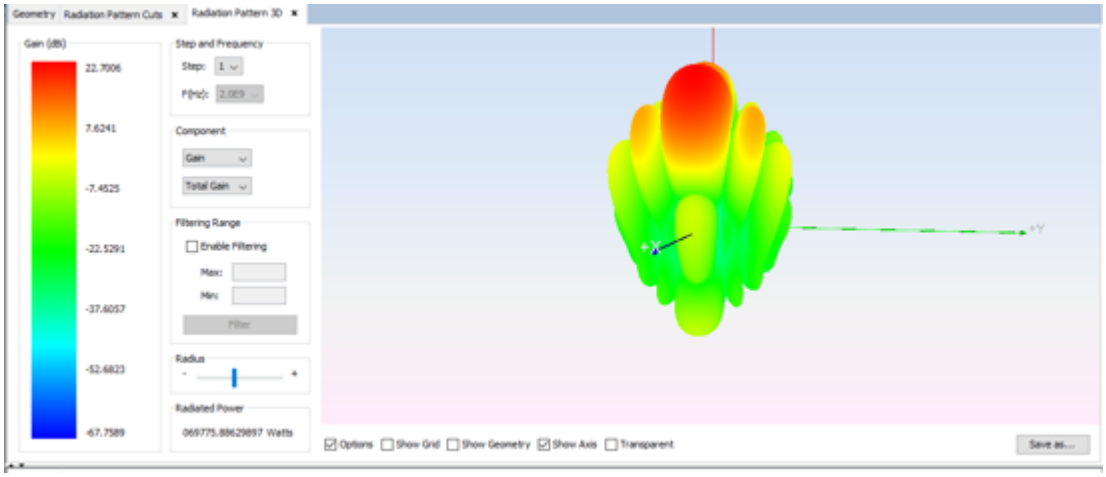


Figure 1411: Radiation Pattern 3D

Example 2: 20x20 Pattern File Array with Uniform Algorithm

This case explains how to use the uniform algorithms to calculates the pointing parameters in a bidimensional pattern file array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New option.**

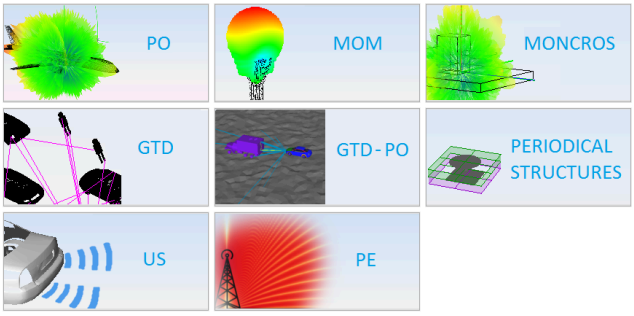


Figure 1412: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1413: Simulation panel

Step 3 Import the pattern file

Select **Source - Import Pattern File** and click on the **Import** button.

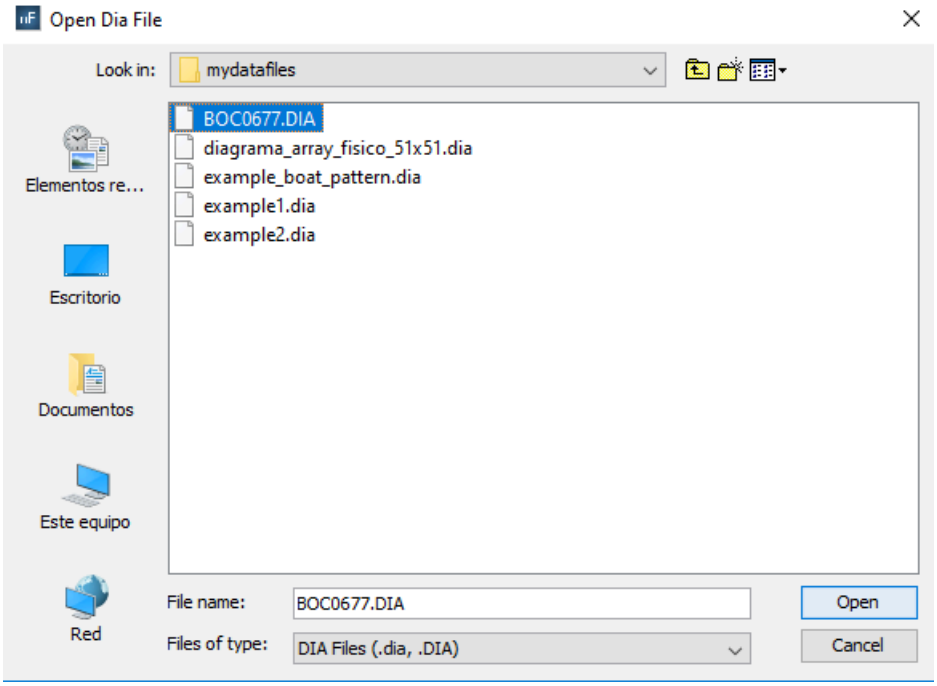


Figure 1414: Open DIA File panel

The file is now imported.

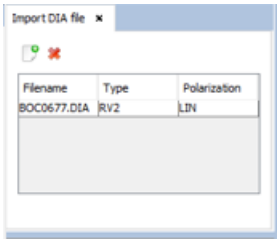


Figure 1415: Import DIA File panel

Step 4 Create the array

Click on **Source - Pattern File - Pattern File Array** to create an array of **20x20** pattern file dipoles, with a spacing of **0.04m**, that are oriented following the **z -axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz.**

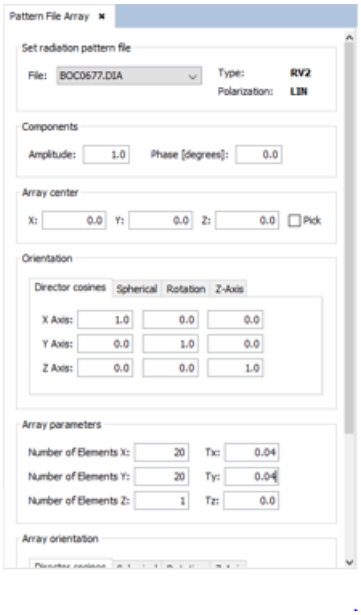


Figure 1416: Pattern File Array panel

Step 5 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

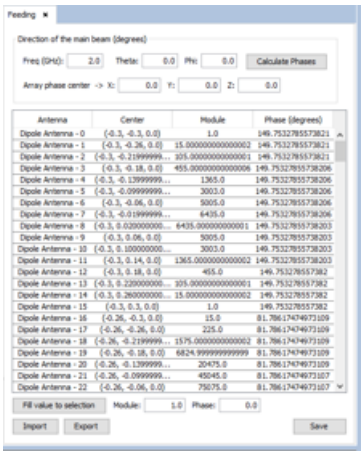


Figure 1417: Antenna Feeding panel

This is the default setting. To use the **uniform algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE** To use the bidimensional uniform function it is needed to download both the bidimensional and the unidimensional functions.

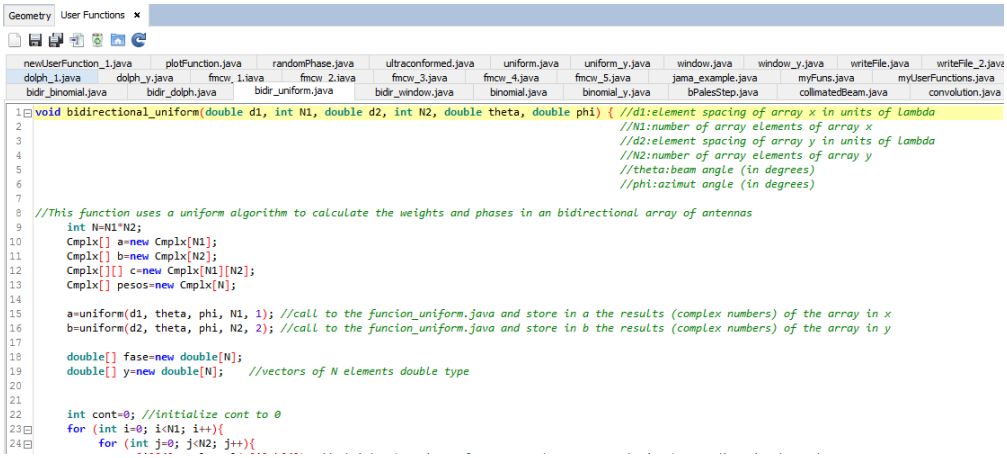


Figure 1418: Bidimensional Uniform function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the newFASANT directory.

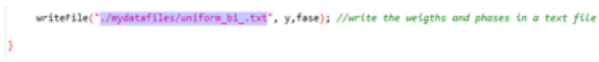


Figure 1419: Bidimensional Uniform function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

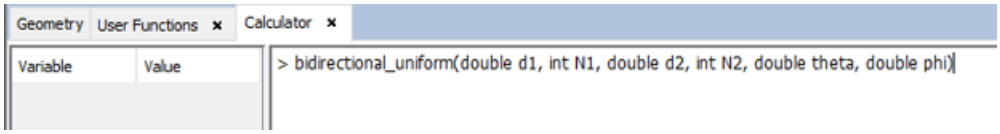


Figure 1420: Calculator panel

The parameters to set are:

· **d1** element spacing of the array in the x-axis in units of lambda

· **N1** number of array elements of the array in the x-axis

· **d2** element spacing of the array in the y-axis in units of lambda

· **N2** number of array elements of the array in the y-axis

· **theta** beam angle, in degrees

· **phi** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=20º** and **phi=0º** are selected as an example.

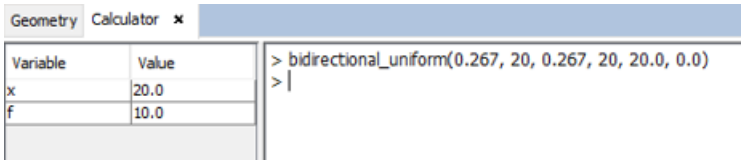


Figure 1421: Calculator panel

The text file will be automatically generated in the **mydatafiles** folder.

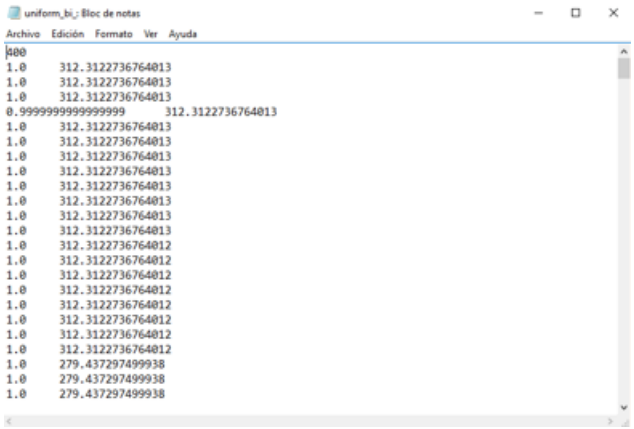


Figure 1422: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **uniform algorithm**, click on **Import.**

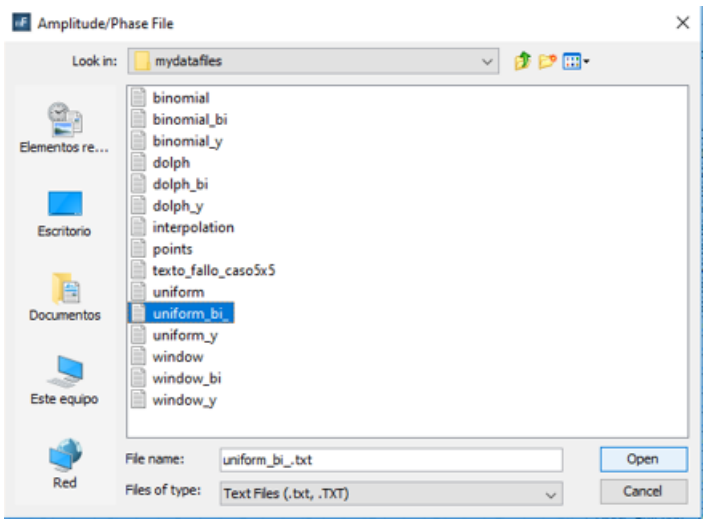


Figure 1423: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 6 Create ground plane.

In order to avoid unwanted radiation to go below the array, create a ground plane using the **plane** command, or **Geometry - Surface - Plane**. The array is situated in the XY plane with **z=0**, so the z coordinate has to be negative.

```
command> plane
First corner of plane [x y z]: -0.5 -0.5 -0.05
Plane size [width depth]: 1.0 1.0
command>
```

Figure 1424: Plane parameters

View of the pattern file array.

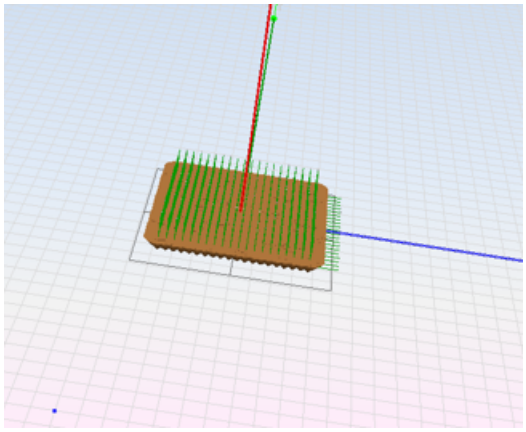


Figure 1425: Array view

Step 7 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

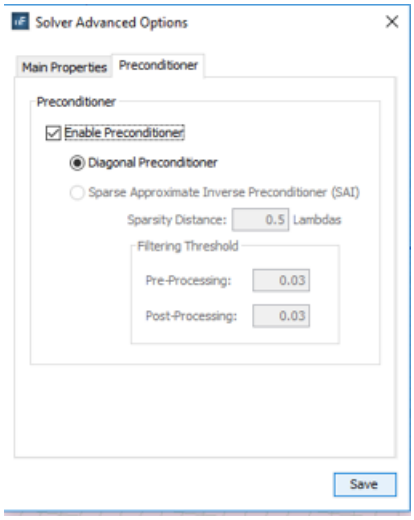


Figure 1426: Solver Advanced Options panel

Step 8 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

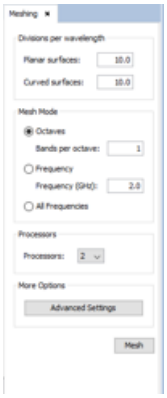


Figure 1427: Meshing panel

Click on **Mesh**.

Step 9 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

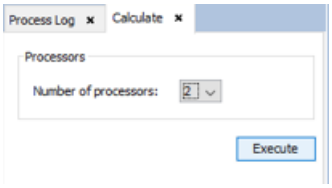


Figure 1428: Calculate panel

Step 10 Show Results

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

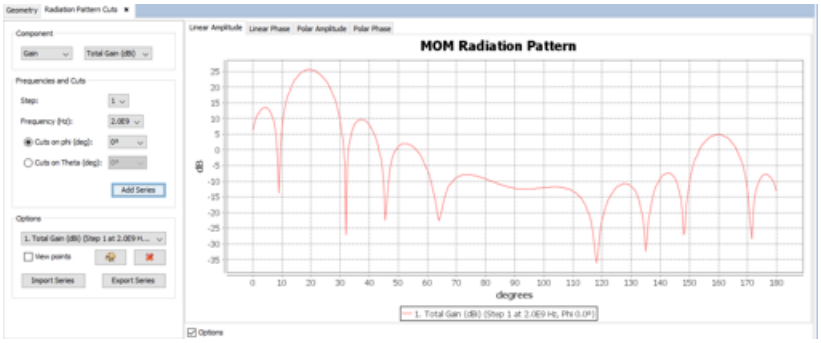


Figure 1429: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

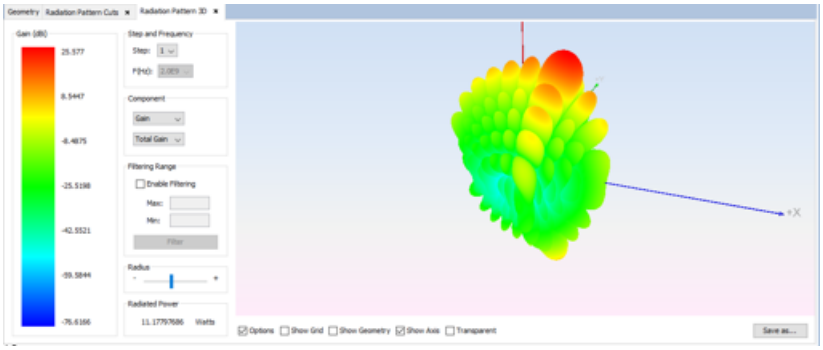


Figure 1430: Radiation Pattern 3D

Example 3: 14x14 Coaxial Feed Array with Uniform Algorithm

This case explains how to use the uniform algorithms to calculates the pointing parameters in a bidimensional coaxial feed array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New option.**

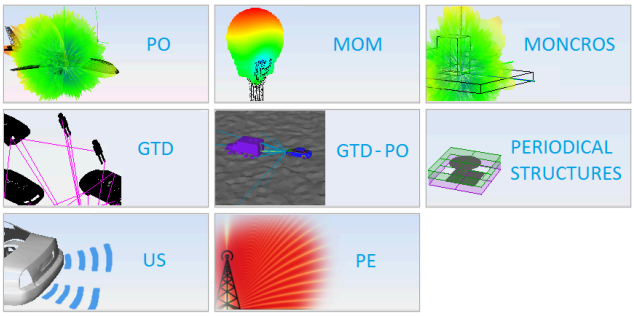


Figure 1431: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1432: Simulation panel

Step 3 Create the array

First, select 'milimeters' on units list on the bar at the bottom of the main window.

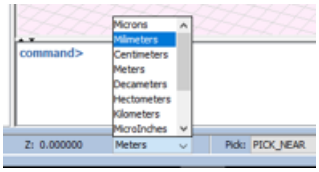


Figure 1433: The first element is created using Source - Coaxial Feed - Add Feed Point.

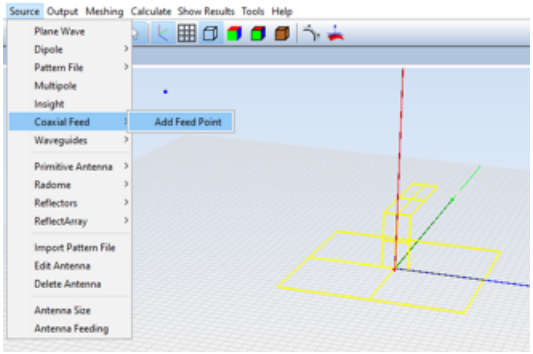


Figure 1434: Add feed point

Select the surfaces and click on **Add**.

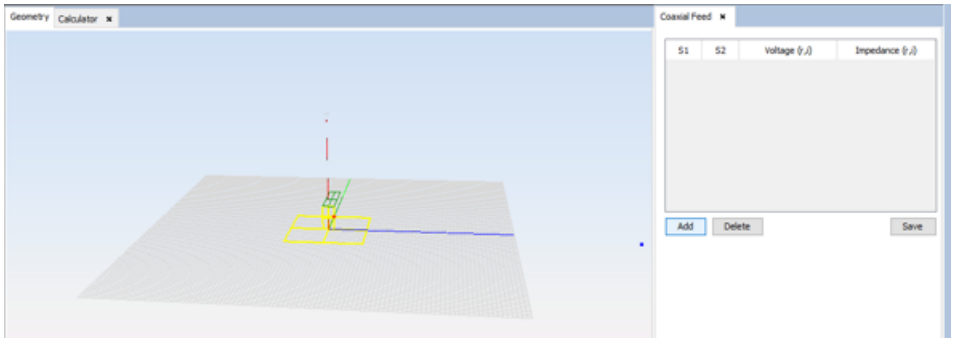


Figure 1435: Coaxial Feed panel

Then save the feed point.

Use the **array** command and enter the characteristics of the array.

```
command> array
Select objects on screen (Press enter when done):
Number of objects in X axis (default=1): 14
Number of objects in Y axis (default=1): 14
Number of objects in Z axis (default=1): 1
Select start point [x y z] (default=(0,0,0)): 0 0 0
Select end point [x y z]}: 2.10407 2.10407 0
command>
```

Figure 1436: Array parameters

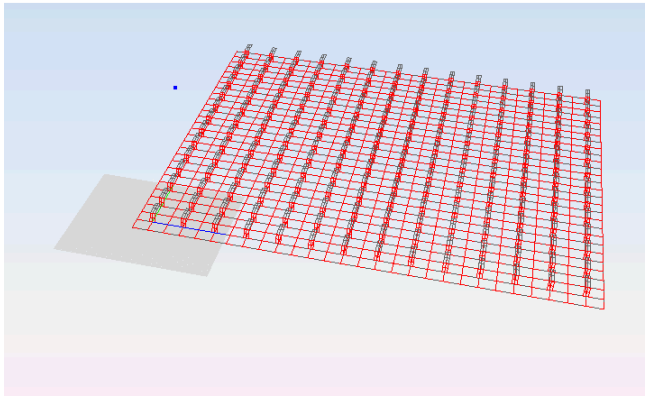


Figure 1437: Array view

Step 4 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

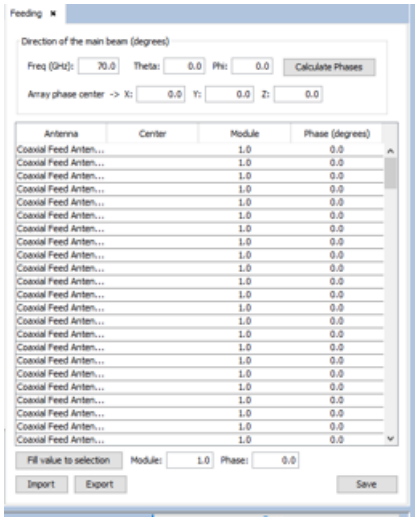


Figure 1438: Antenna Feeding panel

This is the default setting. To use the **uniform algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE** To use the bi-dimensional uniform function it is needed to download both the bi-dimensional and the unidimensional functions.

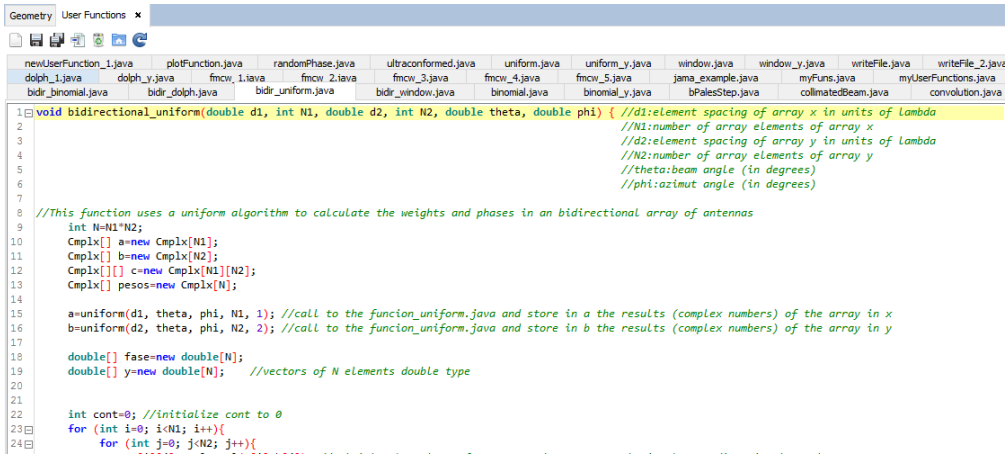


Figure 1439: Bid-imensional Uniform function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the newFASANT directory.

Figure 1440: Bi-dimensional Uniform function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

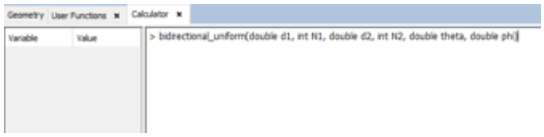


Figure 1441: Calculator panel

The parameters to set are:

· **d1** element spacing of the array in the x-axis in units of lambda

· **N1** number of array elements of the array in the x-axis

· **d2** element spacing of the array in the y-axis in units of lambda

· **N2** number of array elements of the array in the y-axis

· **theta** beam angle, in degrees

· **phi** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=30º** and **phi=0º** are selected as an example.

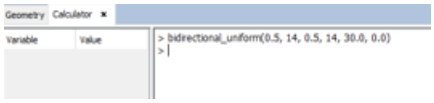


Figure 1442: Calculator panel

A spacing of **0.5** in units of lambdas is selected because is equivalent to a spacing of **2.104** millimeters at **70 GHz.**

The text file will be automatically generated in the **mydatafiles** folder.

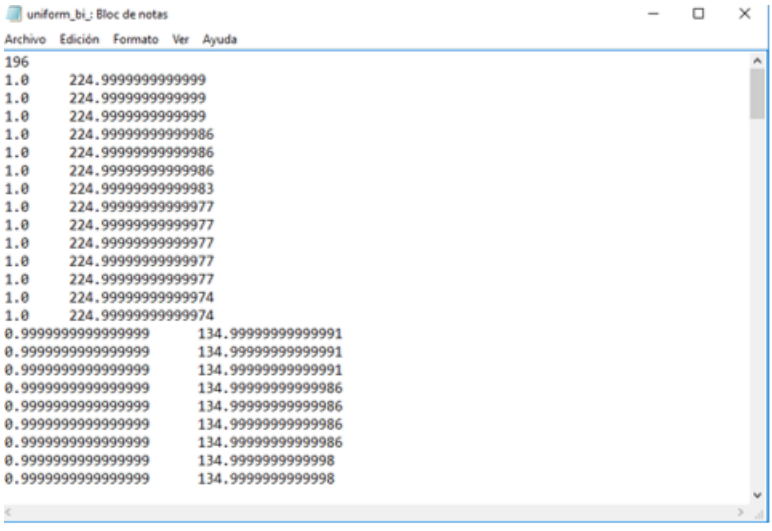


Figure 1443: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **uniform algorithm**, click on **Import.**

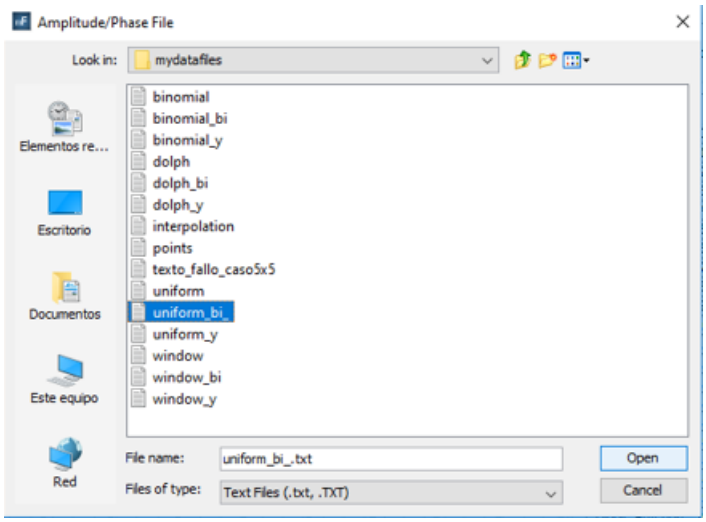


Figure 1444: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 5 Solver parameters

Select **Solver - Advanced Options** and set the parameters as shown.

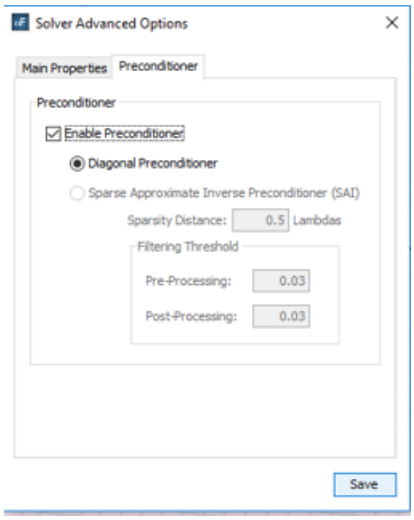


Figure 1445: Solver Advanced Options panel

Step 6 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

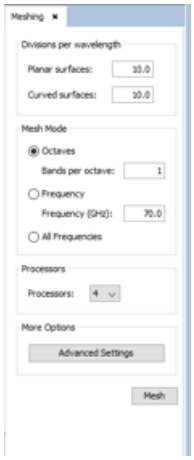


Figure 1446: Meshing panel

Step 7 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

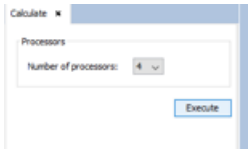


Figure 1447: Calculate panel

Step 8 Show Results

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

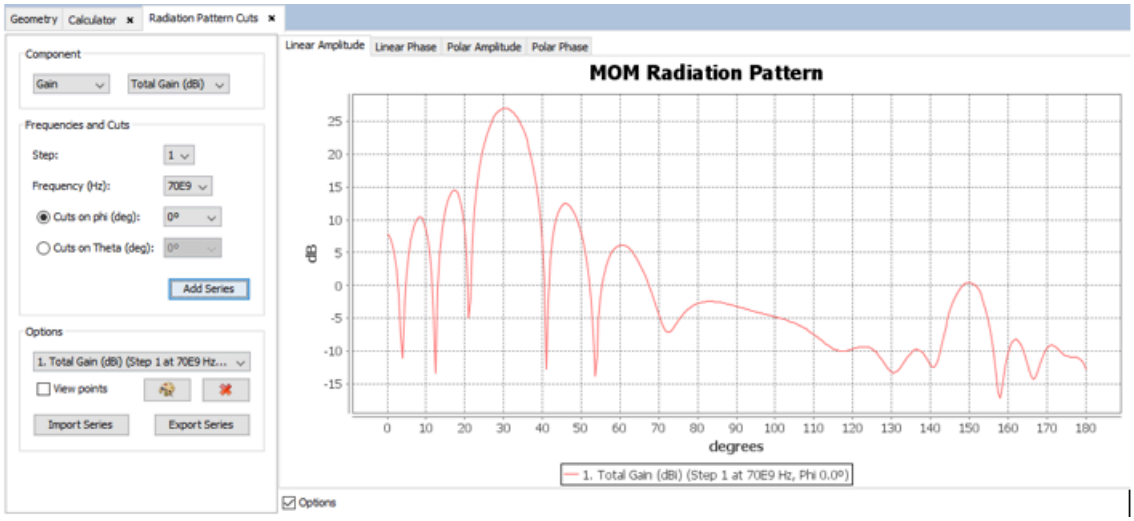


Figure 1448: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

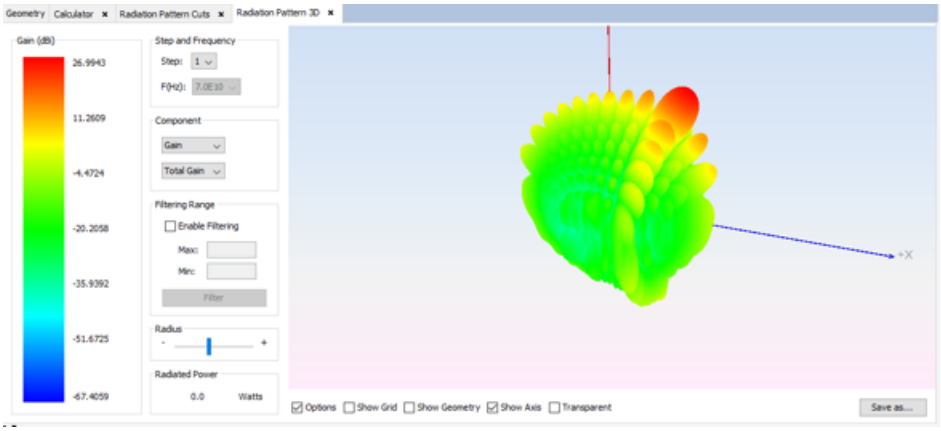


Figure 1449: Radiation Pattern 3D

Example 4: 16x16 Short Dipole Array with Dolph-Chebychev Algorithm

This case explains how to use the Dolph-Chebychev algorithms to calculates the pointing parameters in a bidimensional short dipole array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New option.**

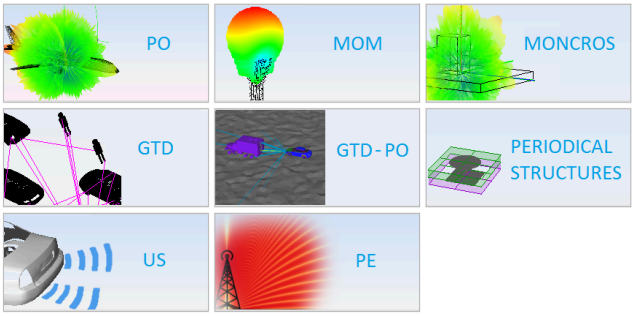


Figure 1450: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1451: Simulation panel

Step 3 Create the array.

Click on **Source - Dipole - Dipole Array** to create an array of **16x16** electrical short dipoles, with a spacing of **0.04 m**, that are oriented following the **y-axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz.**

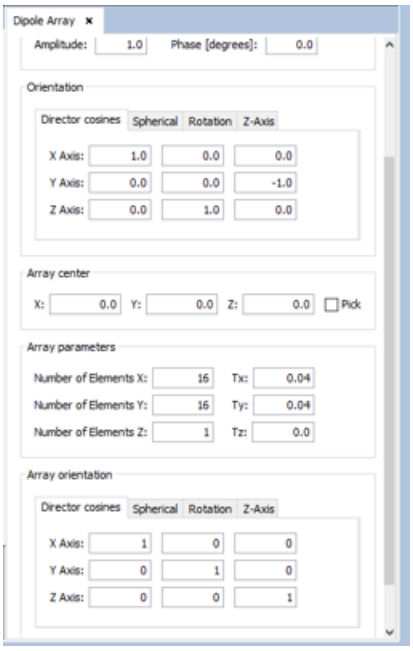


Figure 1452: Dipole Array panel

Step 4 Feed the array.

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

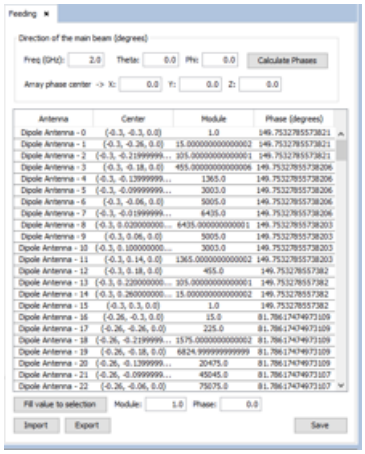
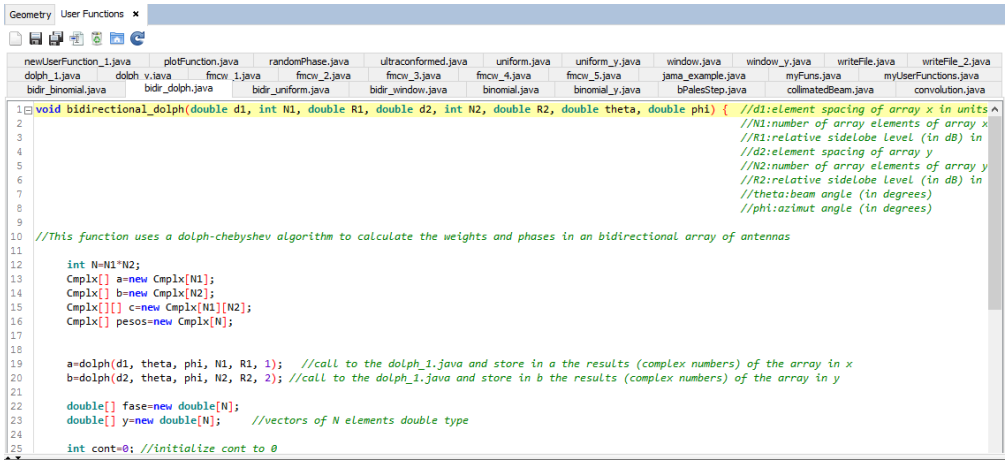


Figure 1453: Antenna Feeding panel

This is the default setting. To use the **Dolph-Chebychev algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE:** To use the bidimensional Dolph-Chebychev function it is needed to download both the bidimensional and the unidimensional functions.



```
void bidirectional_dolph(double d1, int N1, double R1, double d2, int N2, double R2, double theta, double phi) {  //d1:element spacing of array x in units
                                                                                                                   //N1:number of array elements of array x
                                                                                                                   //R1:relative sidelobe level (in dB) in
                                                                                                                   //d2:element spacing of array y
                                                                                                                   //N2:number of array elements of array y
                                                                                                                   //R2:relative sidelobe level (in dB) in
                                                                                                                   //theta:beam angle (in degrees)
                                                                                                                   //phi:azimut angle (in degrees)

//This function uses a dolph-chebyshev algorithm to calculate the weights and phases in an bidirectional array of antennas

    int N=N1*N2;
    Cmplx[] a=new Cmplx[N1];
    Cmplx[] b=new Cmplx[N2];
    Cmplx[][] c=new Cmplx[N1][N2];
    Cmplx[] pesos=new Cmplx[N];


    a=dolph(d1, theta, phi, N1, R1, 1);   //call to the dolph_1.java and store in a the results (complex numbers) of the array in x
    b=dolph(d2, theta, phi, N2, R2, 2); //call to the dolph_1.java and store in b the results (complex numbers) of the array in y

    double[] fase=new double[N];
    double[] y=new double[N];     //vectors of N elements double type

    int cont=0; //initialize cont to 0
```

Figure 1454: Bidimensional Dolph-Chebychev function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

```
    writeFile("./mydatafiles/dolph_bi.txt", y,fase); //write the weigths and phases in a text file


}
```

Figure 1455: Bidimensional Dolph-Chebychev function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

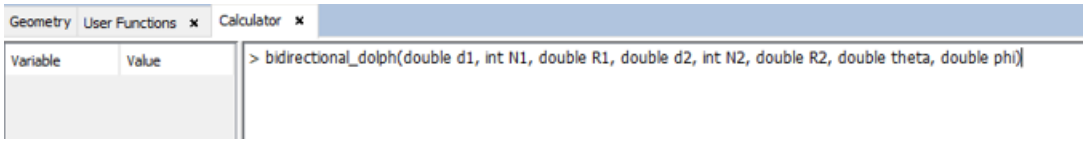


Figure 1456: Calculator panel

The parameters to set are:

- **d1:** element spacing of the array in the x axis in units of lambda
- **N1:** number of array elements of the array in the x axis
- **R1:** relative side lobe level (in dB) in the x array
- **d2:** element spacing of the array in the y axis in units of lambda
- **N2:** number of array elements of the array in the y axis
- **R2:** relative side lobe level (in dB) in the y array
- **theta:** beam angle, in degrees
- **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=45º** and **phi=0º**, and a side lobe level of **50dB**, are selected as an example.

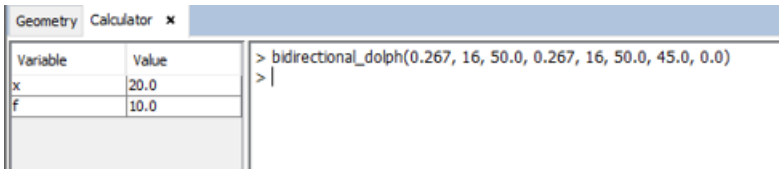


Figure 1457: Calculator panel

The text file will be automatically generated in the **mydatafiles** folder.

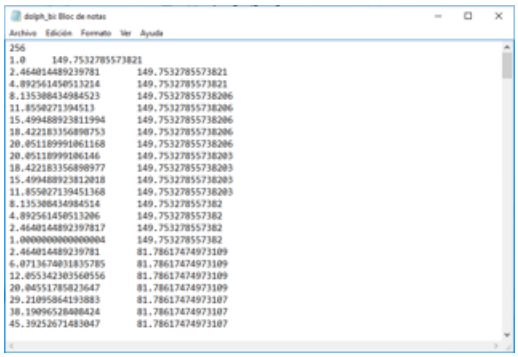


Figure 1458: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **Dolph-Chebychev algorithm**, click on **Import.**

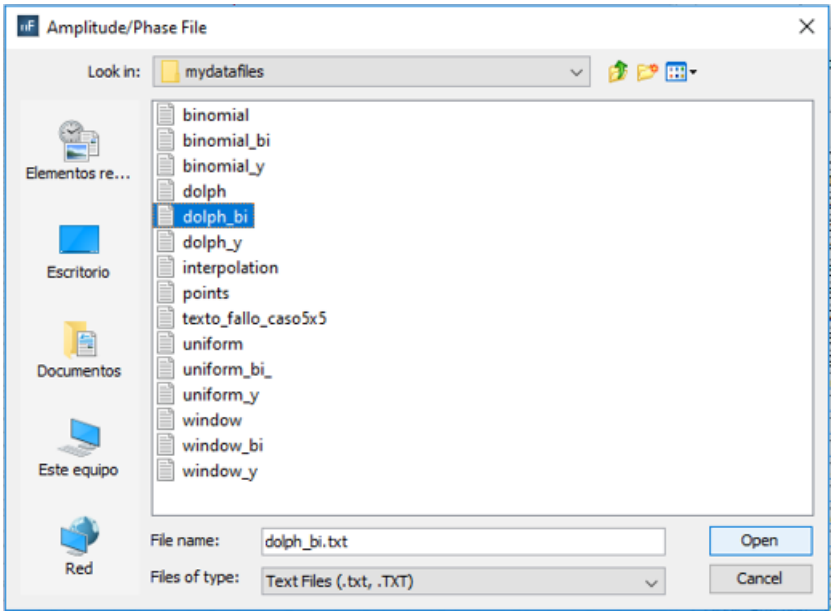


Figure 1459: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 5 Create ground plane.

In order to avoid unwanted radiation to go below the array, create a ground plane using the **plane** command, or **Geometry - Surface - Plane.** The array is situated in the XY plane with **z=0**, so the z coordinate has to be negative.

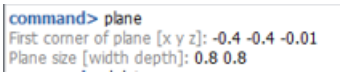


Figure 1460: Plane parameters

View of the dipole array.

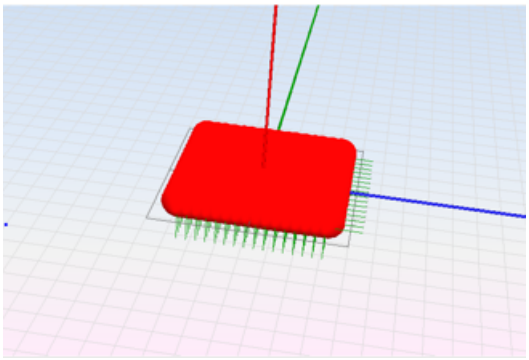


Figure 1461: Array view

Step 6 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

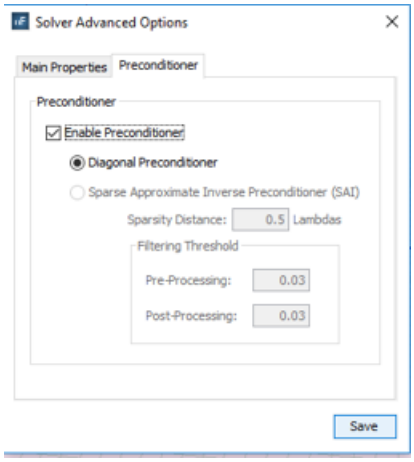


Figure 1462: Solver Advanced Options panel

Step 7 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

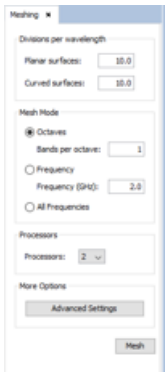


Figure 1463: Meshing panel

Click on **Mesh.**

Step 8 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

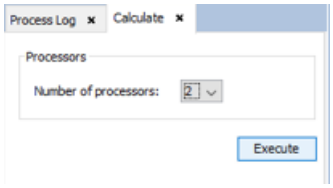


Figure 1464: Calculate panel

Step 9 Show Results.

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

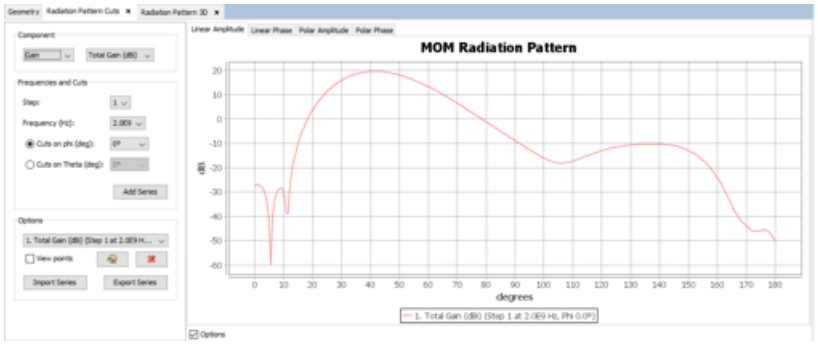


Figure 1465: Radiation Pattern Cuts

Note that the side lobe level selected earlier must be represented here.

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

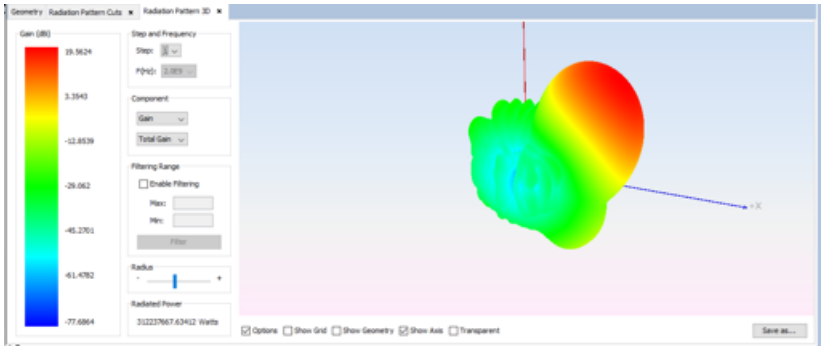


Figure 1466: Radiation Pattern 3D

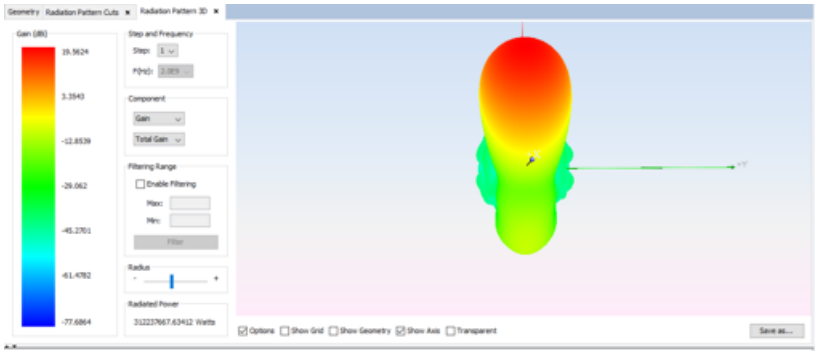


Figure 1467: Radiation Pattern 3D

Example 5: 20x20 Pattern File Array with Dolph-Chebychev Algorithm

This case explains how to use the Dolph-Chebychev algorithms to calculates the pointing parameters in a bidimensional pattern file array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

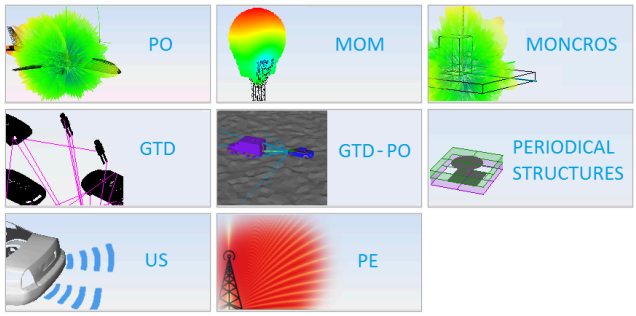


Figure 1468: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1469: Simulation panel

Step 3 Import the pattern file

Select **Source - Import Pattern File** and click on the **Import** button.

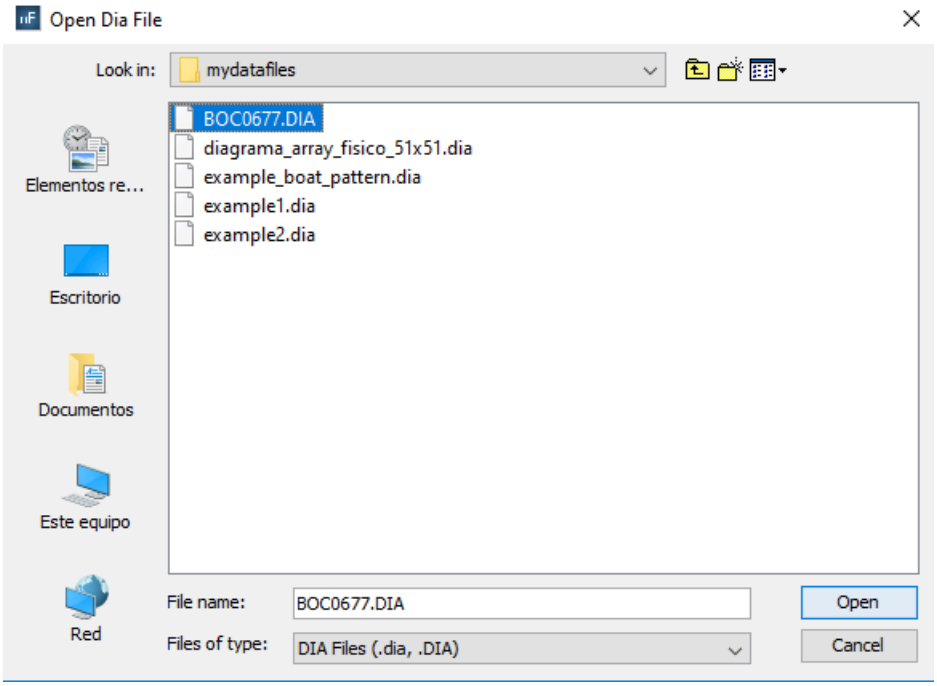


Figure 1470: Open DIA File panel

The file is now imported.

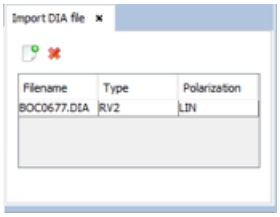


Figure 1471: Import DIA File panel

Step 4 Create the array

Click on **Source - Pattern File - Pattern File Array** to create an array of **20x20** pattern file dipoles, with a spacing of **0.04m**, that are oriented following the **z-axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz**.

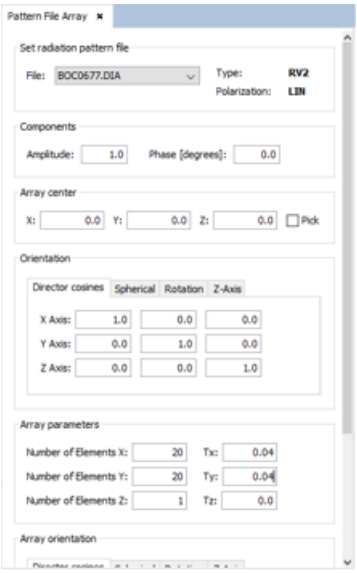


Figure 1472: Pattern File Array panel

Step 5 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

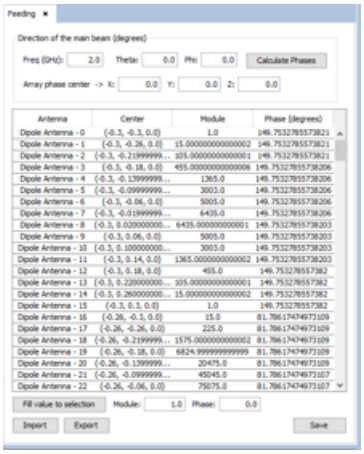


Figure 1473: Antenna Feeding panel

This is the default setting. To use the **Dolph-Chebychev algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE** To use the bidimensional Dolph-Chebychev function it is needed to download both the bidimensional and the unidimensional functions.

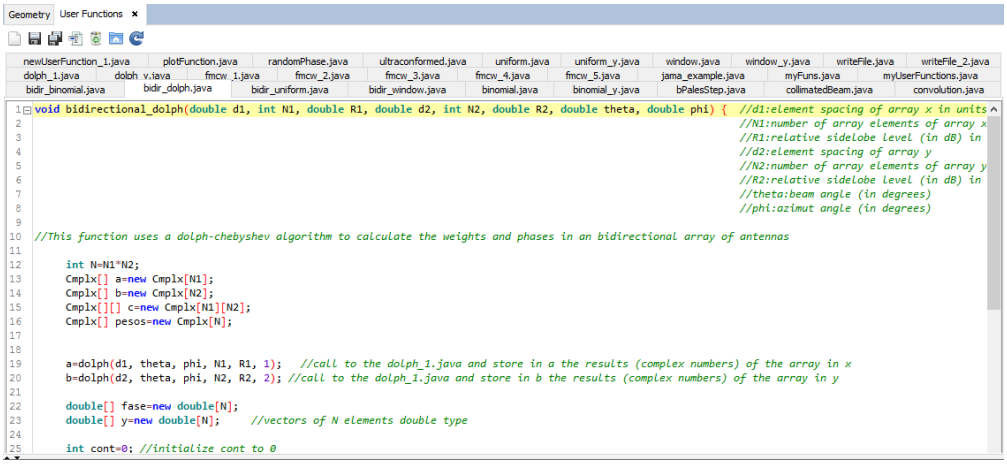


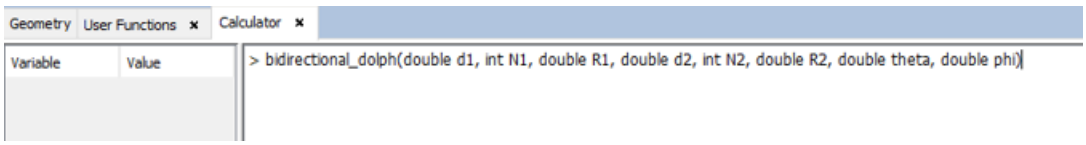
Figure 1474: Bidirectional Dolph-Chebychev function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

```
    writeFile("./mydatafiles/dolph_bi.txt", y,fase); //write the weigths and phases in a text file


}
```

Figure 1475: Bidirectional Dolph-Chebychev function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.



The parameters to set are:

- **d1:** element spacing of the array in the x axis in units of lambda
- **N1:** number of array elements of the array in the x axis
- **R1:** relative side lobe level (in dB) in the x array
- **d2:** element spacing of the array in the y axis in units of lambda
- **N2:** number of array elements of the array in the y axis
- **R2:** relative side lobe level (in dB) in the y array
- **theta:** beam angle, in degrees
- **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=20º** and **phi=0º**, and a side lobe level of **50dB**, are selected as an example.

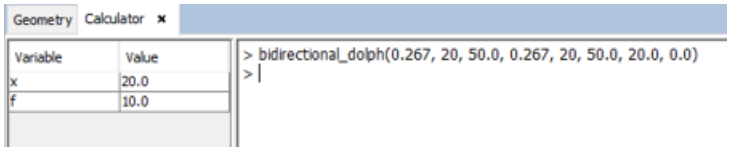


Figure 1476: Calculator panel

The text file will be automatically generated in the **mydatafiles** folder.

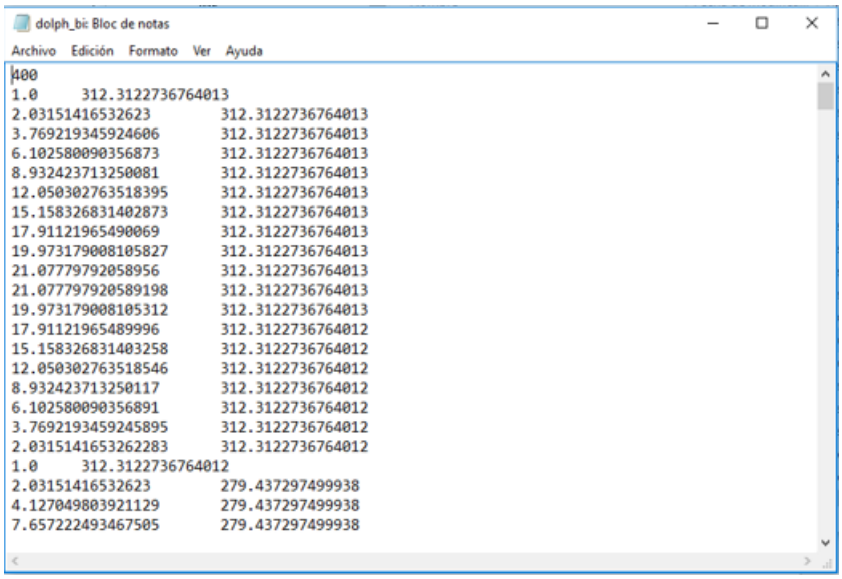


Figure 1477: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **Dolph-Chebychev algorithm**, click on **Import**.

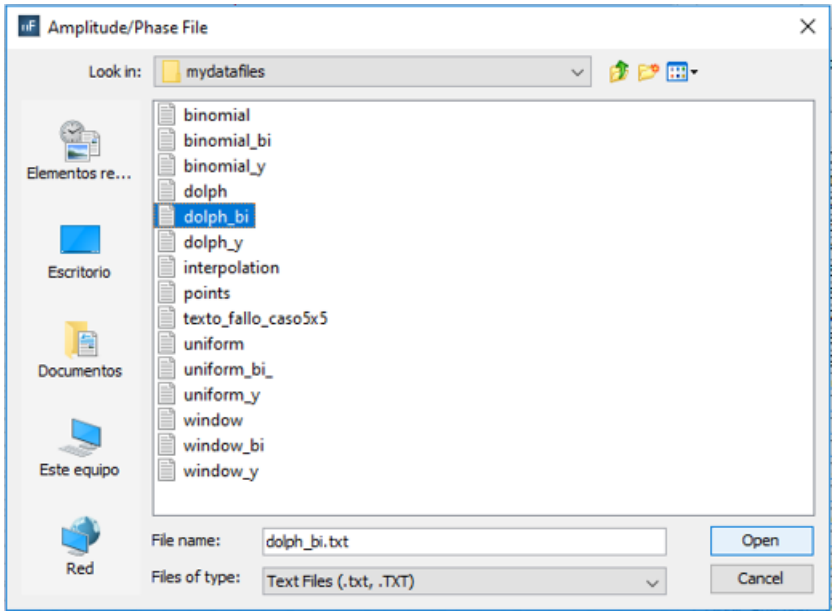


Figure 1478: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 6 Create ground plane.

In order to avoid unwanted radiation to go below the array, create a ground plane using the **plane** command, or **Geometry - Surface - Plane.** The array is situated in the XY plane with **z=0**, so the z coordinate has to be negative.

```
command> plane
First corner of plane [x y z]: -0.5 -0.5 -0.05
Plane size [width depth]: 1.0 1.0
command>
```

Figure 1479: Plane parameters

View of the pattern file array.

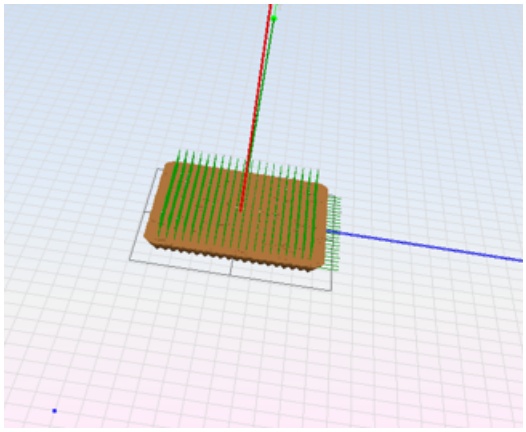


Figure 1480: Array view

Step 7 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

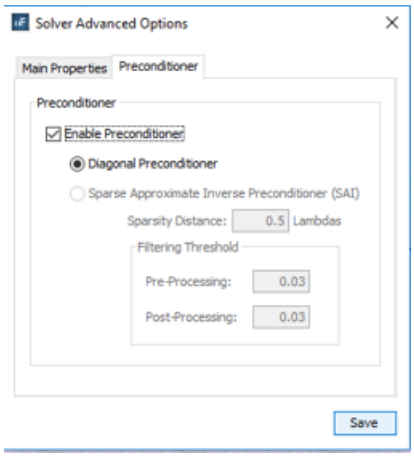


Figure 1481: Solver Advanced Options panel

Step 8 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

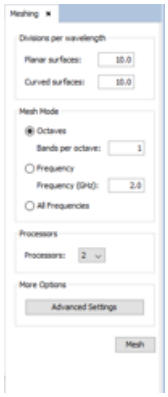


Figure 1482: Meshing panel

Click on **Mesh.**

Step 9 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

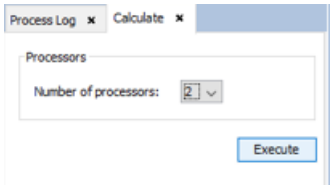


Figure 1483: Calculate panel

Step 10 Show Results.

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

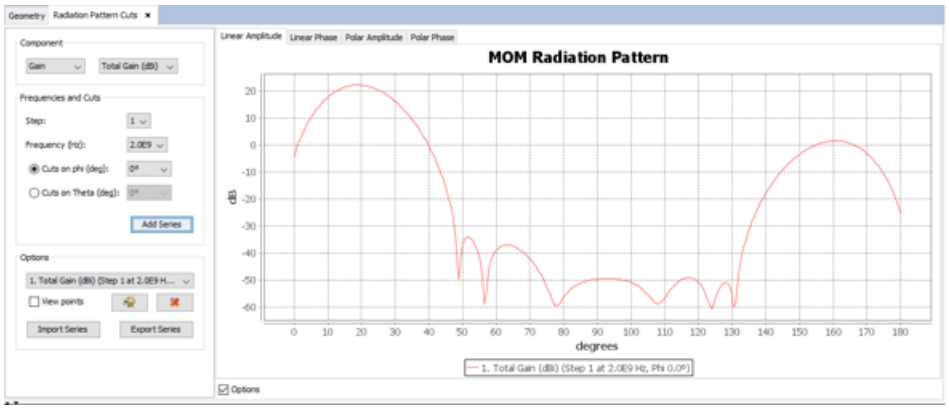


Figure 1484: Radiation Pattern Cuts

Note that the side lobe level selected earlier must be represented here.

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

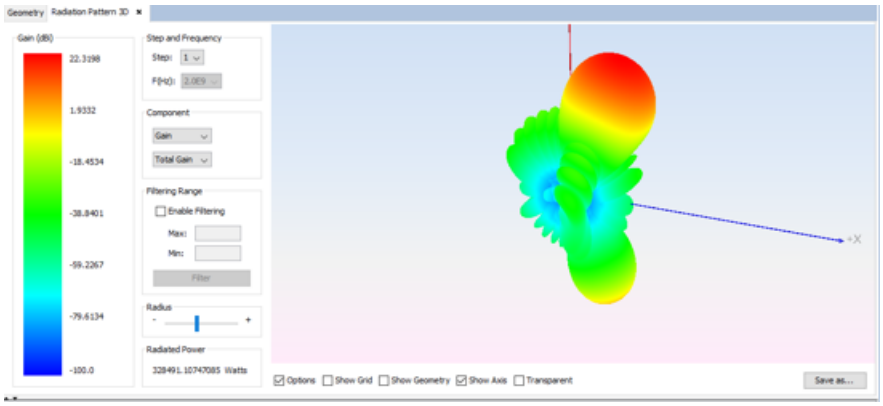


Figure 1485: Radiation Pattern 3D

Example 6: 14x14 Coaxial Feed Array with Dolph-Chebychev Algorithm

This case explains how to use the Dolph-Chebychev algorithms to calculates the pointing parameters in a bidimensional coaxial feed array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

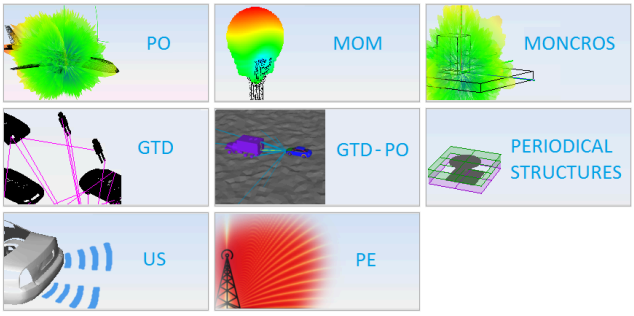


Figure 1486: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1487: Simulation panel

Step 3 Create the array

First, select 'milimeters' on units list on the bar at the bottom of the main window.

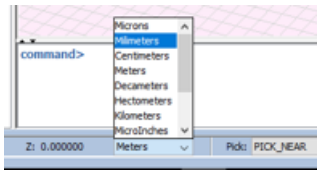


Figure 1488: The first element is created using Source - Coaxial Feed - Add Feed Point.

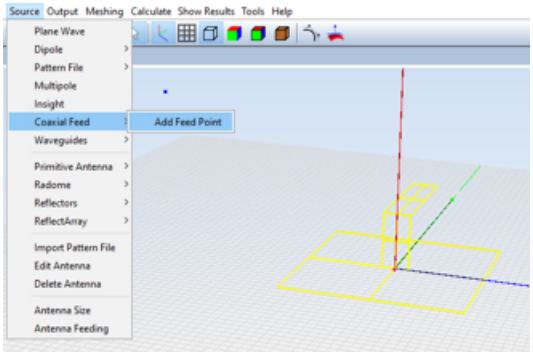


Figure 1489: Add feed point

Select the surfaces and click on **Add.**

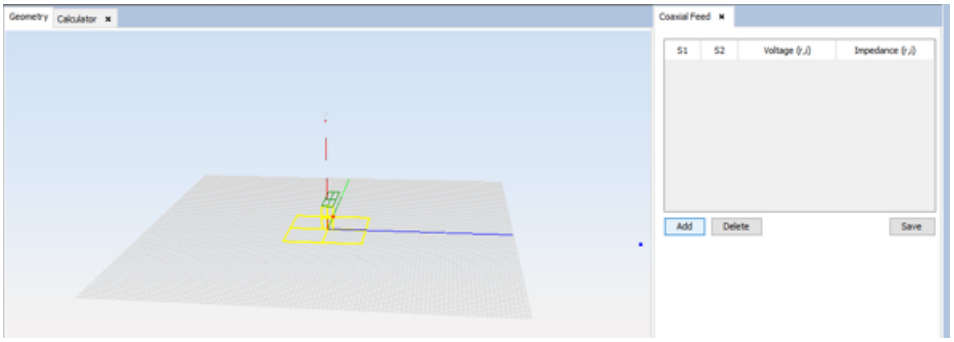


Figure 1490: Coaxial Feed panel

Then save the feed point.

Use the **array** command and enter the characteristics of the array.

```
command> array
Select objects on screen (Press enter when done):
Number of objects in X axis (default=1): 14
Number of objects in Y axis (default=1): 14
Number of objects in Z axis (default=1): 1
Select start point [x y z] (default=(0,0,0)): 0 0 0
Select end point [x y z]}: 2.10407 2.10407 0
command>
```

Figure 1491: Array parameters

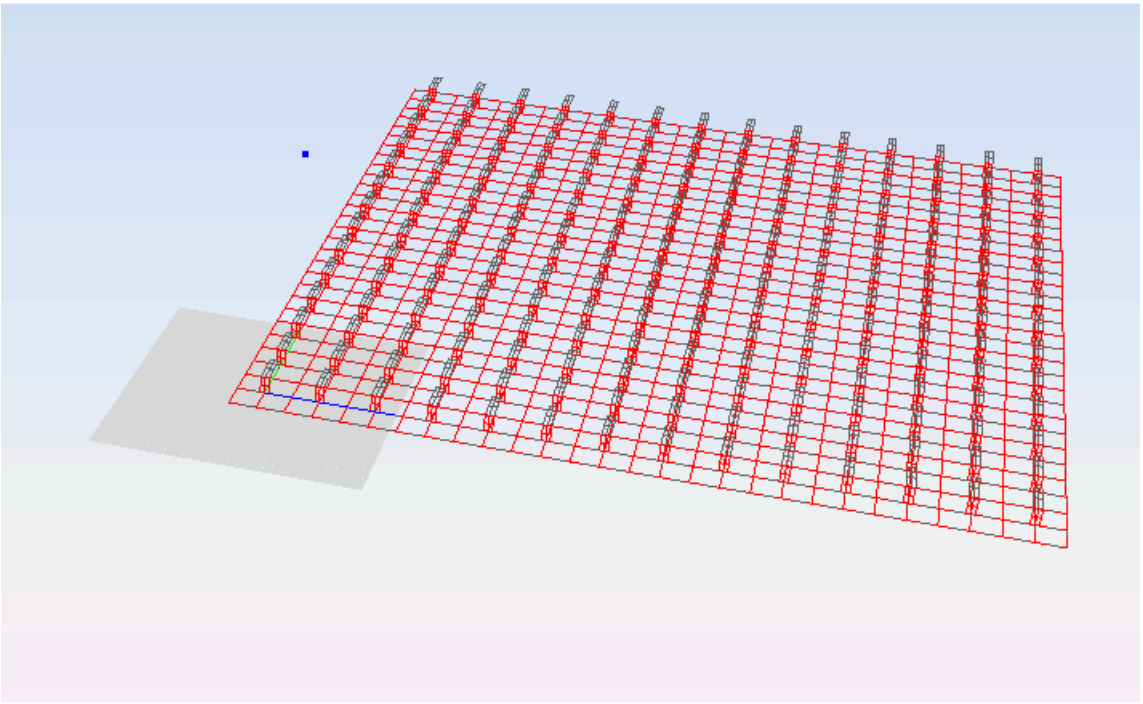


Figure 1492: Array view

Step 4 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

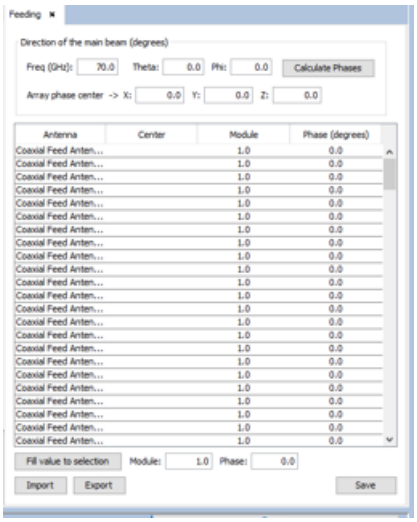


Figure 1493: Antenna Feeding panel

This is the default setting. To use the **Dolph-Chebychev algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE** To use the bidimensional Dolph-Chebychev function it is needed to download both the bidimensional and the unidimensional functions.

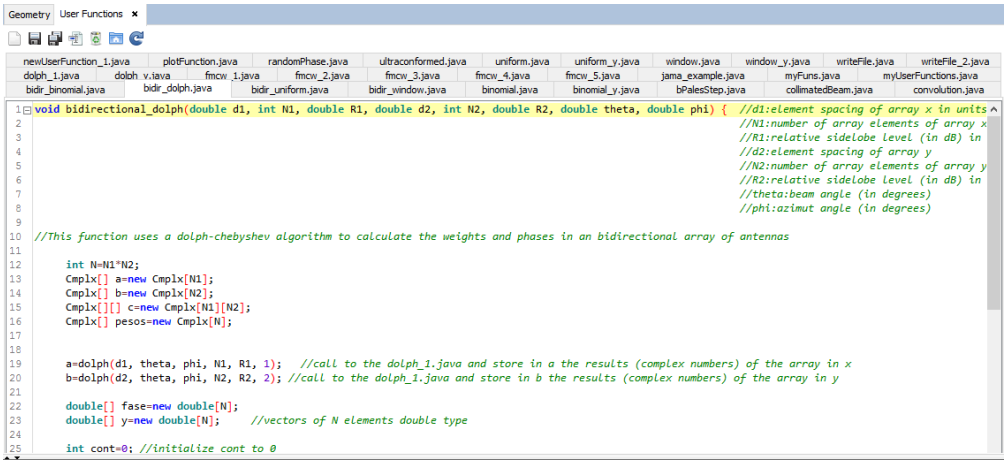


Figure 1494: Bidimensional Dolph-Chebychev function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

```
    writeFile("./mydatafiles/dolph_bi.txt", y,fase); //write the weigths and phases in a text file


}
```

Figure 1495: Bidimensional Dolph-Chebychev function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

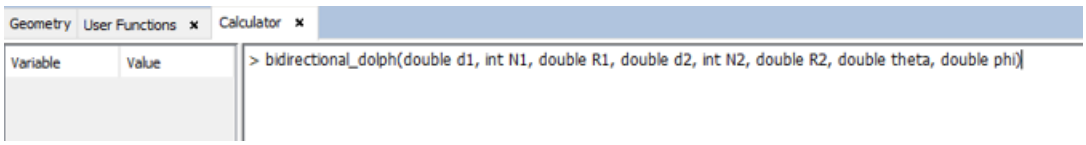


Figure 1496: Calculator panel

The parameters to set are:

· **d1:** element spacing of the array in the x axis in units of lambda

· **N1:** number of array elements of the array in the x axis

· **R1:** relative side lobe level (in dB) in the x array

· **d2:** element spacing of the array in the y axis in units of lambda

· **N2:** number of array elements of the array in the y axis

· **R2:** relative side lobe level (in dB) in the y array

· **theta:** beam angle, in degrees

· **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=30º** and **phi=0º**, and a side lobe level of **50dB**, are selected as an example.

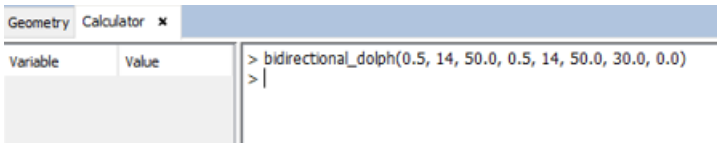


Figure 1497: Calculator panel

A spacing of **0.5** in units of lambdas is selected because is equivalent to a spacing of **2.104** millimetres at **70 GHz**.

The text file will be automatically generated in the **mydatafiles** folder.

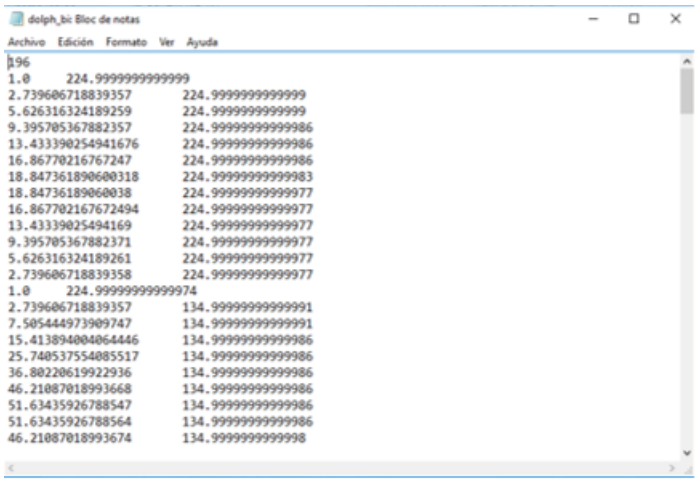


Figure 1498: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **Dolph-Chebychev algorithm**, click on **Import**.

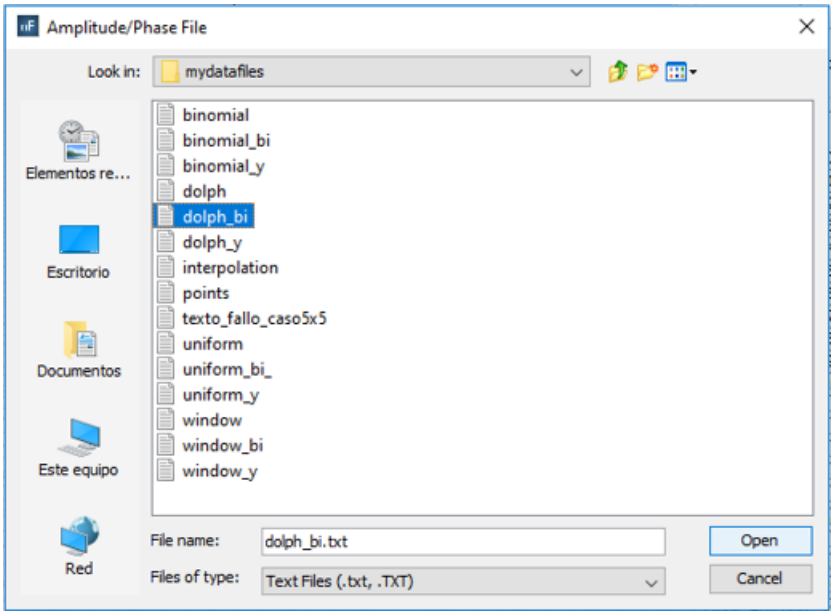


Figure 1499: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 5 Solver parameters

Select **Solver - Advanced Options** and set the parameters as shown.

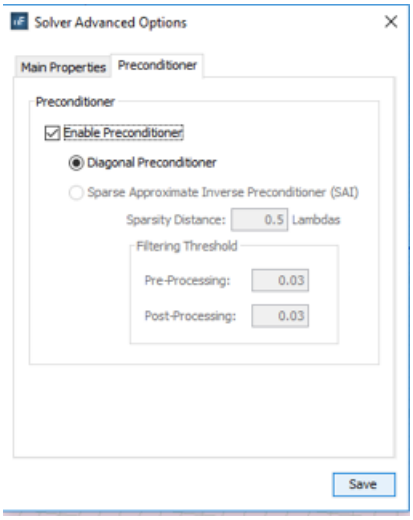


Figure 1500: Solver Advanced Options panel

Step 6 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

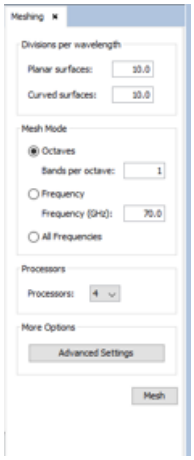


Figure 1501: Meshing panel

Step 7 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

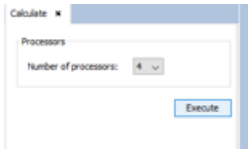


Figure 1502: Calculate panel

Step 8 Show Results

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

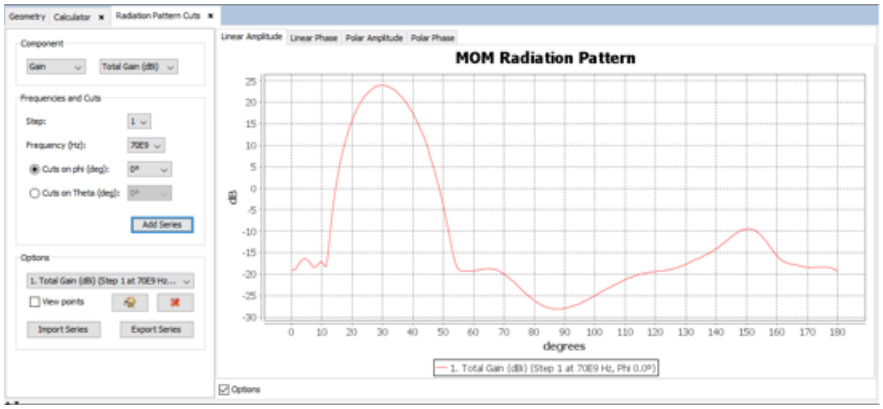


Figure 1503: Radiation Pattern Cuts

Note that the side lobe level selected earlier must be represented here.

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

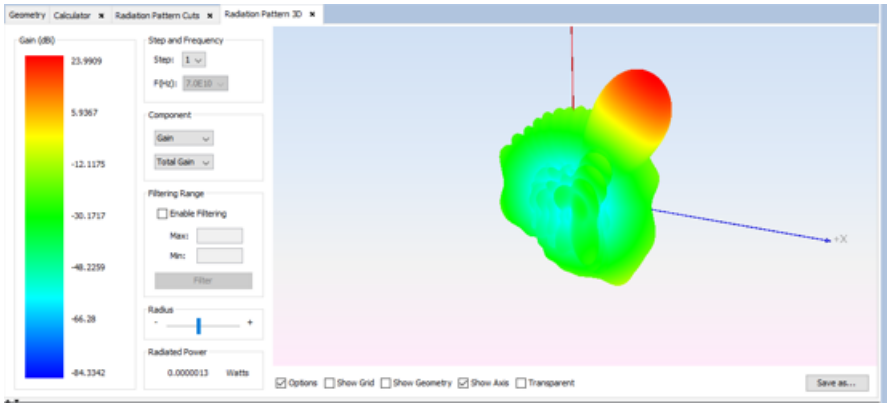


Figure 1504: Radiation Pattern 3D

Example 7: 16x16 Short Dipole Array with Binomial Algorithm

This case explains how to use the binomial algorithms to calculates the pointing parameters in a bidimensional short dipole array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

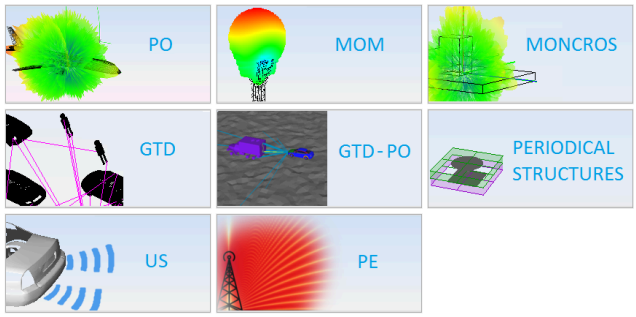


Figure 1505: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1506: Simulation panel

Step 3 Create the array.

Click on **Source - Dipole - Dipole Array** to create an array of **16x16** electrical short dipoles, with a spacing of **0.04 m**, that are oriented following the **y-axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz.**

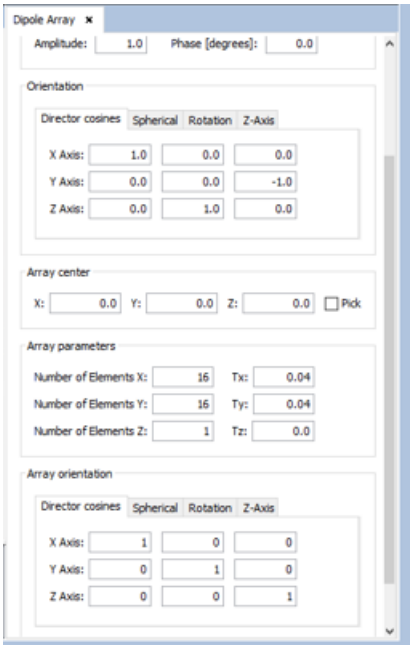


Figure 1507: Dipole Array panel

Step 4 Feed the array.

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

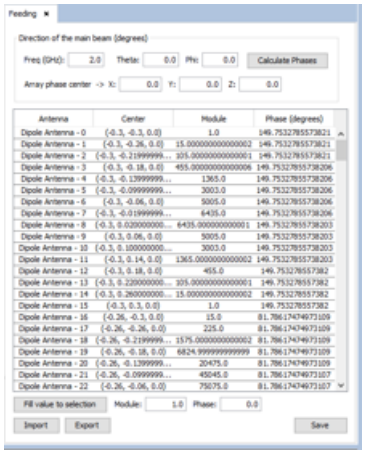
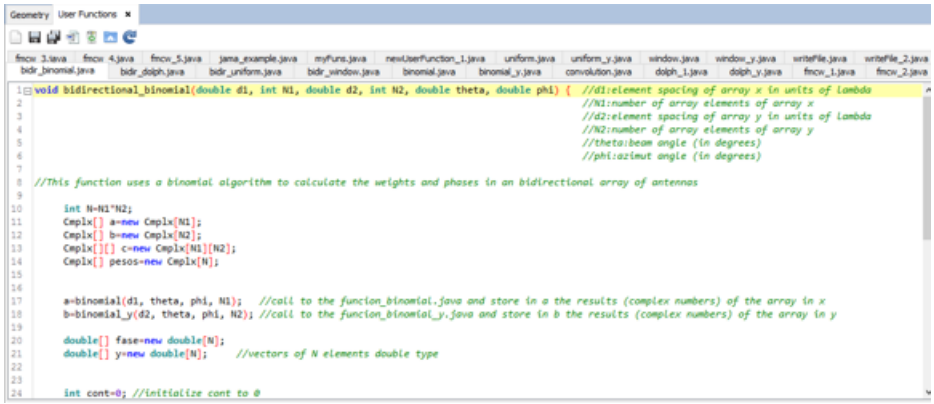


Figure 1508: Antenna Feeding panel

This is the default setting. To use the **binomial algorithm** click on **Tools - User Function** and select the corresponding function. **NOTE:** To use the bidimensional binomial function it is needed to download both the bidimensional and the unidimensional functions.



```
void bidirectional_binomial(double d1, int N1, double d2, int N2, double theta, double phi) {  //d1:element spacing of array x in units of lambda
                                                                                               //N1:number of array elements of array x
                                                                                               //d2:element spacing of array y in units of lambda
                                                                                               //N2:number of array elements of array y
                                                                                               //theta:beam angle (in degrees)
                                                                                               //phi:azimut angle (in degrees)

//This function uses a binomial algorithm to calculate the weights and phases in an bidirectional array of antennas

    int N=N1*N2;
    Cmplx[] a=new Cmplx[N1];
    Cmplx[] b=new Cmplx[N2];
    Cmplx[][] c=new Cmplx[N1][N2];
    Cmplx[] pesos=new Cmplx[N];


    a=binomial(d1, theta, phi, N1);   //call to the funcion_binomial.java and store in a the results (complex numbers) of the array in x
    b=binomial_y(d2, theta, phi, N2); //call to the funcion_binomial_y.java and store in b the results (complex numbers) of the array in y

    double[] fase=new double[N];
    double[] y=new double[N];     //vectors of N elements double type


    int cont=0; //initialize cont to 0
```

Figure 1509: Bidimensional Uniform function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

```
    writeFile("./mydatafiles/binomial_bi.txt", y,fase); //write the weigths and phases in a text file

}
```

Figure 1510: Bidimensional Uniform function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

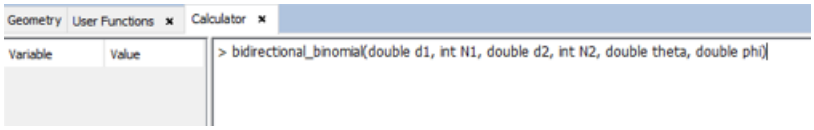


Figure 1511: Calculator panel

The parameters to set are:

- **d1:** element spacing of the array in the x axis in units of lambda
- **N1:** number of array elements of the array in the x axis
- **d2:** element spacing of the array in the y axis in units of lambda
- **N2:** number of array elements of the array in the y axis
- **theta:** beam angle, in degrees
- **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=45º** and **phi=0º** are selected as an example.

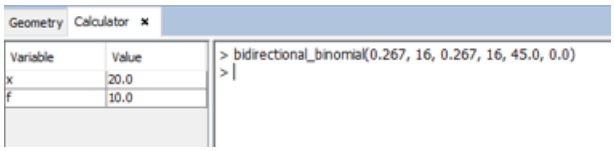


Figure 1512: The text file will be automatically generated in the mydatafiles folder.

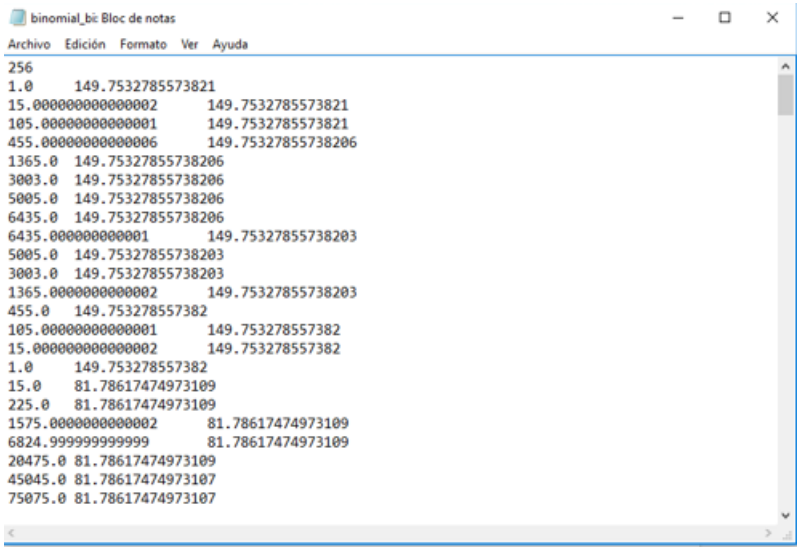


Figure 1513: Results file

Now, apply these results to the array created before by clicking on S **ource - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **binomial algorithm**, click on **Import**.

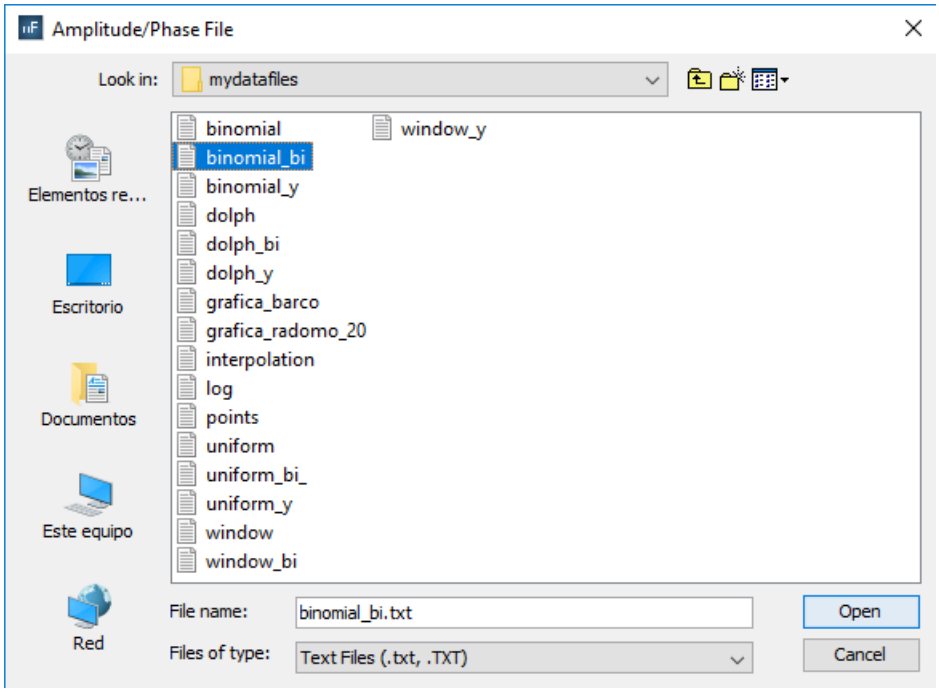


Figure 1514: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 5 Create ground plane.

In order to avoid unwanted radiation to go below the array, create a ground plane using the **plane** command, or **Geometry - Surface - Plane**. The array is situated in the XY plane with **z=0**, so the z coordinate has to be negative.

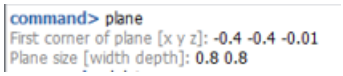


Figure 1515: Plane parameters

View of the dipole array.

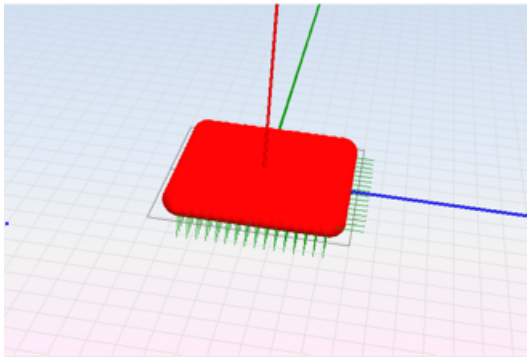


Figure 1516: Array view

Step 6 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

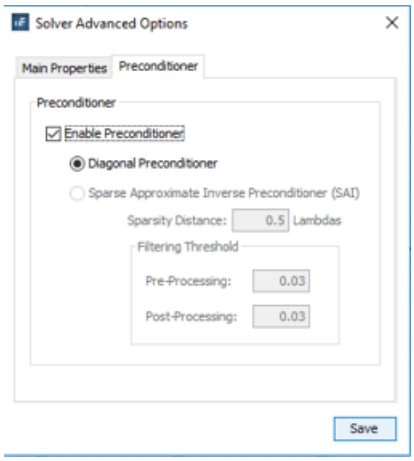


Figure 1517: Solver Advanced Options panel

Step 7 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

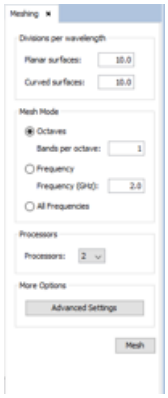


Figure 1518: Meshing panel

Click on **Mesh**.

Step 8 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

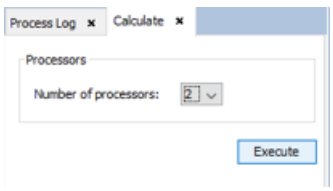


Figure 1519: Calculate panel

Step 9 Show Results.

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

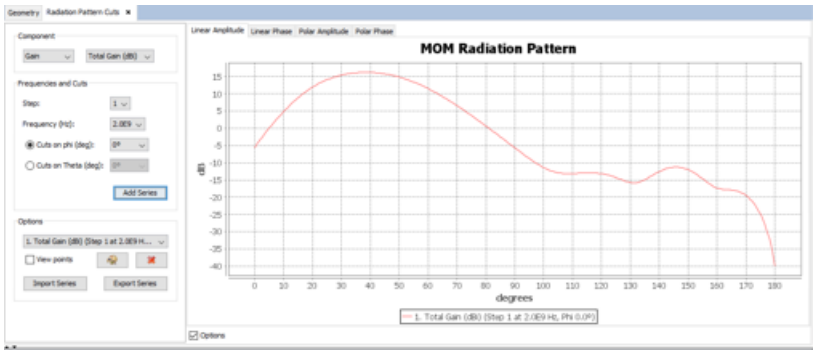


Figure 1520: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

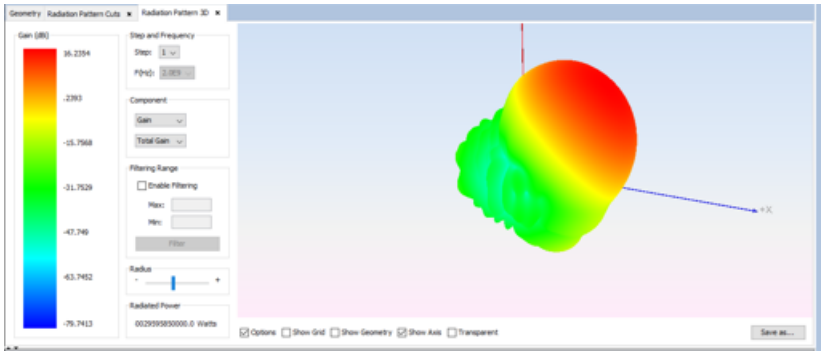


Figure 1521: Radiation Pattern 3D

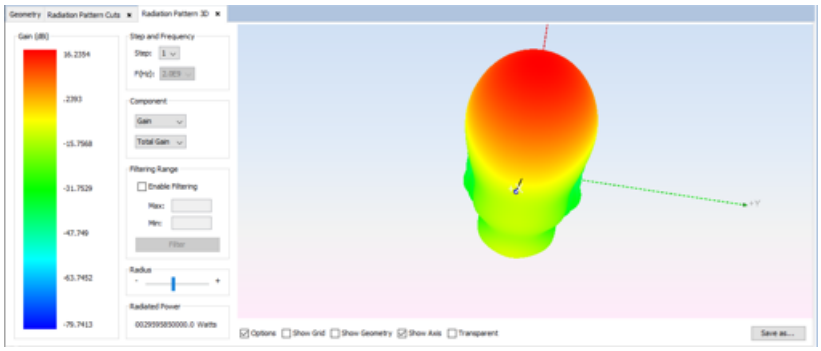


Figure 1522: Radiation Pattern 3D

Example 8: 20x20 Pattern File Array with Binomial Algorithm

This case explains how to use the binomial algorithms to calculates the pointing parameters in a bidimensional pattern file array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

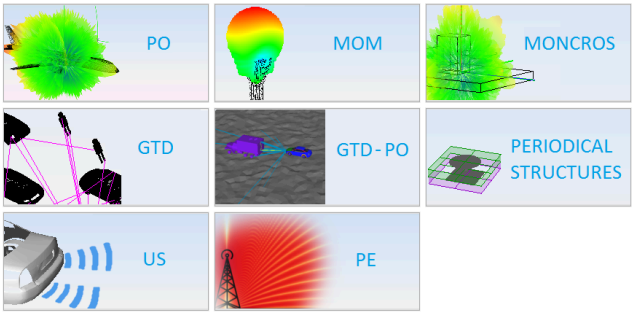


Figure 1523: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1524: Simulation panel

Step 3 Import the pattern file

Select **Source - Import Pattern** File and click on the **Import** button.

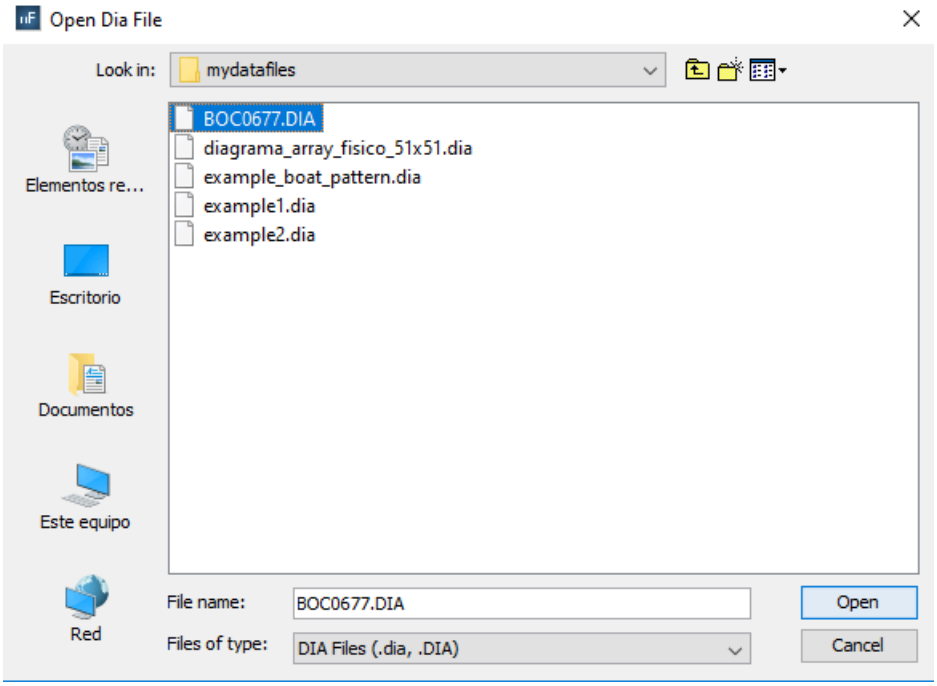


Figure 1525: Open DIA File panel

The file is now imported.

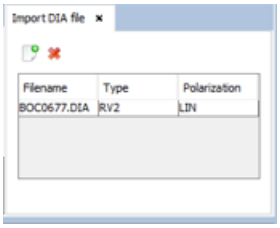


Figure 1526: Import DIA File panel

Step 4 Create the array

Click on **Source - Pattern File - Pattern File Array** to create an array of **20x20** pattern file dipoles, with a spacing of **0.04m**, that are oriented following the **z-axis**. The array is located on the **XY** plane. The spacing of **0.04m** is equivalent to a spacing of **0.267** in units of lambda, at a frequency of **2 GHz.**

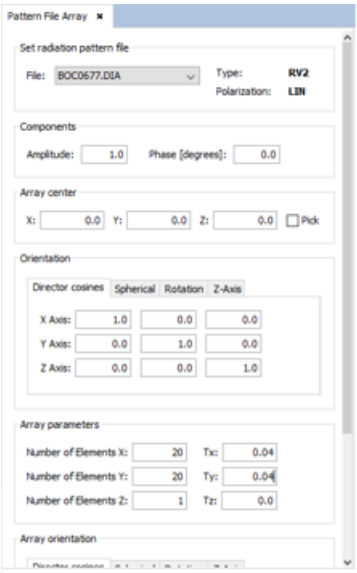


Figure 1527: Pattern File Array panel

Step 5 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

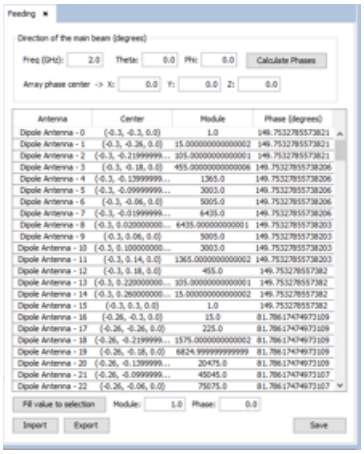


Figure 1528: Antenna Feeding panel

This is the default setting. To use the **binomial algorithm** click on **Tools - User Function** and select the corresponding function (which can be downloaded). **NOTE** To use the bidimensional binomial function it is needed to download both the bidimensional and the unidimensional functions.

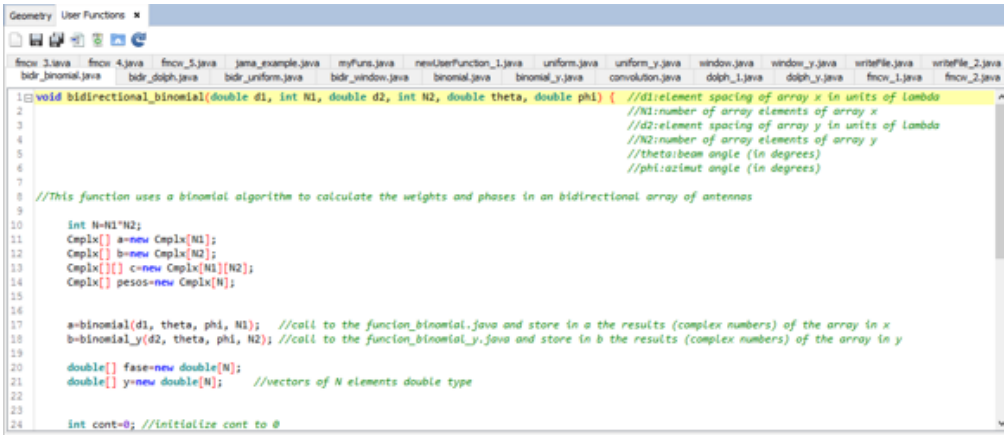


Figure 1529: Bidimensional Binomial function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

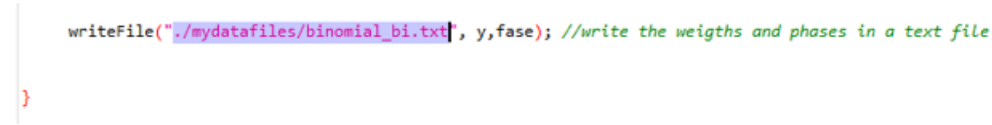


Figure 1530: Bidimensional Binomial function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

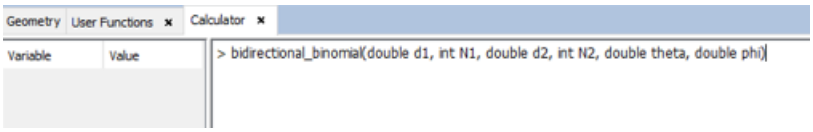


Figure 1531: Calculator panel

The parameters to set are:

- **d1:** element spacing of the array in the x axis in units of lambda
- **N1:** number of array elements of the array in the x axis
- **d2:** element spacing of the array in the y axis in units of lambda
- **N2:** number of array elements of the array in the y axis
- **theta:** beam angle, in degrees
- **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=20º** and **phi=0º** are selected as an example.

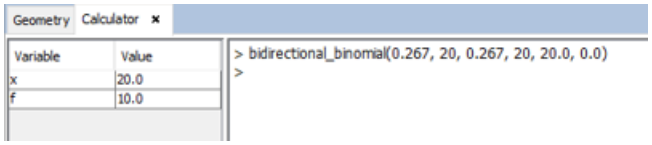


Figure 1532: Calculator panel

The text file will be automatically generated in the **mydatafiles** folder.

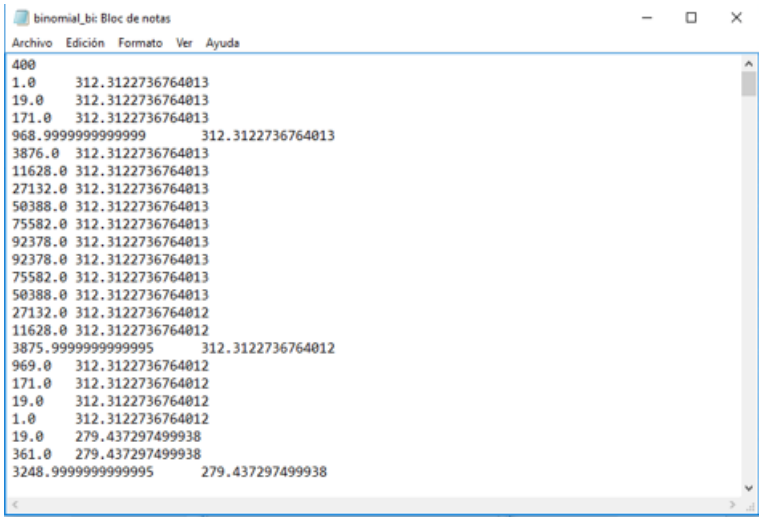


Figure 1533: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **binomial algorithm**, click on **Import.**

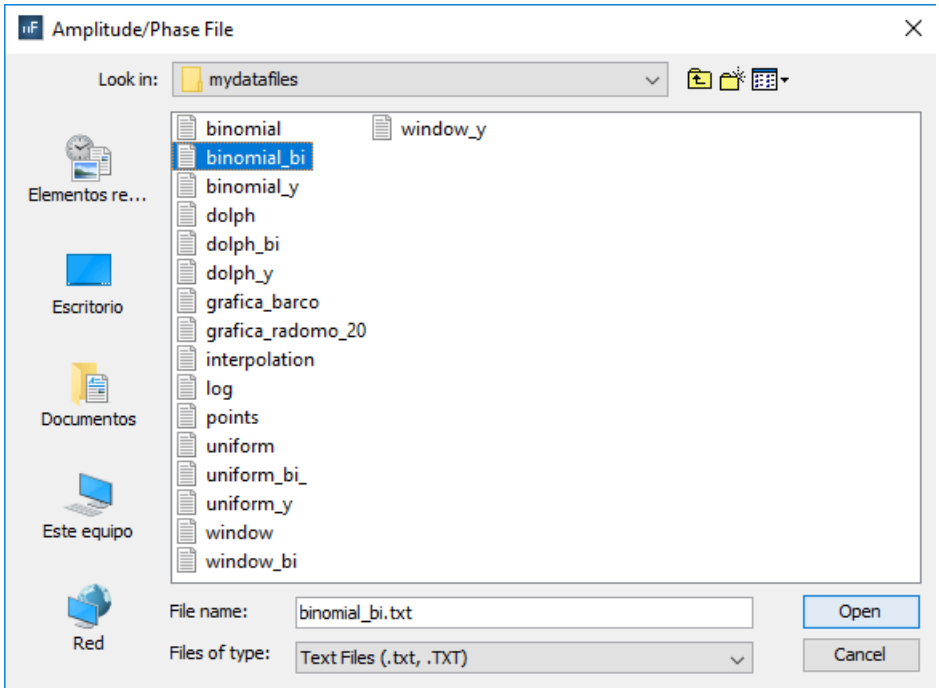


Figure 1534: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 6 Create ground plane.

In order to avoid unwanted radiation to go below the array, create a ground plane using the **plane** command, or **Geometry - Surface - Plane.** The array is situated in the XY plane with **z=0**, so the z coordinate has to be negative.

```
command> plane
First corner of plane [x y z]: -0.5 -0.5 -0.05
Plane size [width depth]: 1.0 1.0
command>
```

Figure 1535: Plane parameters

View of the pattern file array.

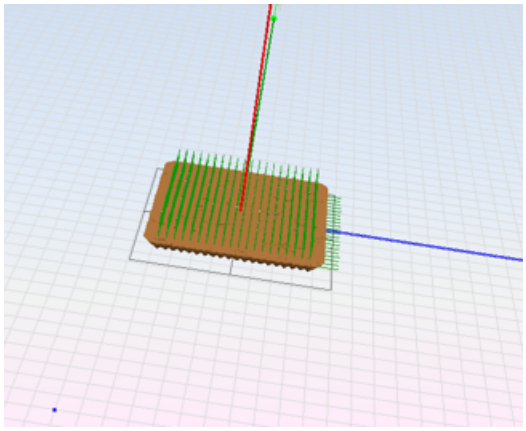


Figure 1536: Array view

Step 7 Solver parameters.

Select **Solver - Advanced Options** and set the parameters as shown.

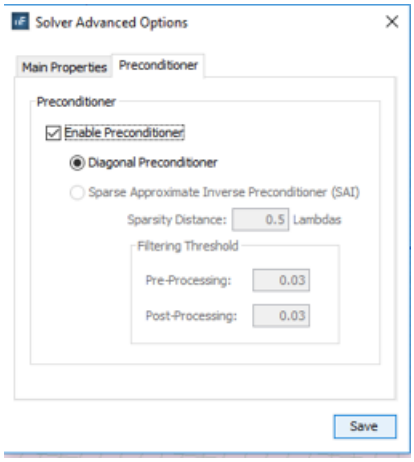


Figure 1537: Solver Advanced Options panel

Step 8 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

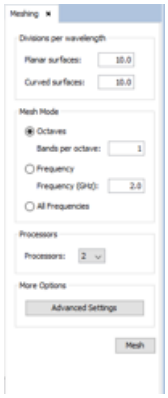


Figure 1538: Meshing panel

Click on **Mesh.**

Step 9 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

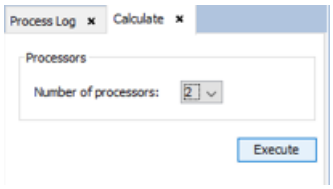


Figure 1539: Calculate panel

Step 10 Show Results

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

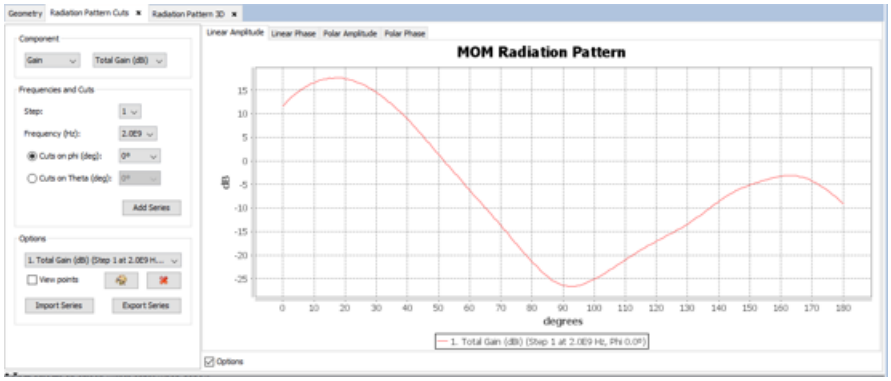


Figure 1540: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

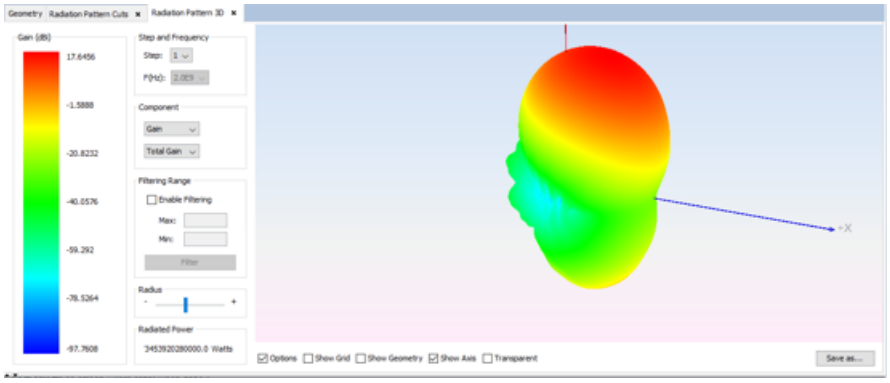


Figure 1541: Radiation Pattern 3D

Example 9: 14x14 Coaxial Feed Array with Binomial Algorithm

This case explains how to use the binomial algorithms to calculates the pointing parameters in a bidimensional coaxial feed array.

Step 1 Create a new MoM Project.

Open newFASANT and select **File - New** option.

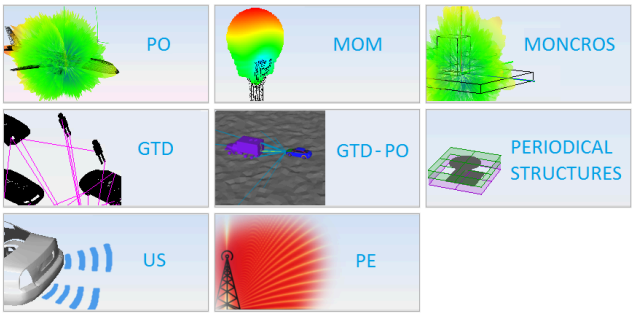


Figure 1542: New Project panel

Select **MOM** option on the previous figure and start to configure the project.

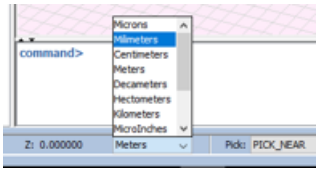
Step 2 Set the simulation parameters as shown.

Select **Simulation - Parameters** option, set the parameters and save it.

Figure 1543: Simulation panel

Step 3 Create the array

First, select 'milimeters' on units list on the bar at the bottom of the main window.



The first element is created using **Source** - **Coaxial Feed** - **Add Feed Point**.

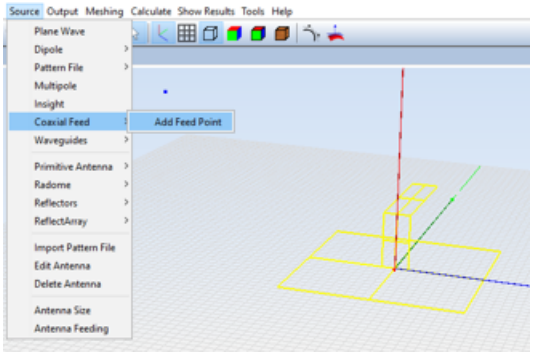


Figure 1544: Add feed point

Select the surfaces and click on **Add.**

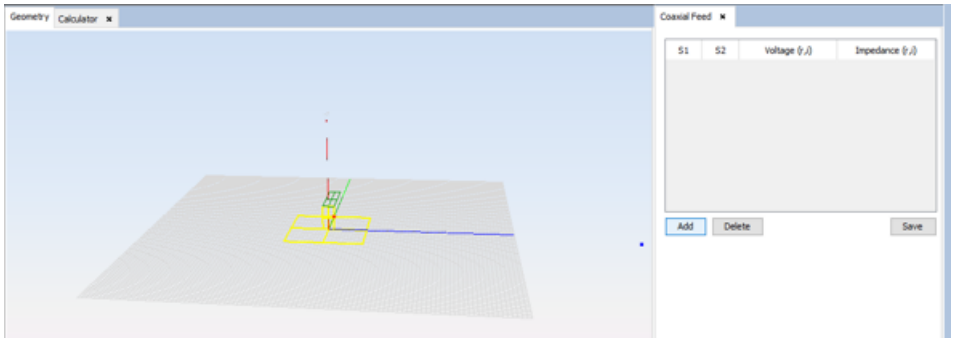


Figure 1545: Coaxial Feed panel

Then save the feed point.

Use the **array** command and enter the characteristics of the array.

```
command> array
Select objects on screen (Press enter when done):
Number of objects in X axis (default=1): 14
Number of objects in Y axis (default=1): 14
Number of objects in Z axis (default=1): 1
Select start point [x y z] (default=(0,0,0)): 0 0 0
Select end point [x y z]}: 2.10407 2.10407 0
command>
```

Figure 1546: Array parameters

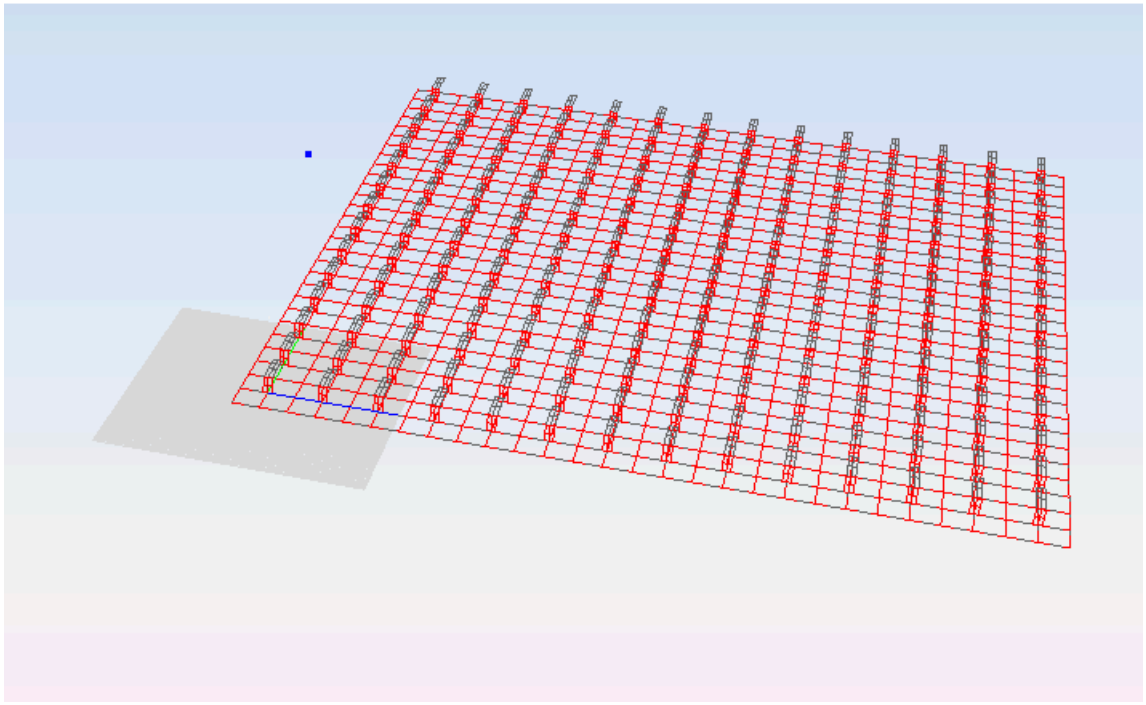


Figure 1547: Array view

Step 4 Feed the array

To set the feeding of the array select **Source - Antenna Feeding** and the following panel will open.

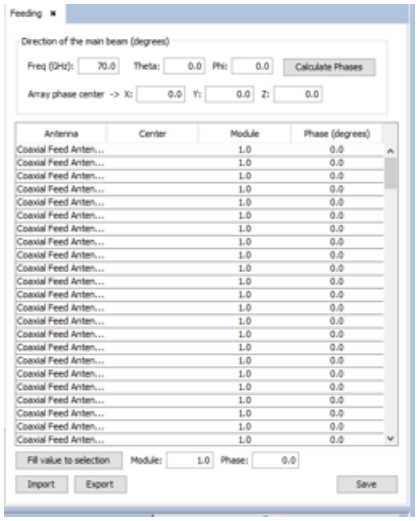


Figure 1548: Antenna Feeding panel

This is the default setting. To use the **binomial algorithm** click on **Tools - User Function** and select the corresponding function (which can be downloaded). **NOTE** To use the bidimensional binomial function it is needed to download both the bidimensional and the unidimensional functions.

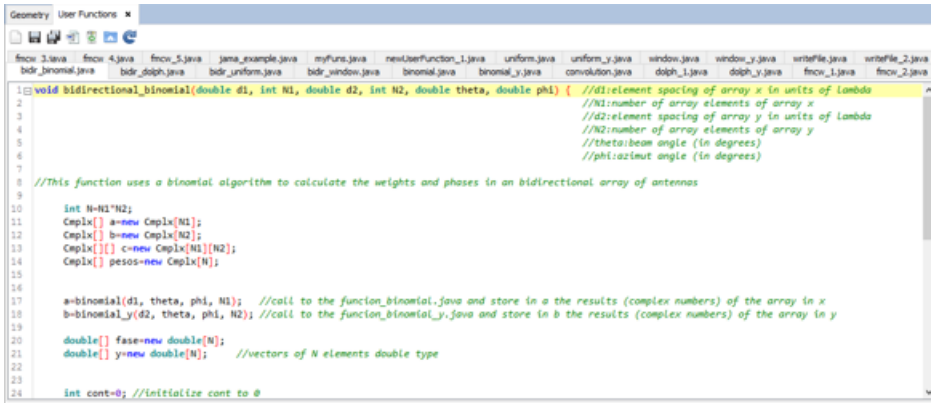


Figure 1549: Bidimensional Binomial function

A path has been selected by default so the files will be created on the **mydatafiles** folder in the **newFASANT** directory.

```
    writeFile("./mydatafiles/binomial_bi.txt", y,fase); //write the weigths and phases in a text file

}
```

Figure 1550: Bidimensional Binomial function

The next step is generating the text file. To do so click on **Tools - Calculator** and write the call to the function.

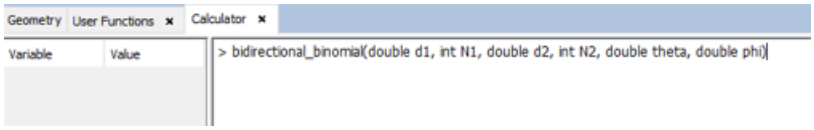


Figure 1551: Calculator panel

The parameters to set are:

· **d1:** element spacing of the array in the x-axis in units of lambda

· **N1:** number of array elements of the array in the x-axis

· **d2:** element spacing of the array in the y-axis in units of lambda

· **N2:** number of array elements of the array in the y-axis

· **theta:** beam angle, in degrees

· **phi:** azimuth angle, in degrees

In this case, set the parameters as shown. Angles of **theta=30º** and **phi=0º** are selected as an example.

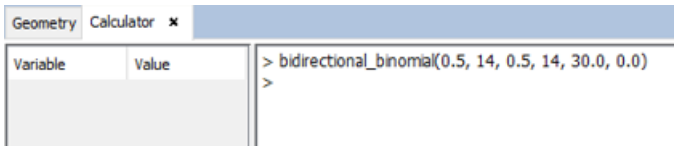


Figure 1552: Calculator panel

A spacing of **0.5** in units of lambdas is selected because is equivalent to a spacing of **2.104** millimeters at **70 GHz.**

The text file will be automatically generated in the **mydatafiles** folder.

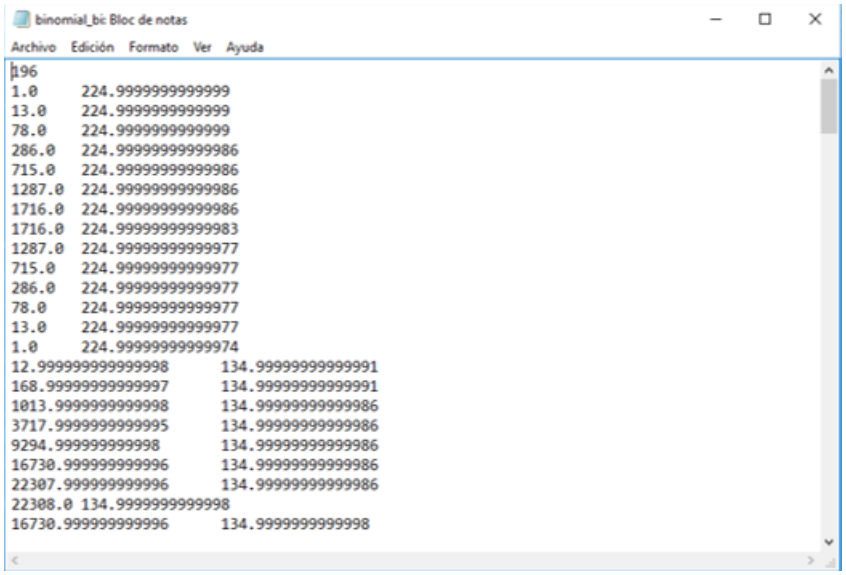


Figure 1553: Results file

Now, apply these results to the array created before by clicking on **Source - Antenna Feeding.**

The panel shown before will appear. To use the weights and phases calculated with the **binomial algorithm**, click on **Import.**

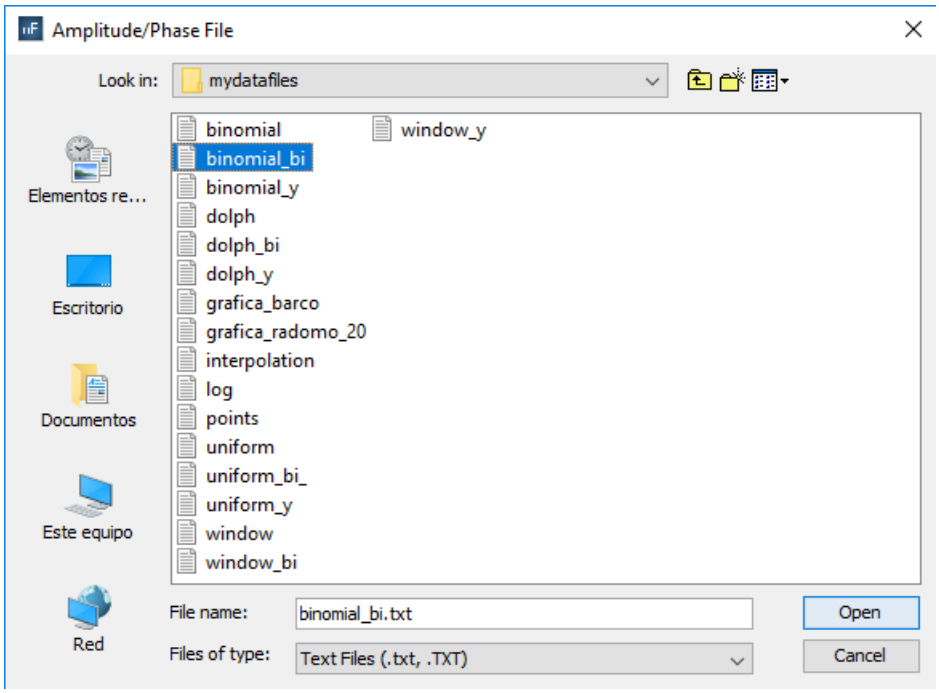


Figure 1554: Amplitude/Phase File panel

Select the corresponding file and save the feeding.

Step 5 Solver parameters

Select **Solver - Advanced Options** and set the parameters as shown.

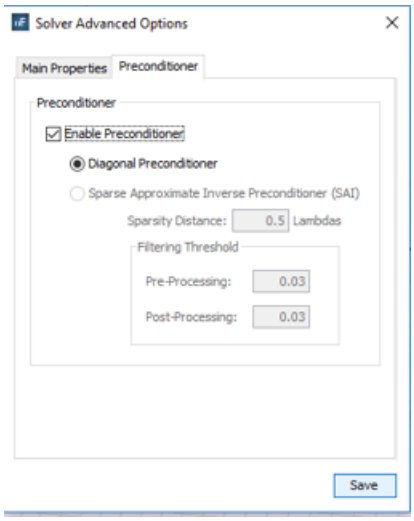


Figure 1555: Solver Advanced Options panel

Step 6 Meshing the geometry model.

Select **Mesh - Create Mesh** to open the meshing configuration panel and then set the parameters as show the next figure.

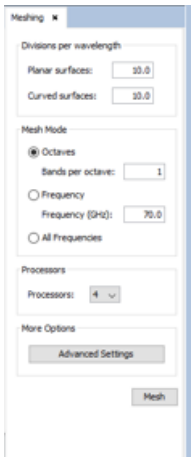


Figure 1556: Meshing panel

Step 7 Execute the simulation.

Select **Calculate - Execute** option to open simulation panel.

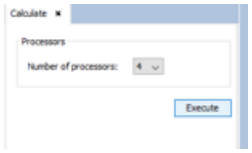


Figure 1557: Calculate panel

Step 8 Show Results

The radiation cuts can be visualized by clicking on **Show Results - Radiation Pattern - View Cuts.**

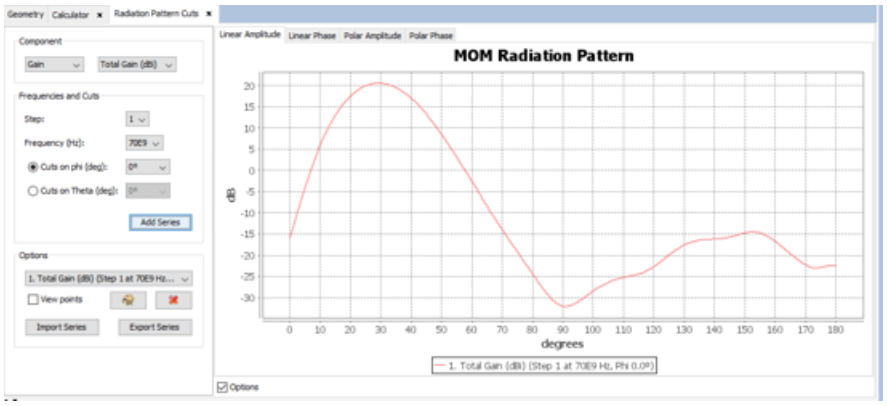


Figure 1558: Radiation Pattern Cuts

The radiation pattern can be visualized by clicking on **Show Results - Radiation Pattern - View 3D Pattern.**

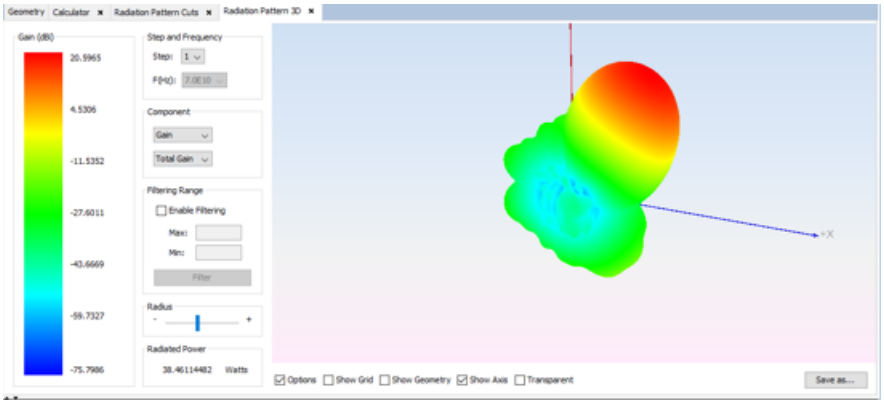


Figure 1559: Radiation Pattern 3D

11.15.11 Corrugated Horn Examples

Example 1: Design and Analyze a Corrugated Horn in the Band from 10 to 20 GHz

This case explains how to calculate far-field, radiation pattern, current density, charge density and near field of a corrugated horn.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

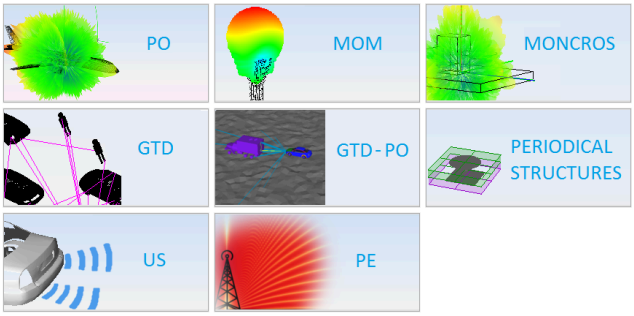


Figure 1560: New Project panel

Select MOM option on the previous figure and start to configure the project.

Step 2: Change the scale to millimeters.

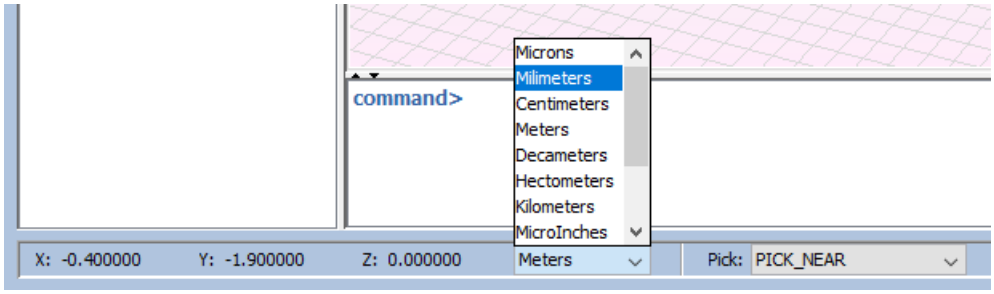


Figure 1561: Scale settings

Step 3: Create the geometry of the corrugated horn.

Execute the function " **corrugated_horn(fmin,fmax)**" (which can be downloaded). To execute the function, click on Tools - Calculator and write the call to the function.

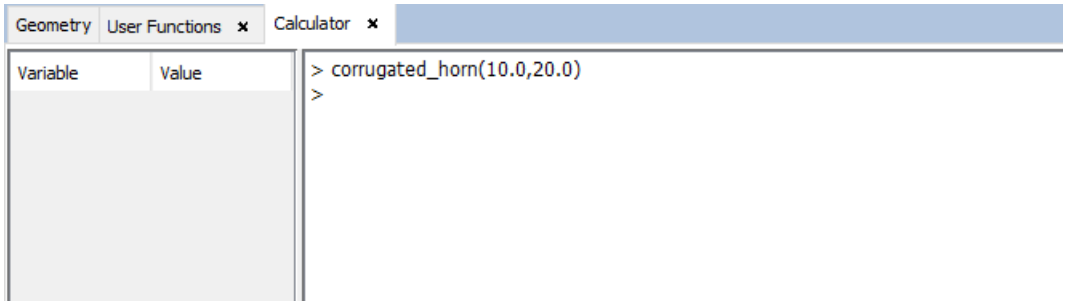


Figure 1562: Calculator panel

The parameters to set are:

- fmin is the lowest operating frequency (GHz)
- fmax is the highest operating frequency (GHz)

The script file, called `script_corrugated_horn.nfs`, will be automatically generated in the **mydatafiles** folder in the newFASANT directory.

The next step is to execute the generated script file. For that, click on Tools – Script - Load and open the script " **script_corrugated_horn.nfs**".

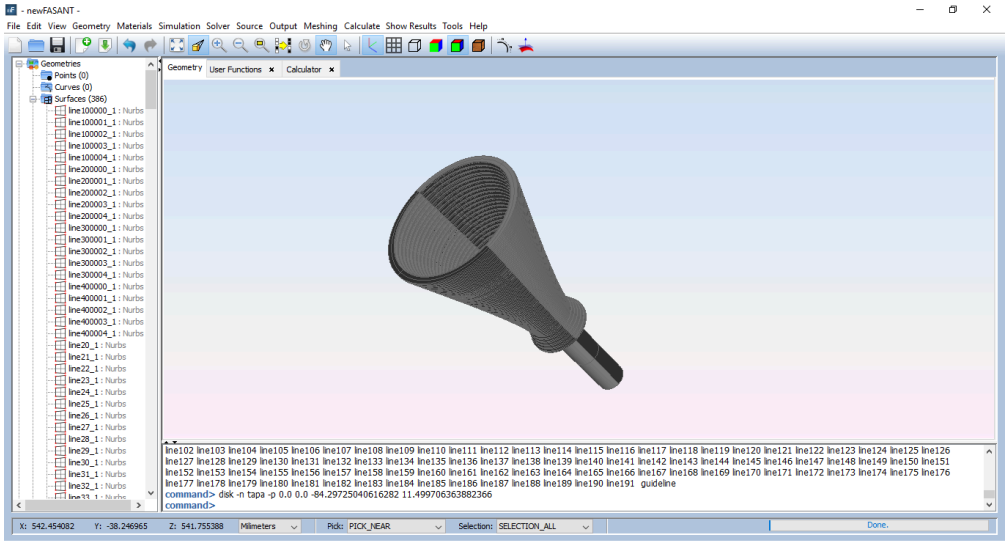


Figure 1563: Horn geometry has been generated.

Step 4: Set Simulation Parameters.

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

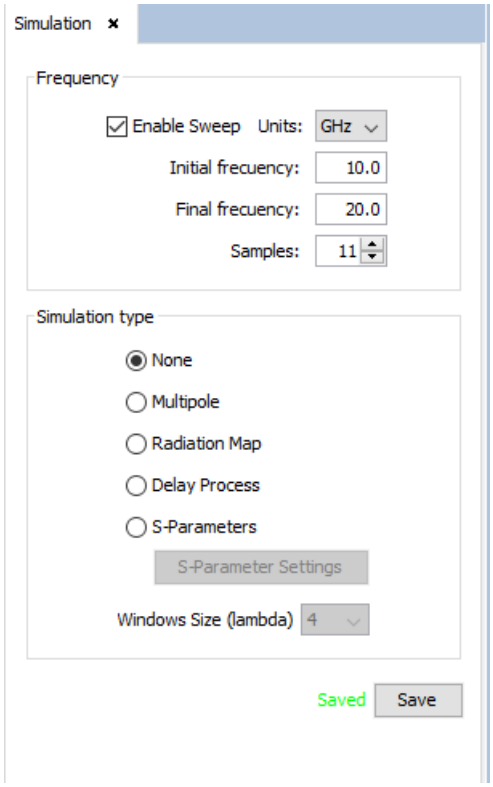


Figure 1564: Simulation Parameters panel

Step 5: Set the source parameters.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

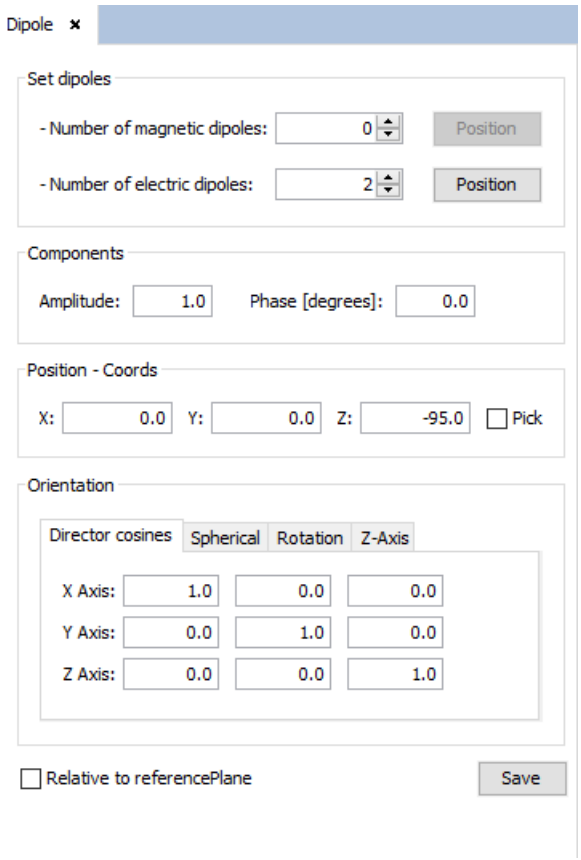


Figure 1565: Dipole Antenna panel

Select 2 electric dipoles and click on “Position”.

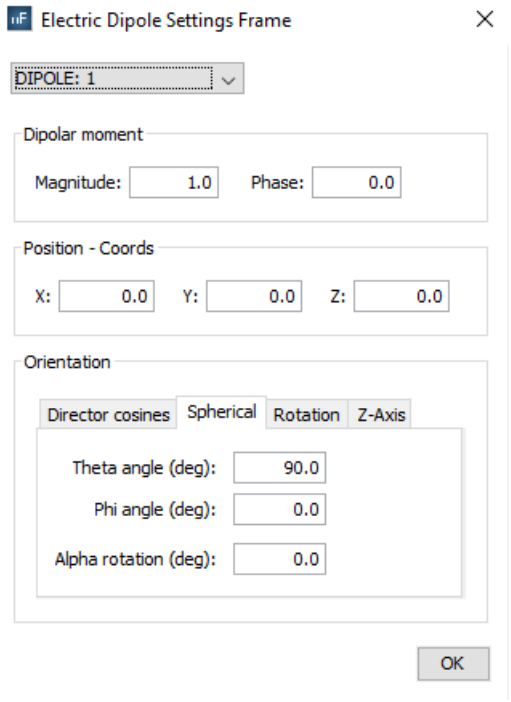


Figure 1566: Electric Dipole 1 Settings

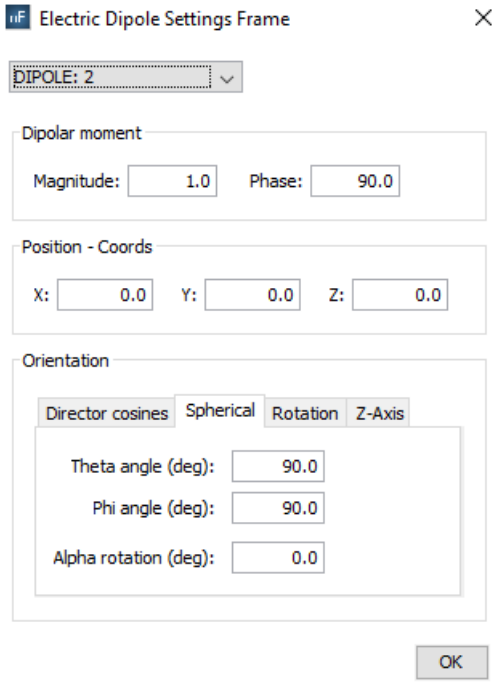


Figure 1567: Electric Dipole 2 Settings

Step 6: Set the solver parameters.

Click on Solver --> Parameters option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on Save button before going to next step.

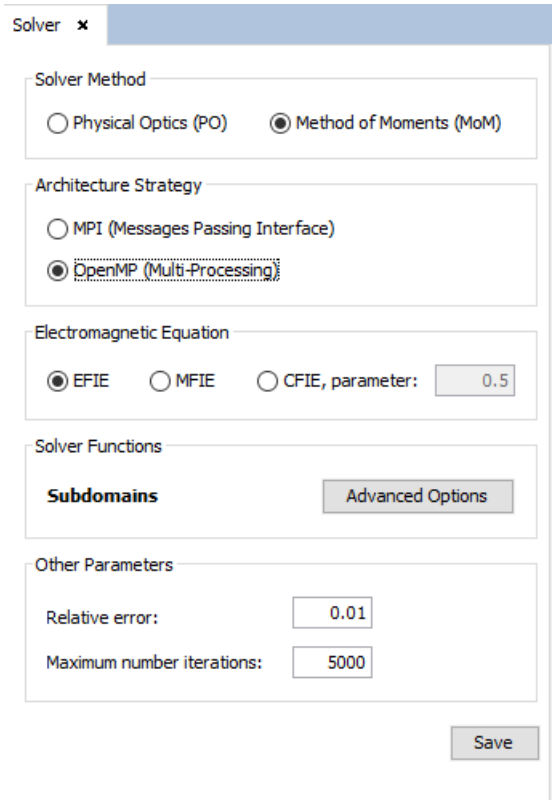


Figure 1568: Solver panel

Select 'Advanced Options' and activate Preconditioner as shown.

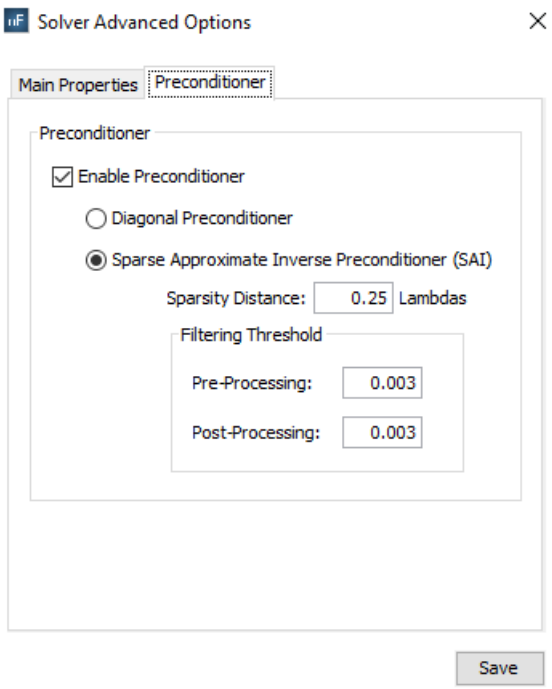


Figure 1569: Preconditioner panel

Step 7: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing, it is recommended to run the process of meshing with the number of physical processors available to the machine.

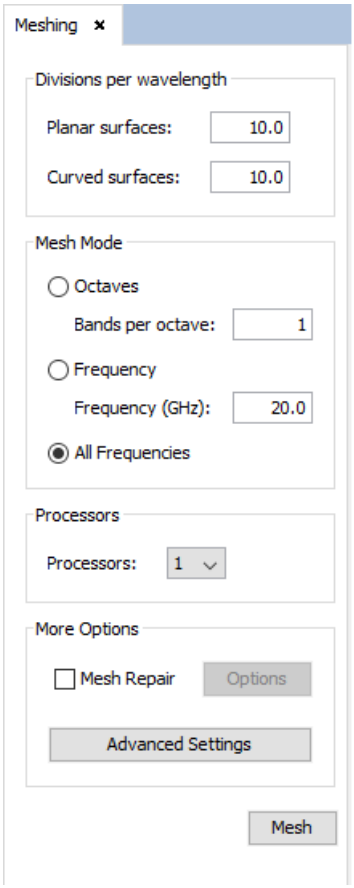


Figure 1570: Meshing panel

The user can choose between:

- **octaves** an automatic frequency range per octave is performed, that depends on the Initial and Final frequency. The user can choose 1 frequency by an octave or several frequencies by an octave. The more frequencies per octave, the greater the precision
- **all frequencies** a meshing by each frequency is built, so it is the more accurate option. **This option will only be chosen when the user needs a very accurate mesh**.

Step 8: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

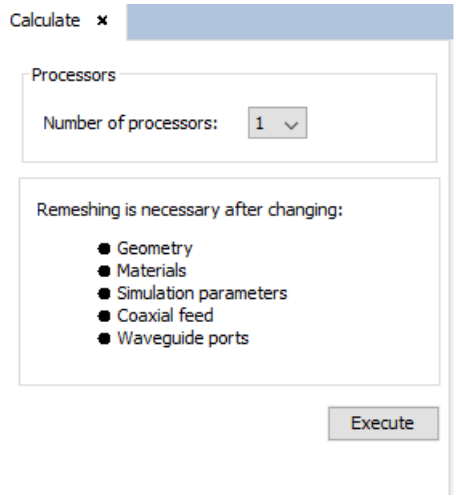


Figure 1571: Execute panel

Then click on 'Execute' button to starting the simulation.

Step 9: Show Results.

To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select **'Show Results --> Far Field --> View Cuts'** option to show the cuts of the observation directions options.

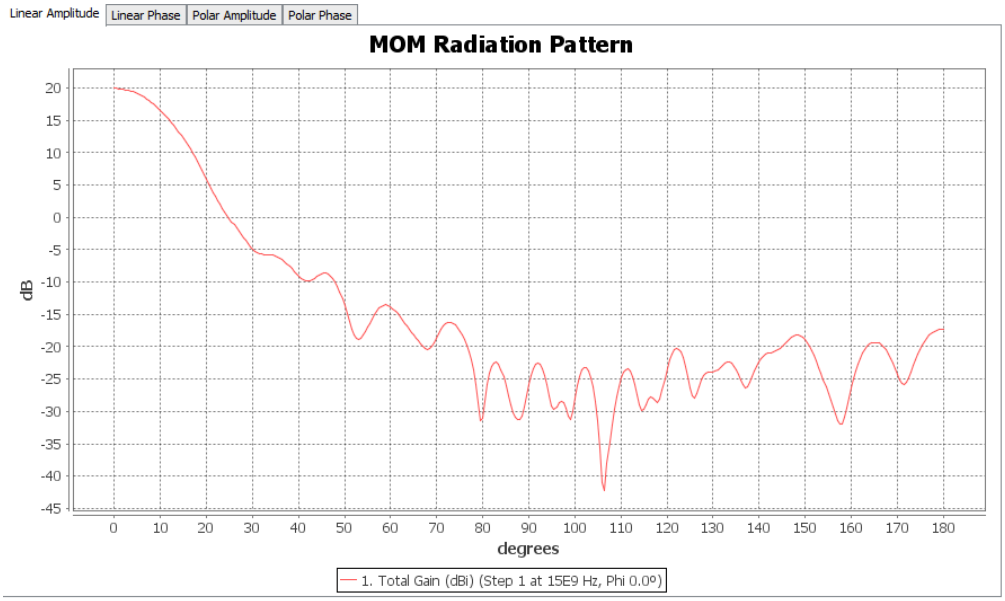


Figure 1572: Far Field cuts - Linear Amplitude

Selecting other values for the component, step, frequency or cut parameters and clicking on **'Add Series'** button a new cut will be added to the selected parameters.

On **'Show Results --> Far Field'** menu, other results are present such as **'View Cuts by Step'** and **'View Cuts by Frequency'** and this option display the values for one selected point for each step or frequency.

On **'Show Results --> Radiation Pattern'** menu, other results are present such as **'View Cuts by Step' and 'View Cuts by Frequency'** and this option display the values for one selected point for each step or frequency.

Select **'Show Results --> Radiation Pattern --> View 3D Pattern'** option to show the cuts of the radiation pattern options.

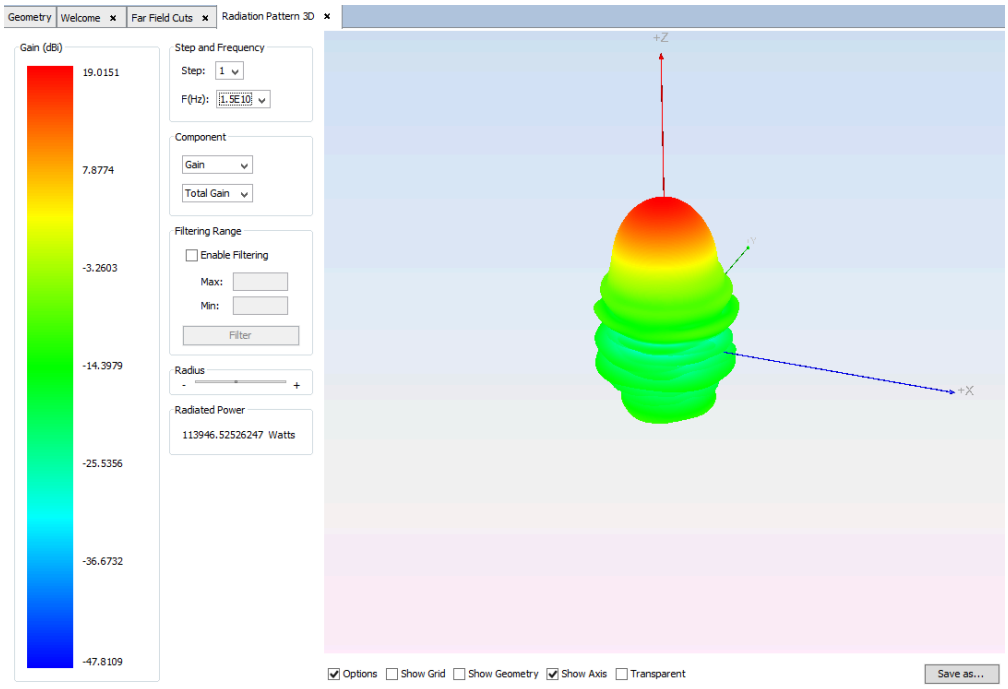


Figure 1573: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select **'Show Results --> View Currents'** option to show the current density.

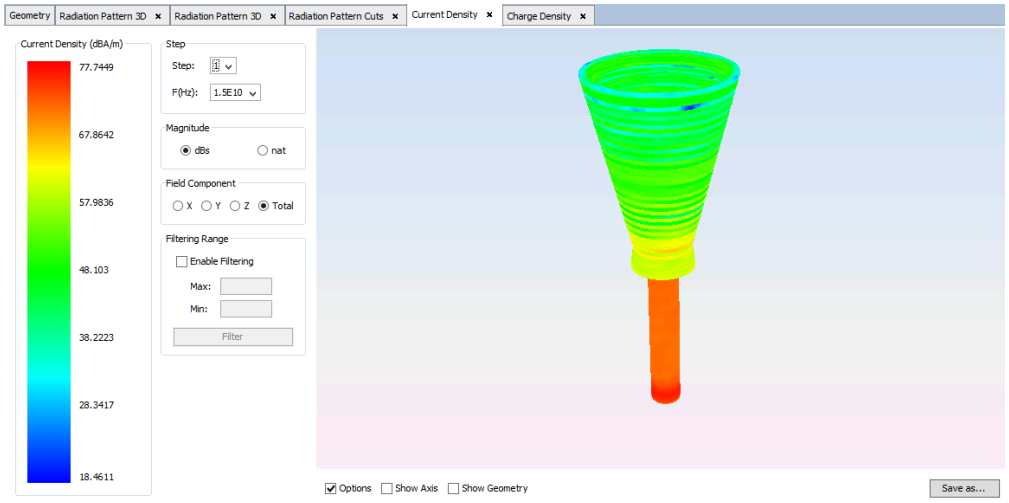


Figure 1574: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

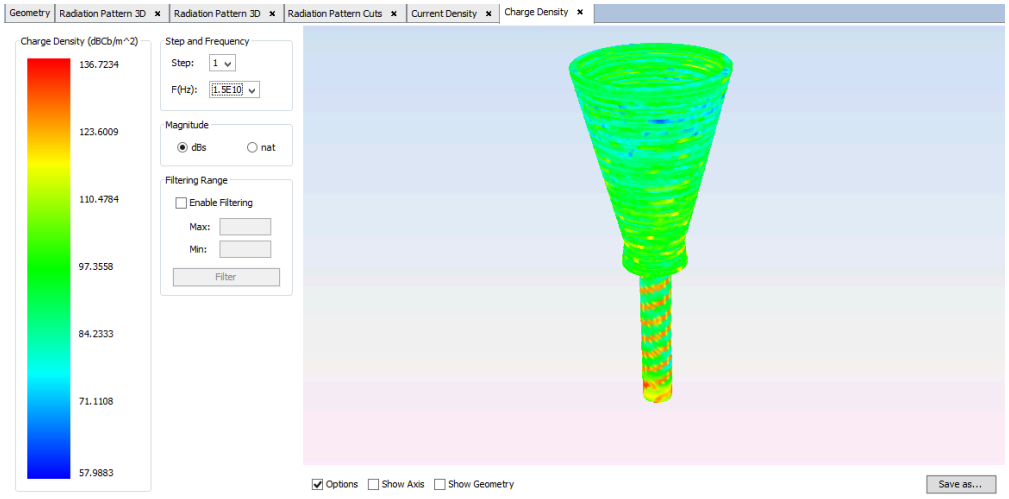


Figure 1575: Charge Density

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

On **'Show Results --> Far Field'** menu, other results are present such as **'View Cuts by Step'** and **'View Cuts by Frequency'** and this option display the values for one selected point for each step or frequency.

Example 2: Design and Analyze a Conical Corrugated Horn at 10 GHz and 22 dB Gain

This case explains how to calculate far-field, radiation pattern, current density, charge density and near field of a corrugated horn with 22 dB gain at 10 GHz.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

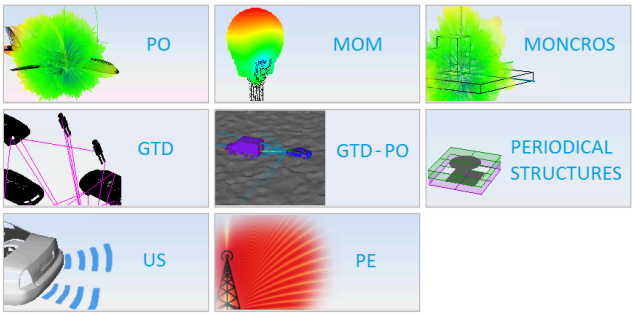


Figure 1576: New Project panel

Select MOM option on the previous figure and start to configure the project.

Step 2: Change the scale to millimeters.

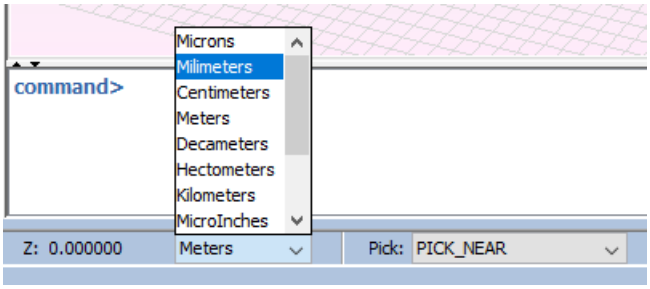


Figure 1577: Scale settings

Step 3: Create the geometry of the corrugated horn.

Execute the function “corrugated_horn_gain(fmin,fmax,alfa,D)". To execute the function, click on Tools - Calculator and write the call to the function.

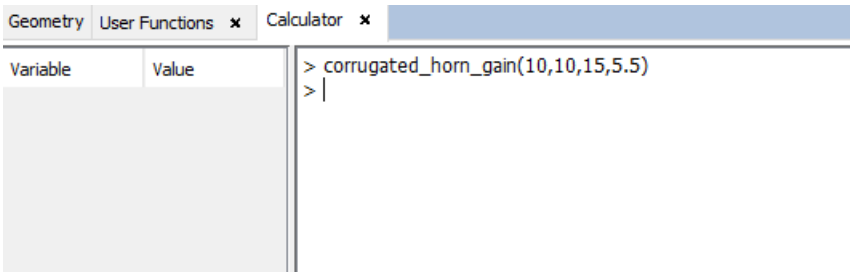


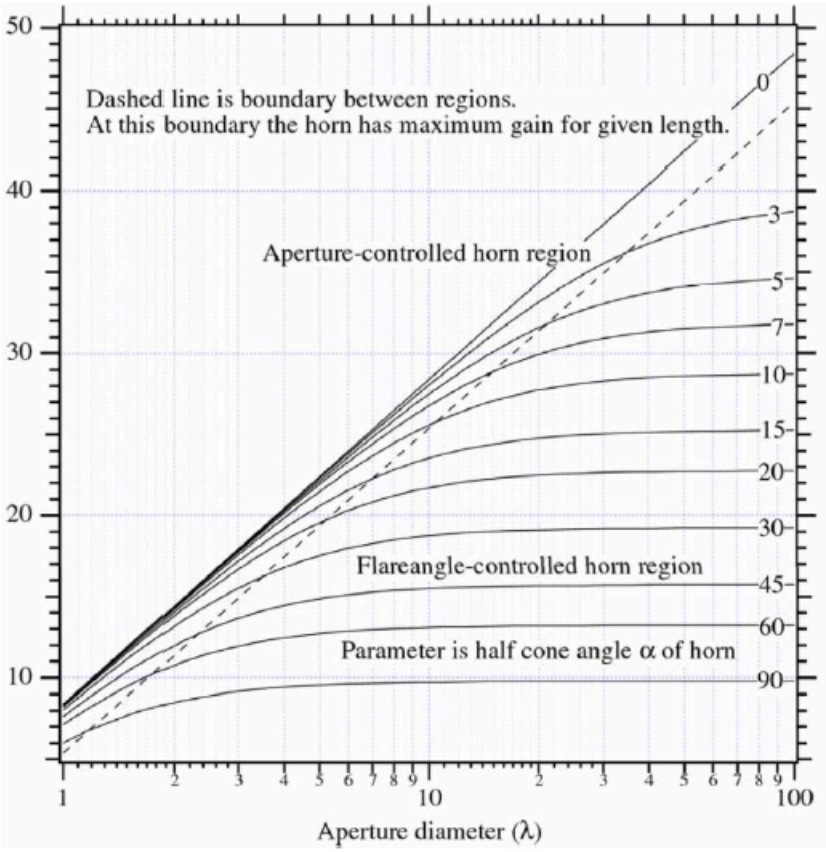
Figure 1578: Calculator panel

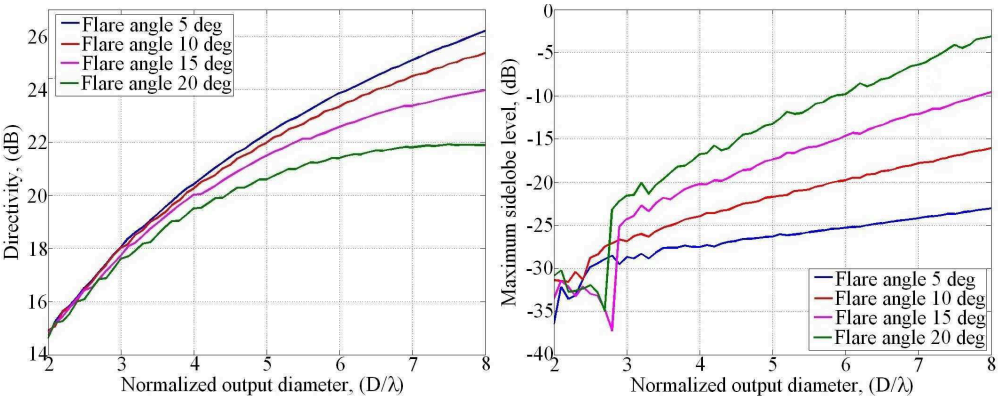
The parameters to set are:

- fmin is the lowest operating frequency (GHz)
- fmax is the highest operating frequency (GHz)
- alfa is the flare angle (obtained from the following graph)
- D is the aperture diameter (obtained from the following graph)

In this case, we will simulate at 10 GHz, so the maximum and minimum frequency are 10 GHz.

To choose the flare angle and the aperture diameter, observe the next graph.





For a gain of 20 dB a flare angle of 15° and a diameter of 5.5 times, λc is chosen. So:

- Alpha = 15

- D = 5.5

The script file, called " script_corrugated_horn_gain.nfs", will be automatically generated in the **mydatafiles** folder in the newFASANT directory.

The next step is to execute the generated script file. For that, click on Tools – Script - Load and open the script `script_corrugated_horn_gain.nfs`.

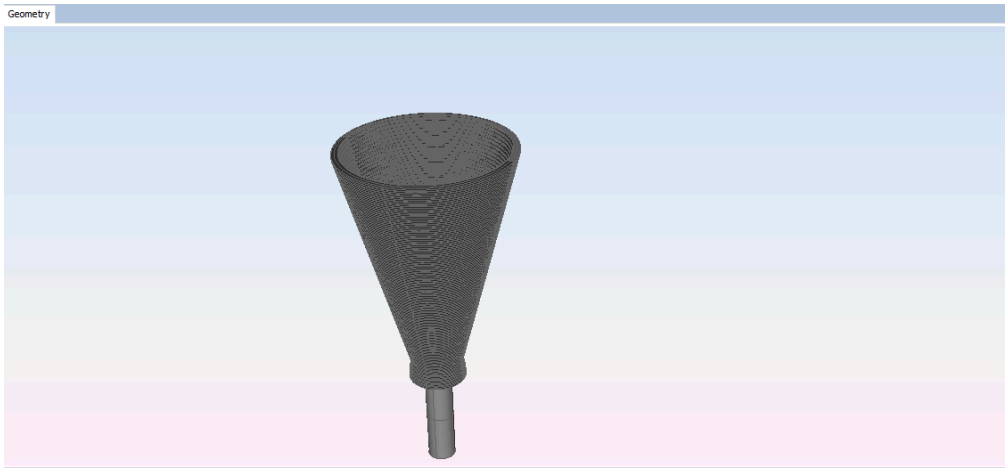


Figure 1579: Horn geometry has been generated.

Step 4: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

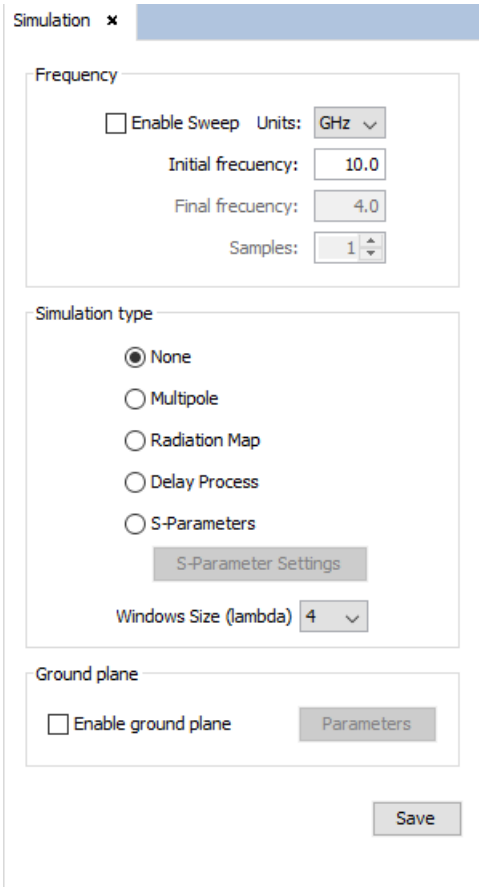


Figure 1580: Simulation Parameters panel

Step 5: Set the source parameters.

Select 'Source --> Dipole --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the dipole appears.

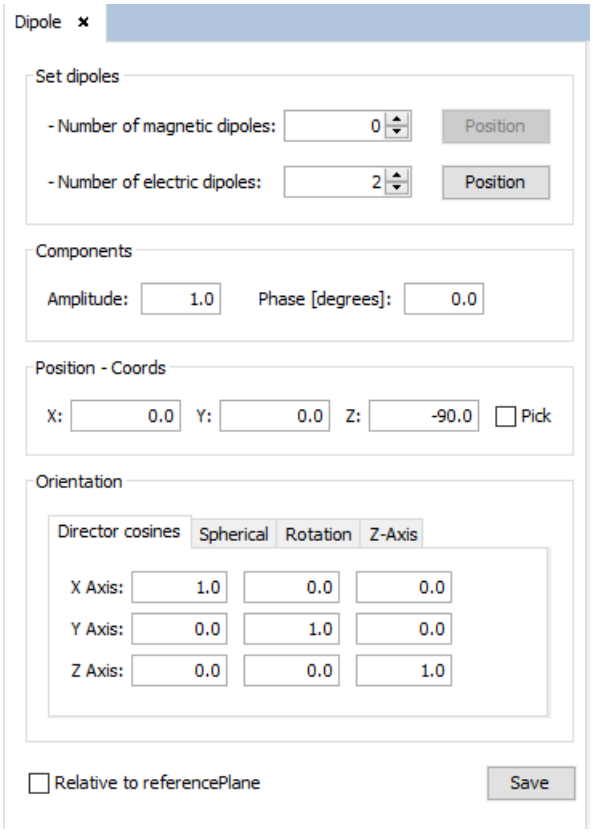


Figure 1581: Dipole Antenna panel

Select 2 electric dipoles and click on “Position”.

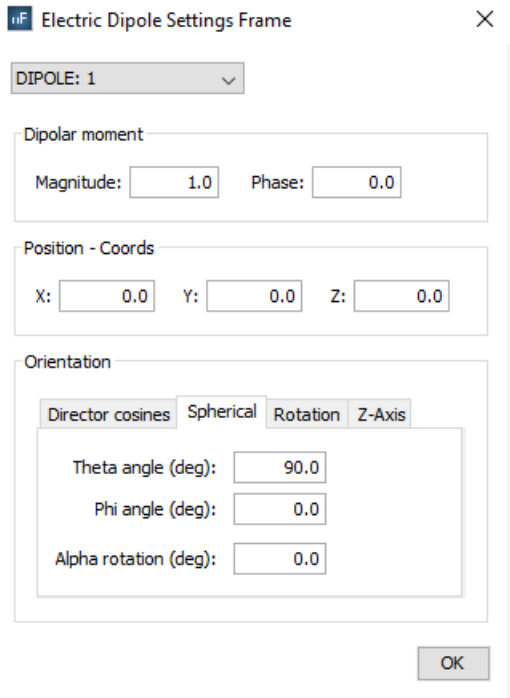


Figure 1582: Electric Dipole 1 Settings

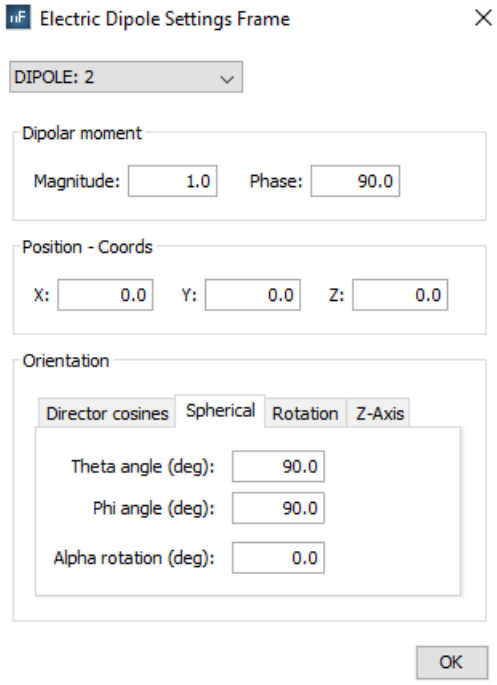
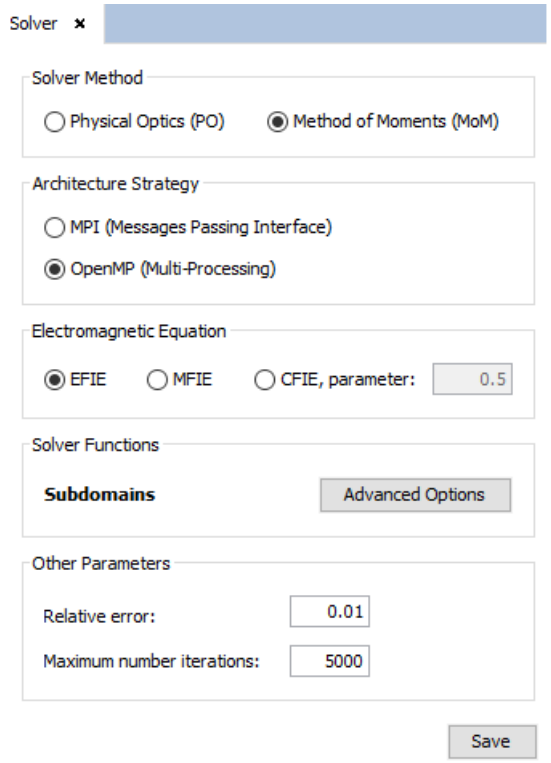


Figure 1583: Electric Dipole 2 Settings

Step 6: Set the solver parameters. Click on Solver --> Parameters option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on Save button before going to next step.



Select 'Advanced Options' and activate Preconditioner as shown.

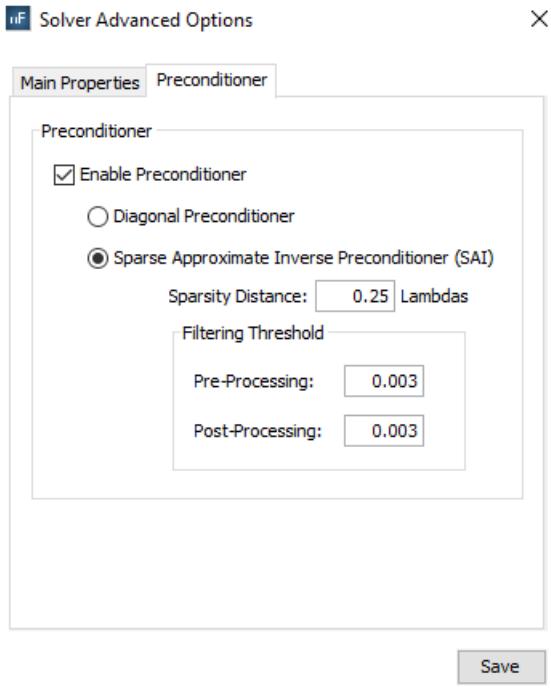


Figure 1584: Preconditioner panel

Step 7: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing, it is recommended to run the process of meshing with the number of physical processors available to the machine.

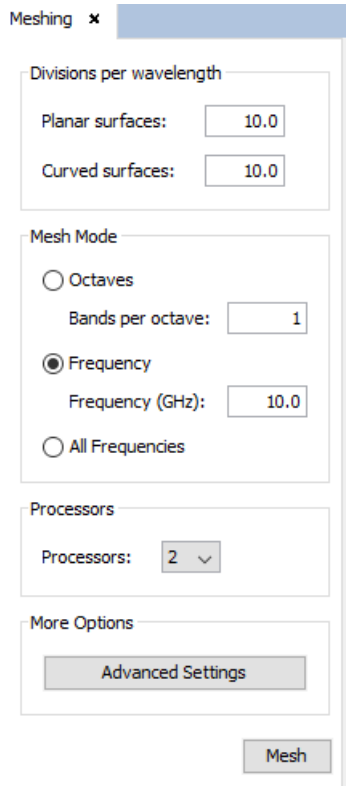


Figure 1585: Meshing panel

Step 8: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

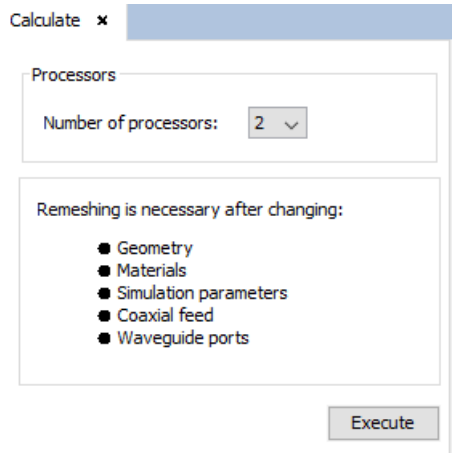


Figure 1586: Execute panel

Then click on 'Execute' button to starting the simulation.

Step 9: Show Results.

To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options.

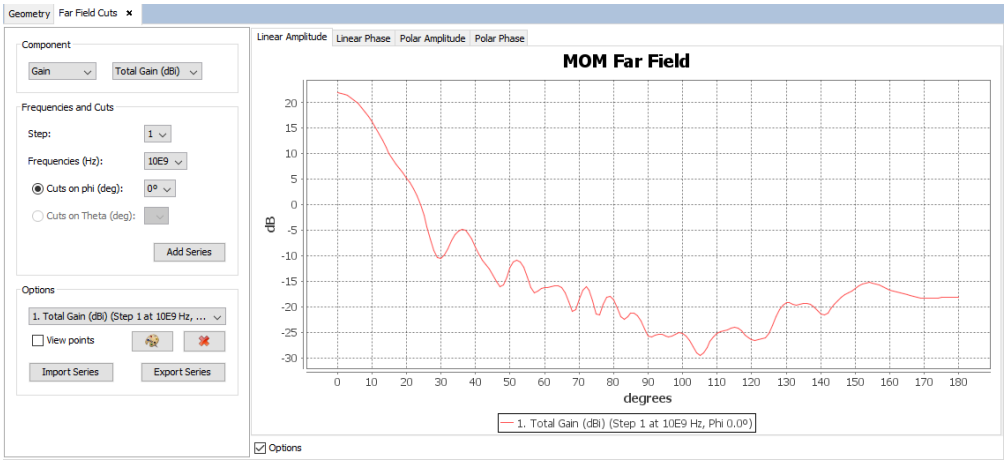


Figure 1587: Far Field cuts - Linear Amplitude

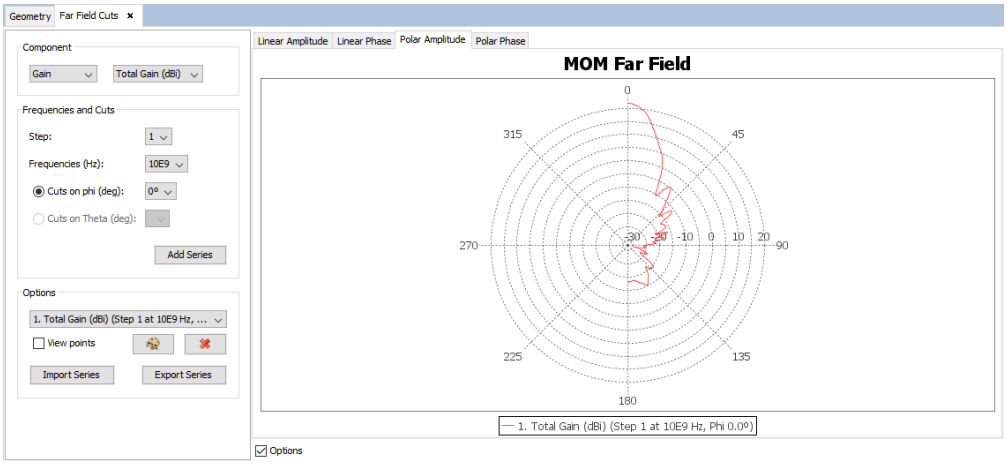


Figure 1588: Far Field cuts - Polar Amplitude

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters.

On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View Cuts' option to show the cuts of the radiation pattern options.

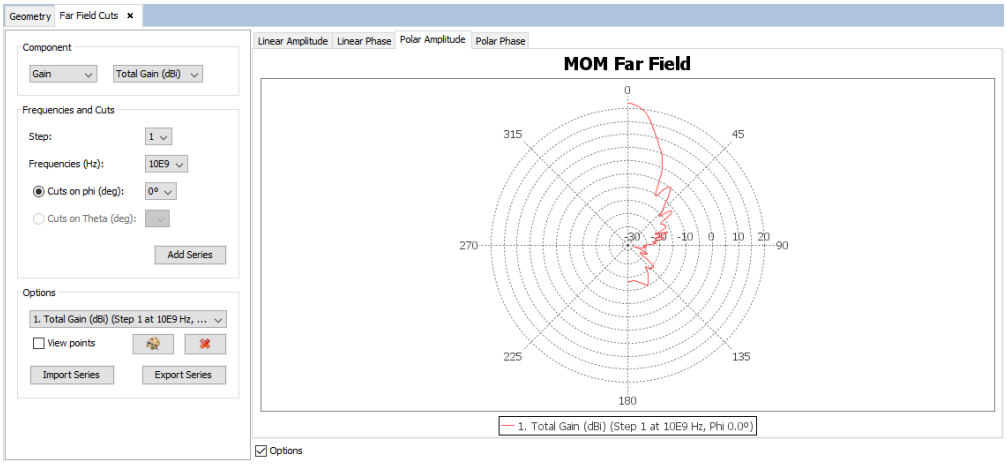


Figure 1589: Radiation Pattern cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters.

On 'Show Results --> Radiation Pattern' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

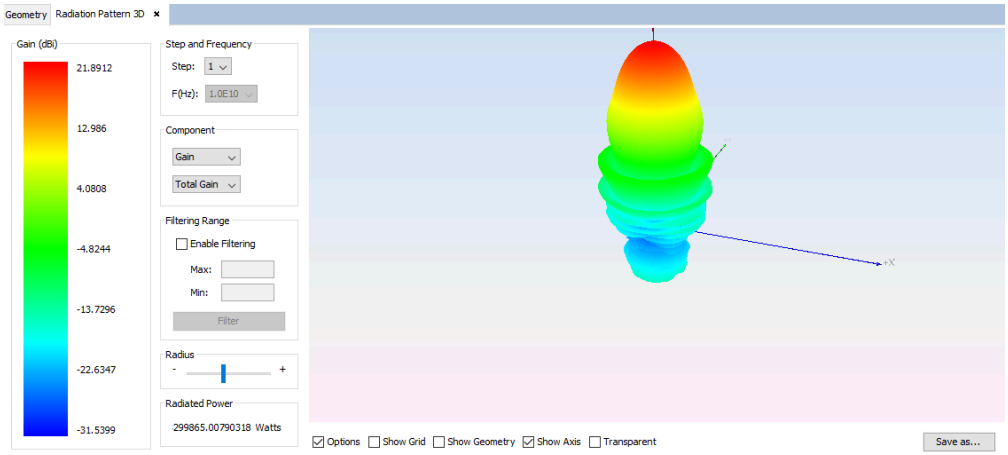


Figure 1590: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Currents' option to show the current density.

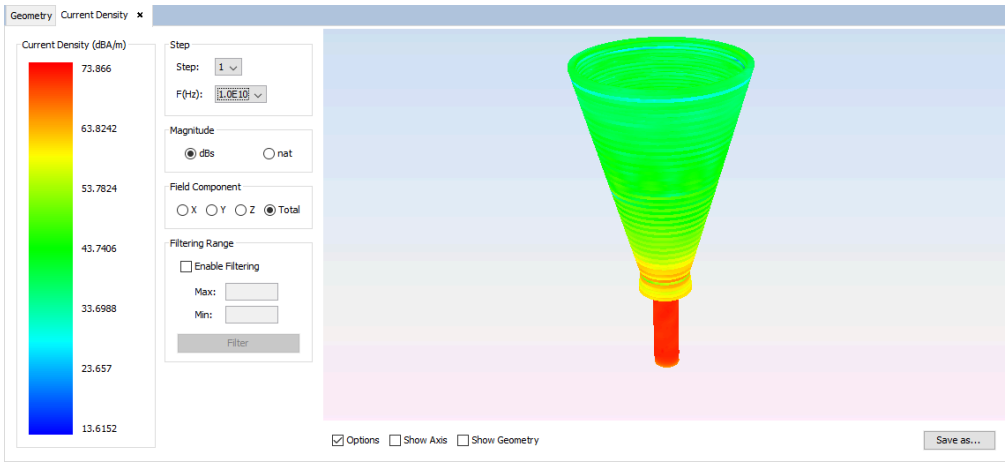


Figure 1591: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Select 'Show Results --> View Charges' option to show the charge density.

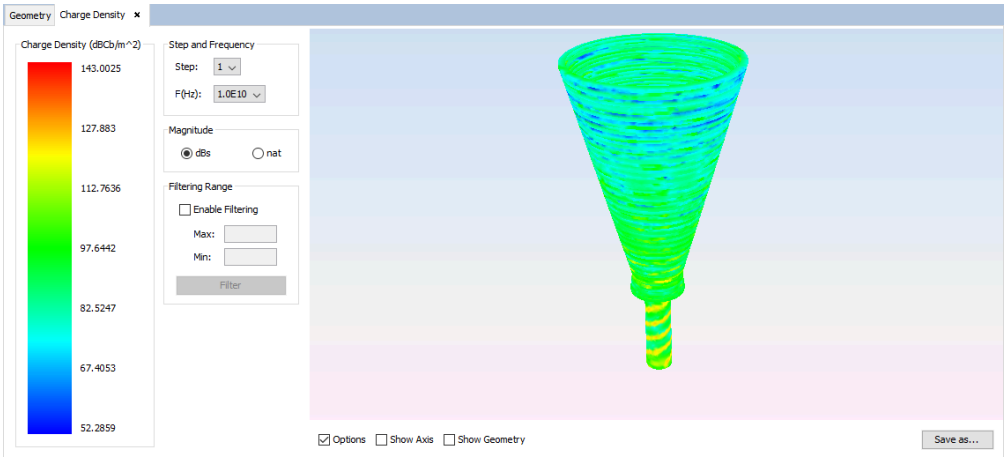


Figure 1592: Charge Density

Changing values for step, frequency, magnitude or filtering parameters the visualization for the new parameters will be shown.

On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Example 3: Design of a Corrugated Horn with a Customized Profile

This case explains how to add a new profile of the corrugated horn.

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

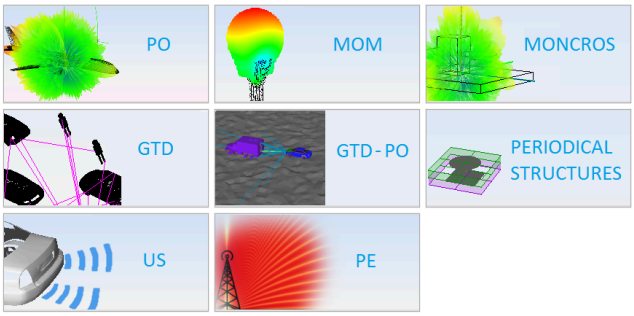


Figure 1593: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Open the function.

Select 'Tools --> User Functions' option on the menu bar and open the function "corrugated_horn.java" or "corrugated_horn_gain.java".

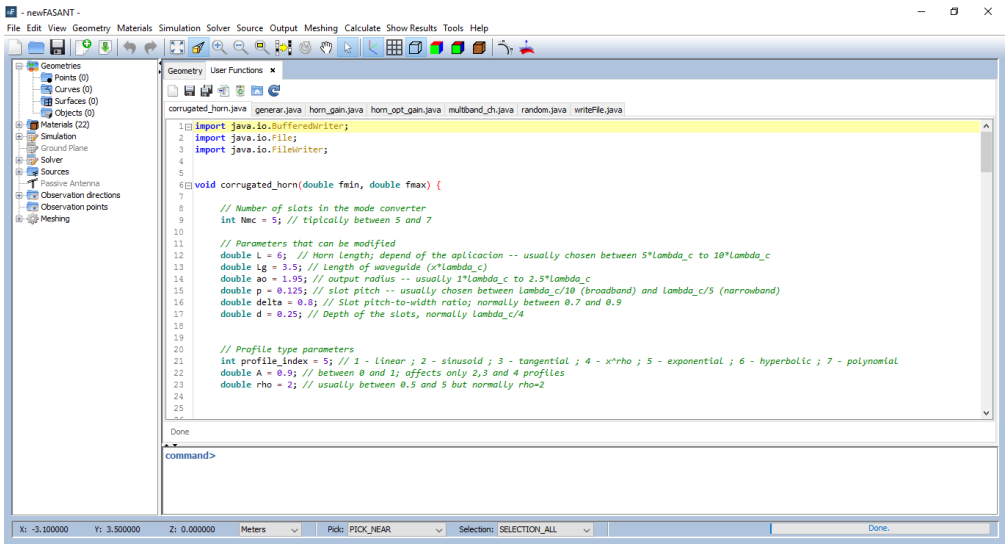


Figure 1594: User Functions panel

Look for the 'try' exception and the IF ... ELSE IF statement with the "profile_index" parameter in the section 'Corrugated surface profile formulations':

Figure 1595: Exception 'try' and if...else statement

The last 2 profiles (profile_index==8 and profile_index==9) are reserved for the user profile formulation. User profile 1 and User profile 2.

Step 3: Introduce the profile formulation:

In this example, we will establish a linear profile in user profile 1 (profile_index==8).

The linear profile has the following formulation:

$$a(z) = a_i + (a_o - a_i)\frac{z}{L} \qquad (10)$$

where:

- ai: input radius

- ao: output radius
- L: length
- z in the variable parameter. It is necessary to enter it as **i*p**

We look for the 'User profile1' within the function:

Figure 1596: User profile 1 and 2

And write our function, which in this case is the linear profile:

az[i]=ai+(ao-ai)*(i*p/L);

Note that 'z' has been changed to 'i*p'.

Step 4: Select the user profile 1.

To activate the profile entered, we have to select it within 'Profile Type Parameters' at the beginning of the function. In this case we select the profile index number 8 (profile_index = 8;)

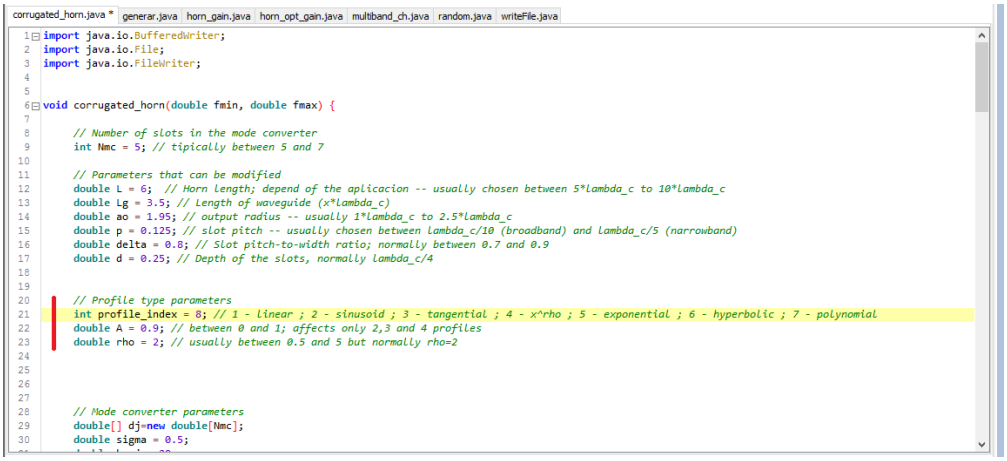


Figure 1597: Select the profile index

Step 5: Save the function.

Finally, save the changes. Save it clicking on 'SAVE' or 'SAVE ALL' button.

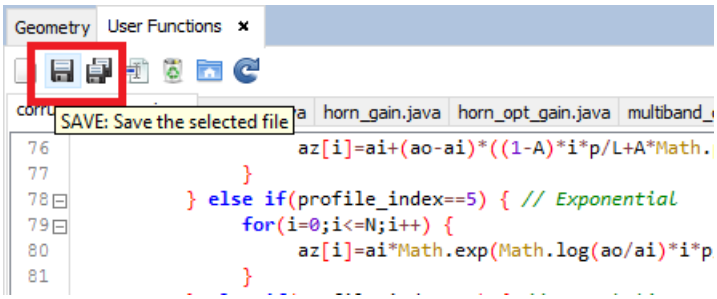
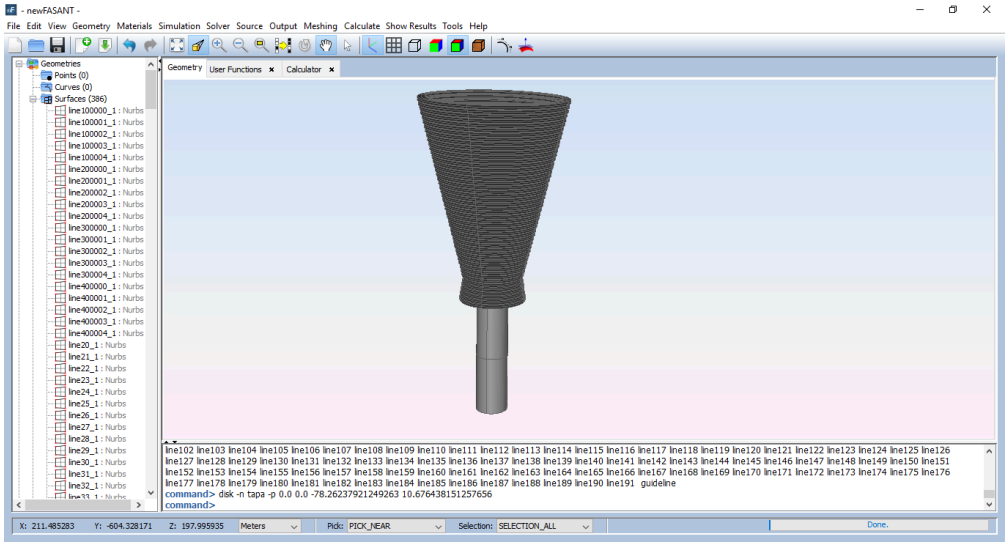


Figure 1598: Save the function

Step 6: Show Results

The corrugated horn with linear profile has been generated.



To simulate it, follow Example 1: Analysis of a Waveguide with 12 Rotated Slots or Example 2: Design of an Array of Slotted Waveguides, depending on the user function where the new profile has been established.

Example 4: Compute the S-Parameters of a Corrugated Horn in the Band from 10 to 20 GHz

This case explains how to calculate S-parameters of a corrugated horn.

Step 1: **Create a new MOM Project**.

Open newFASANT and select 'File --> New' option.

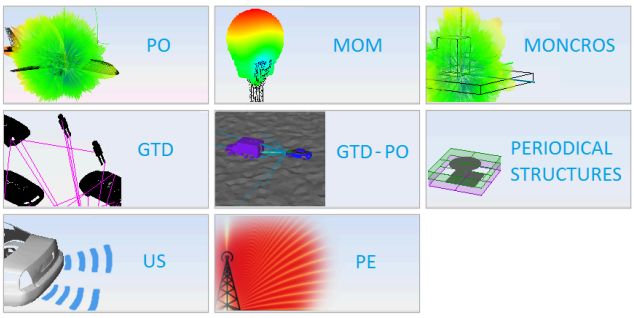


Figure 1599: New Project panel

Select MOM option on the previous figure and start to configure the project.

Step 2: **Change the scale to millimeters**.

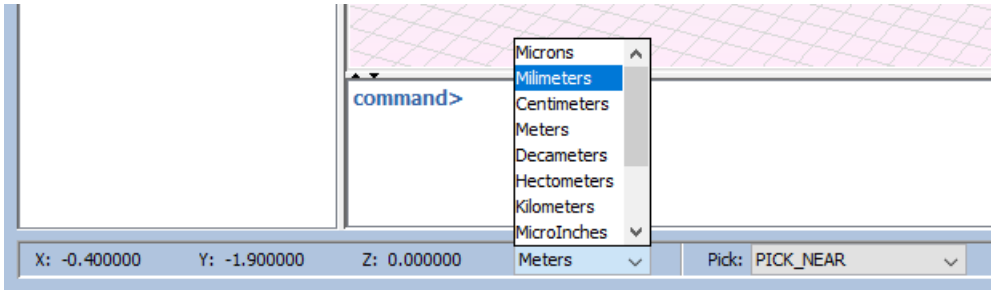


Figure 1600: Scale settings

Step 3: Create the geometry of the corrugated horn.

Execute the function " **corrugated_horn(fmin,fmax)**" (which can be downloaded). To execute the function, click on Tools - Calculator and write the call to the function.

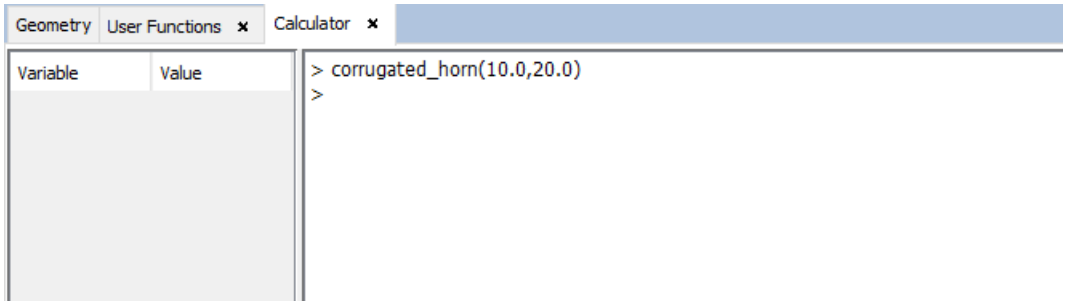


Figure 1601: Calculator panel

The parameters to set are:

- fmin is the lowest operating frequency (GHz)
- fmax is the highest operating frequency (GHz)

The script file, called `script_corrugated_horn.nfs`, will be automatically generated in the **mydatafiles**folder in the newFASANT directory.

The next step is to execute the generated script file. For that, click on Tools – Script - Load and open the script `script_corrugated_horn.nfs`.

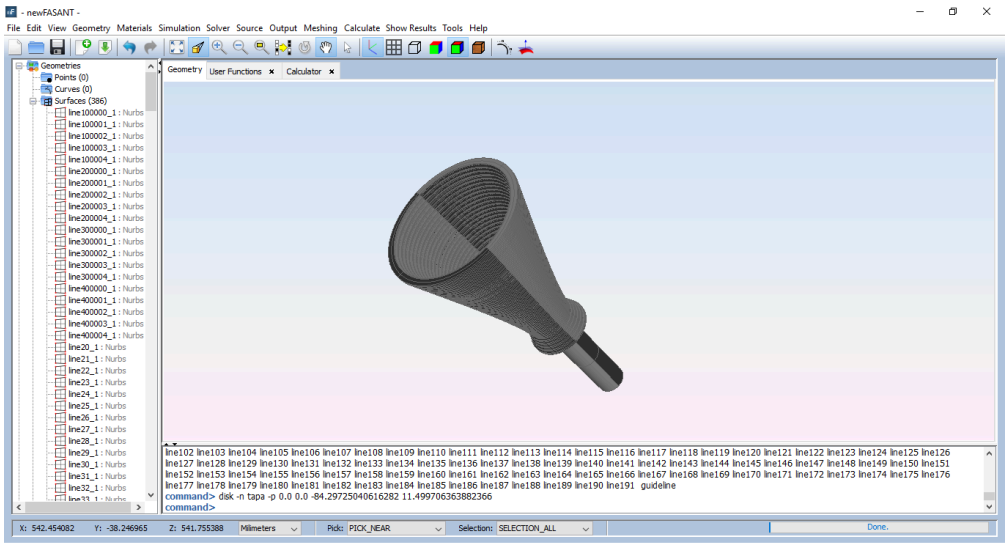


Figure 1602: Horn geometry has been generated.

Step 4: Set Simulation Parameters

Select **Simulation --> Parameters** option on the menu bar and the following panel appears. Set a **Frequency Sweep** and the **S-Parameters Simulation type** as shown in the figure below. Remember clicking on **Save** button to confirm the changes.

Step 5: **Set the source parameters**.

Note this option requires that the geometry must be only one object (use the group command if it is not) and the waveguide cap needs to be removed

To add a waveguide port, select the object and click on 'Source' --> 'Waveguides' --> 'Add Waveguides Port' menu to open the panel.

And follow the steps described in Add Waveguide Port.

Step 6: **Set the solver parameters**.

Click on **Solver --> Parameters** option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on Save button before going to next step.

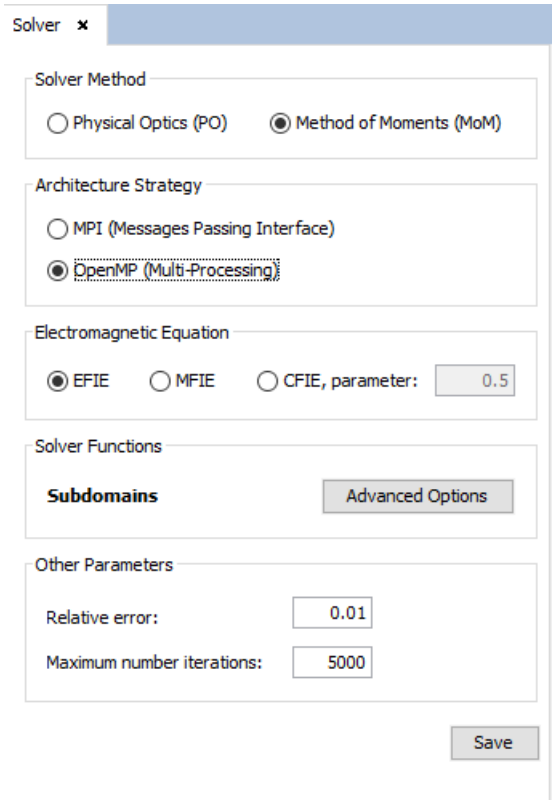


Figure 1603: Solver panel

Select 'Advanced Options' and activate Preconditioner as shown.

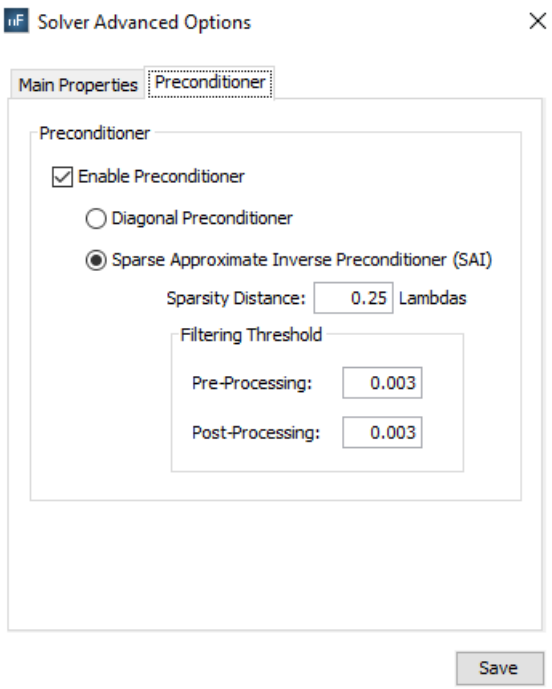


Figure 1604: Preconditioner panel

Step 7: Meshing the geometry model.

Select **'Meshing --> Parameters'** to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing, it is recommended to run the process of meshing with the number of physical processors available to the machine.

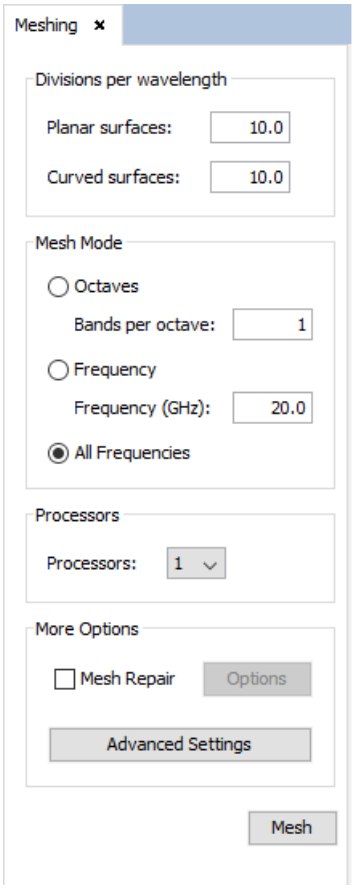


Figure 1605: Meshing panel

The user can choose between:

- **octaves**: an automatic frequency range per octave is performed, that depends on the Initial and Final frequency. The user can choose 1 frequency by an octave or several frequencies by an octave. **The more frequencies per octave, the greater the precision**
- **all frequencies**: a meshing by each frequency is built, so it is the more accurate option. **This option will only be chosen when the user needs a very accurate mesh.**

Step 8: **Execute the simulation**.

Select **'Calculate --> Execute'** option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

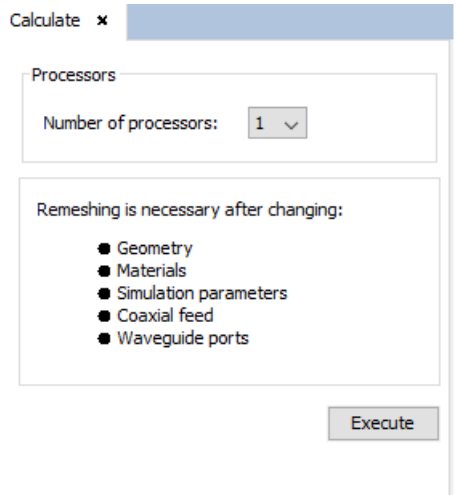


Figure 1606: Execute panel

Then click on 'Execute' button to starting the simulation.

Step 9: **Show Results**.

To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

When the simulation process has finished, the S-Parameters are enabled within the results menu. Click on **Show Results menu and S-Parameters option** to visualize these parameters. The panel with the available options to visualize the S-Parameters is open on the right side.

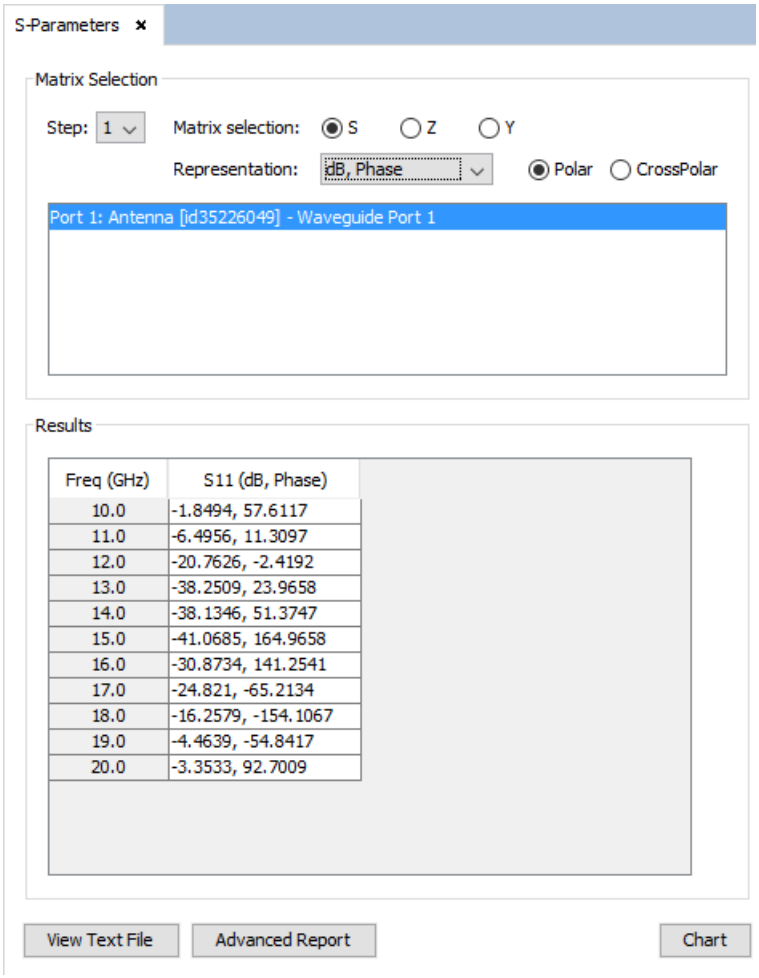


Figure 1607: S-Parameters parameters

Select the S-Matrix Representation and the option db, Phase in Representation section. Then, select the column to represent in the Results table and then click on Chart button to add them to plot them.

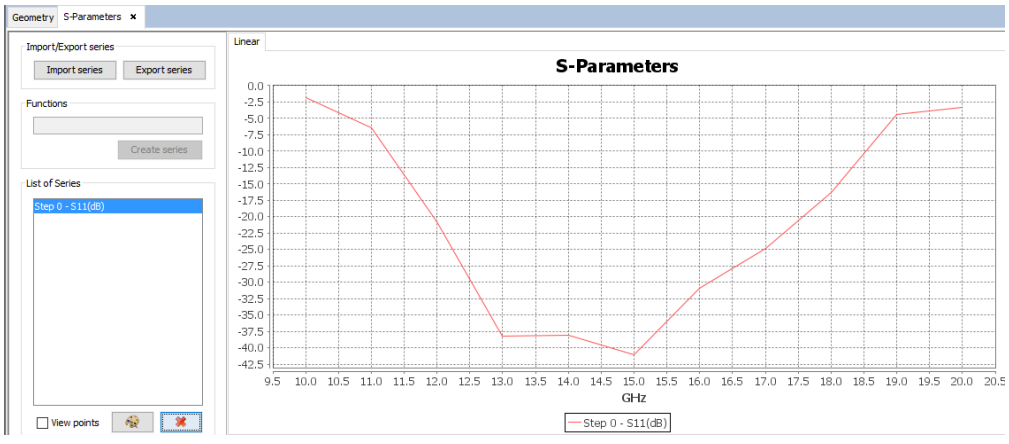


Figure 1608: S11parameters plot

Freq (GHz)	S11 (dB, Phase)
10.0	-1.8494, 57.6117
11.0	-6.4956, 11.3097
12.0	-20.7626, -2.4192
13.0	-38.2509, 23.9658
14.0	-38.1346, 51.3747
15.0	-41.0685, 164.9658
16.0	-30.8734, 141.2541
17.0	-24.821, -65.2134
18.0	-16.2579, -154.1067
19.0	-4.4639, -54.8417
20.0	-3.3533, 92.7009

Figure 1609: S11 parameters table

11.15.12 Design of Waveguides

Example 1: Analysis of a Waveguide with 12 Rotated Slots

This case explains how to create and calculate far-field, radiation pattern, current density, charge density and near field of a 12 Slots Feeding Waveguide.

Step 1: Create a New **MOM Project**.

Open **newFASANT** and select ' **File --> New**' option.

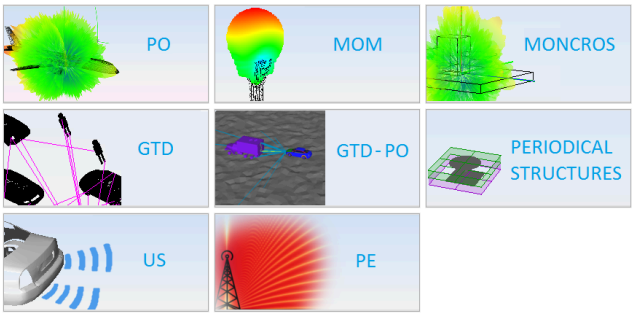


Figure 1610: Figure New Panel Project

Select **MOM** option on the previous figure and start to configure the project.

Step 2: Change the scale to cm.

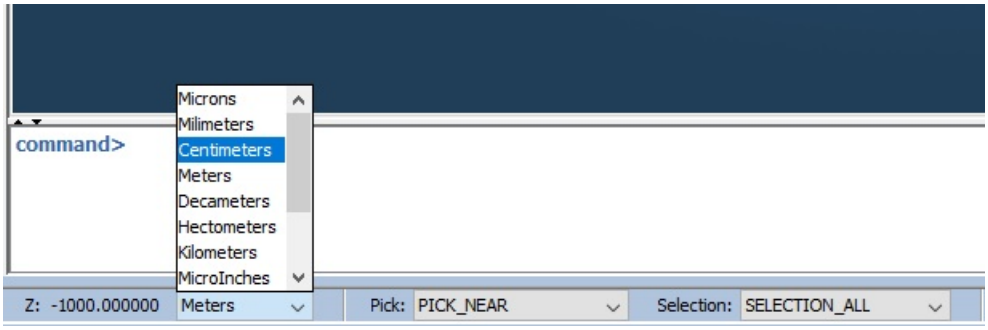


Figure 1611: Scale settings

Step 3: Open the function.

Select **Tools** --> **User Functions** option on the menu bar and open the function **SlottedWaveguide.java**.

```
import java.io.BufferedWriter;
import java.io.File;
import java.io.FileWriter;
import java.io.IOException;
import java.util.logging.Level;
import java.util.logging.Logger;


    void SlottedWaveguide(int N,double F,double angulo,double slotGap) {

        // Parameters that can be modified


        double f = F*1.0E9;    // design frequency
        double c = 3.0E10;    // light speed(cm)


        // Static parameters
        double cte = 10.0/F;
        double lambda = c / f;
        double a = lambda/ Math.sqrt(2);   // waveguide width
        double b = 0.4798*a;     // waveguide heigth
        double fc = c / (2.0 * a);
```

Figure 1612: User Function Code

```
        // Parameters that can be modified


        double f = F*1.0E9;    // design frequency
        double c = 3.0E10;    // light speed(cm)


        // Static parameters
        double cte = 10.0/F;
        double lambda = c / f;
        double a = lambda/ Math.sqrt(2);   // waveguide width
        double b = 0.4798*a;     // waveguide heigth
        double fc = c / (2.0 * a);
        double lambdag = lambda / Math.sqrt(1 - Math.pow(fc / f, 2.0));
        double Length = (N - 1) * (lambdag / 2.0) + (6.0 * lambdag / 4.0);
        double L = lambdag / 2.0;


        double SlotLength = 0.81 * (lambda / 2.0);
        double SlotThin = 0.05 * lambda;
        double dStart = 3 * (lambdag / 4.0);   //distance from the end of the waveguide to the first slot center
        double SlotGap = cte*slotGap;
```

Figure 1613: User Function Code

The parameter **a** represents the width of the guides and it is calculated to be equal to lambdag **/ 2**.

Step 4: Configure the parameters of your Project.

Select **Tools --> Calculator** option on the menu bar and write **SlottedWaveguide(N,F,angle,slotgap)** on the command line.

N is the number of slots, F is the design frequency value in GHz, **angle** is the rotation angle of the slots and **slot gap** is the distance between the slot's center and the meridian of the upper face of the waveguide.

In our case, as we want a 12-slot feeding waveguide **N= 12, F=10 Ghz, angle=0 and slotgap = 0.243953 cm**

Although in this example the slots aren't rotated, the user function allows to rotate them in phase and in counter phase alternately.

The script file, called `script_waveguide.nfs`, will be automatically generated in the `mydatafiles` folder in the newFASANT directory.

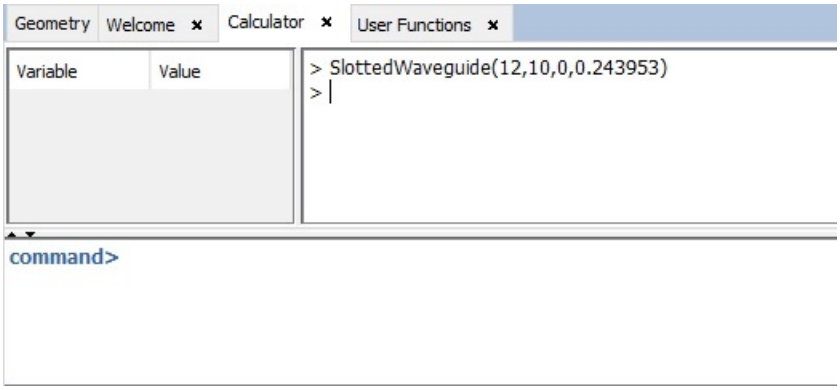


Figure 1614: Calculator panel

Step 5: Load the generated Script.

The next step is to execute the generated script file. For that, click on **Tools – Script - Load** and open the script **script_waveguide.nfs**.

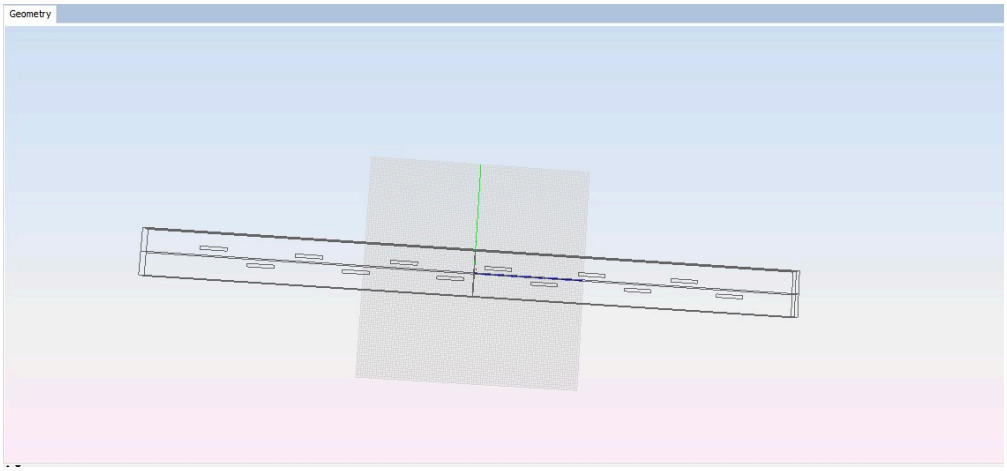


Figure 1615: Geometry

Step 6: Set Simulation Parameters.

Select **Simulation --> Parameters** option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

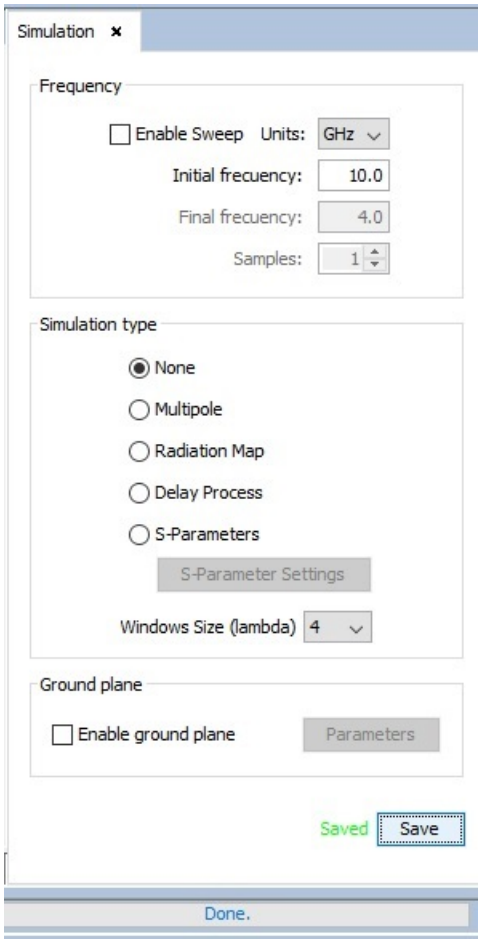


Figure 1616: Simulation Parameters panel

Note: The design frequency and the simulation frequency must be the same.

Step 7: Add a port to the waveguide.

Click on the **waveguide menu** and Select **Source --> Waveguides--> Add Waveguide Port** option on the menu bar and add the port.

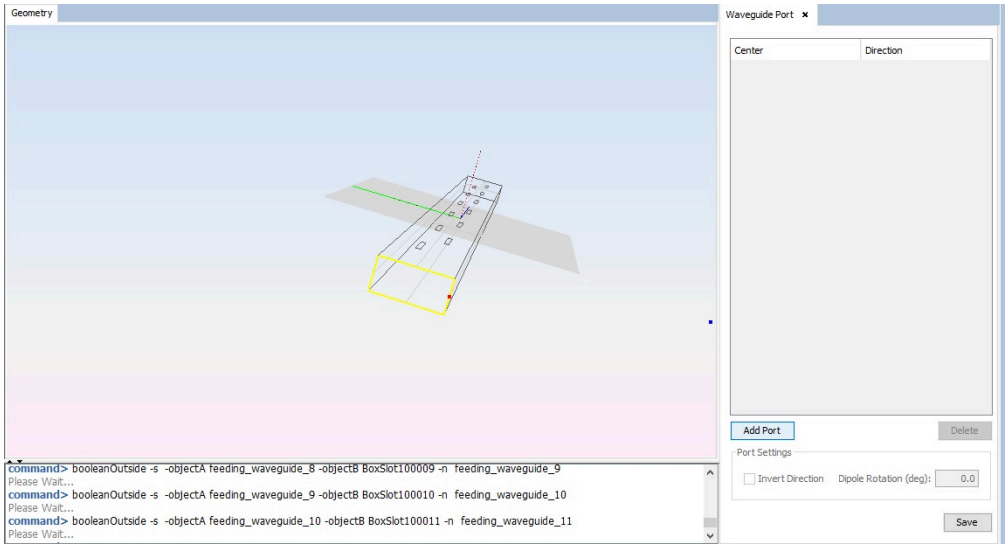


Figure 1617: Add Waveguide Port Panel

Step 8: Set the solver parameters.

Click on **Solver --> Parameters** option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on Save button before going to next step.

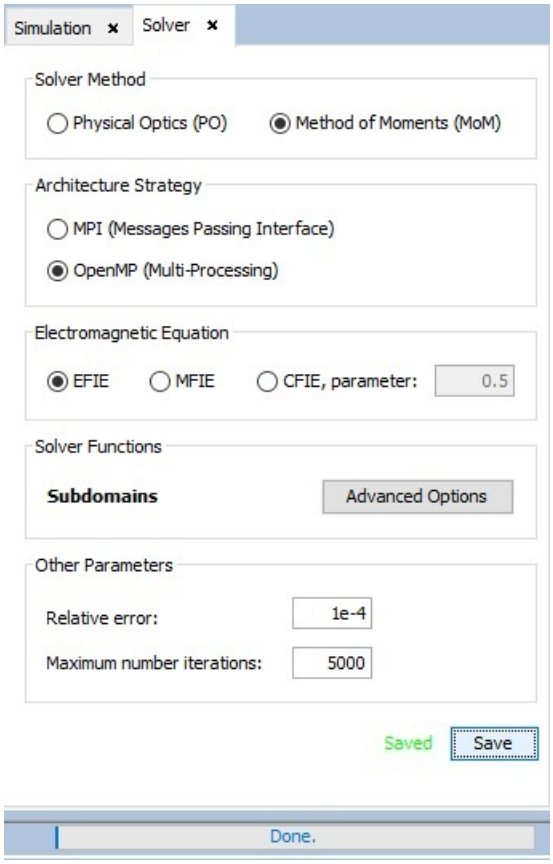


Figure 1618: Solver Parameters panel

Select **Advanced Options** and activate Preconditioner as shown.

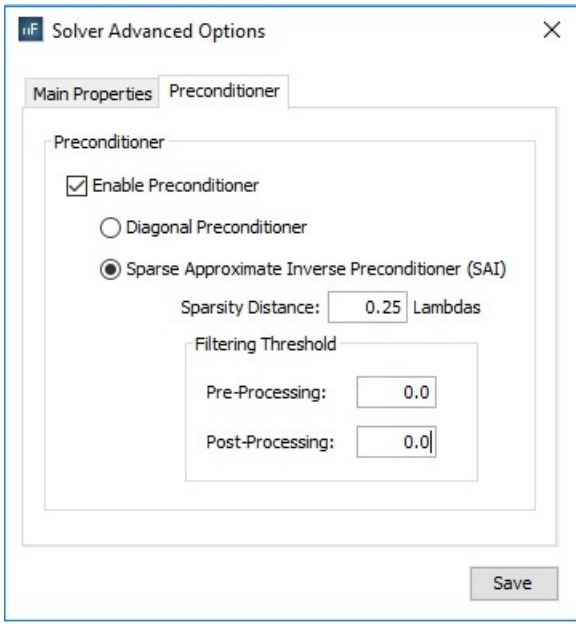


Figure 1619: Solver Advanced Options panel

Step 9: Meshing the geometry model.

Select **Meshing --> Parameters**' to open the meshing configuration panel and then set the parameters as show the next figure.

Because this design requires a high precision, 20 division are set.

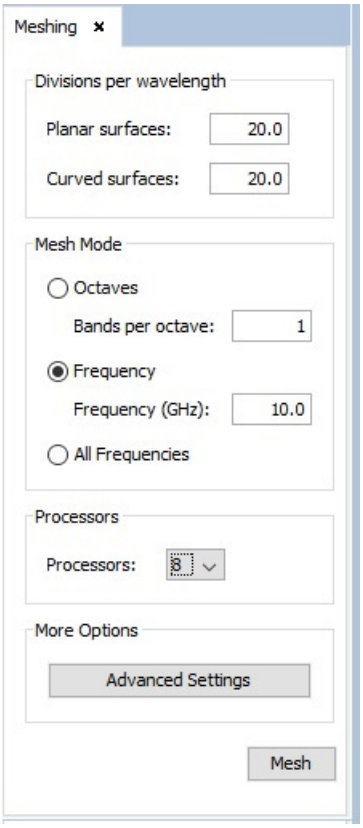


Figure 1620: Meshing panel

Step 10: Execute the simulation.

Select **Calculate --> Execute** option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

Then click on **Execute** button to starting the simulation.

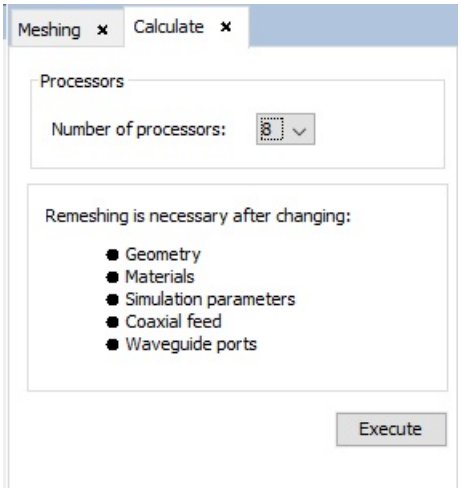


Figure 1621: Execute panel

Step 11: Show Results.

To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select **Show Results --> Radiation Pattern --> View 3D Pattern**' option to show the cuts of the radiation pattern options.

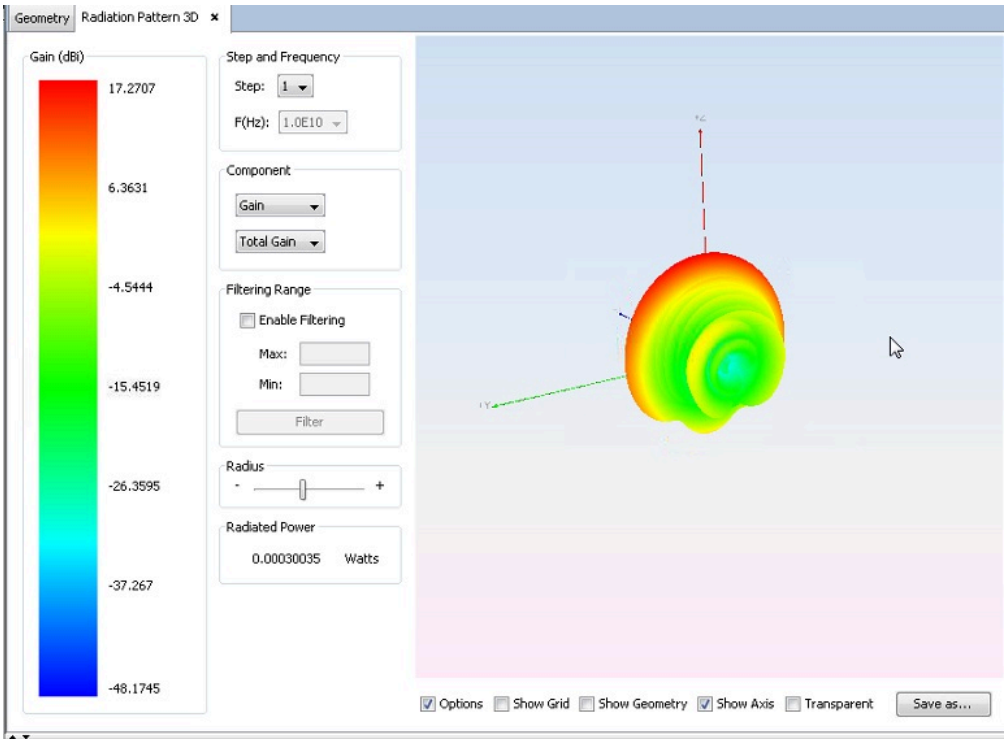


Figure 1622: Radiation Pattern 3D

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

Selecting other values for the component, step, frequency or cut parameters and clicking on **Add Series** button a new cut will be added to the selected parameters.

On **Show Results --> Far Field** menu, other results are present such as **View Cuts by Step** and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

On **Show Results --> Radiation Pattern** menu, other results are present such as **View Cuts by Step** and **View Cuts by Frequency** and this option display the values for one selected point for each step or frequency.

Select **Show Results --> View Currents** option to show the current density.

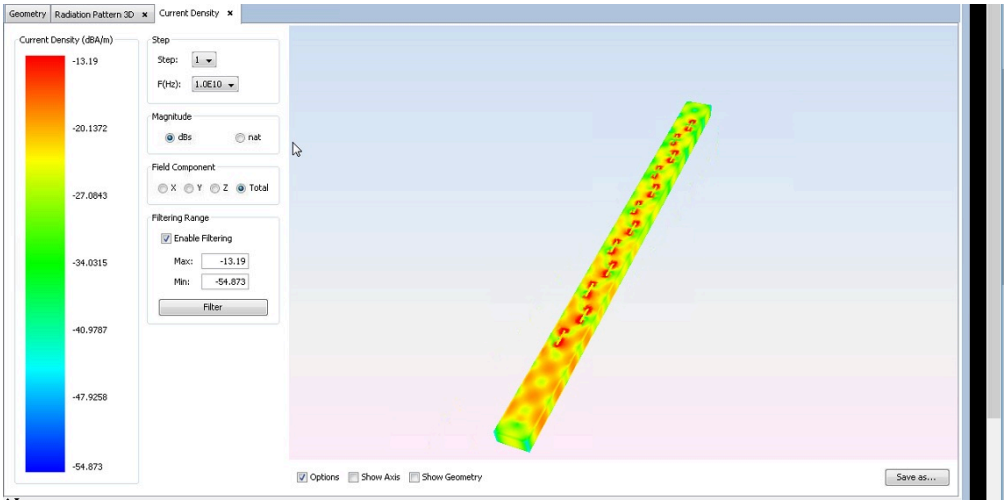
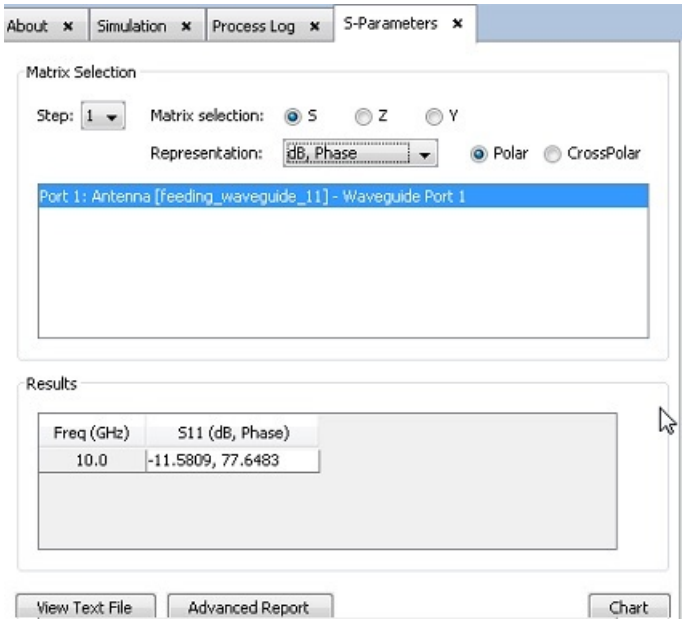


Figure 1623: Current Density

Changing values for step, frequency, magnitude, component or filtering parameters the visualization for the new parameters will be shown.

If you have selected **S-parameters** on the Simulation Panel now you will have enabled the **Show Results --> S-Parameters** option.



Example 2: Design of an Array of Slotted Waveguides

This case explains how to create an array with 12 upper waveguides and calculate its far-field, radiation pattern, current density, charge density and near field.

Step 1 Create a New **MOM Project**

Open ' **newFASANT**' and select ' **File --> New**' option.

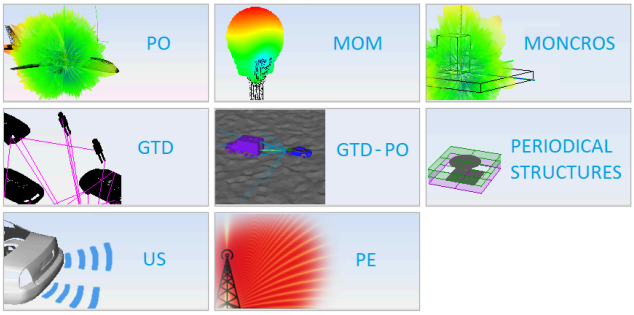


Figure 1624: New Project panel

Select ' **MOM**' option on the previous figure and start to configure the project.

Step 2 Change the scale to cm.

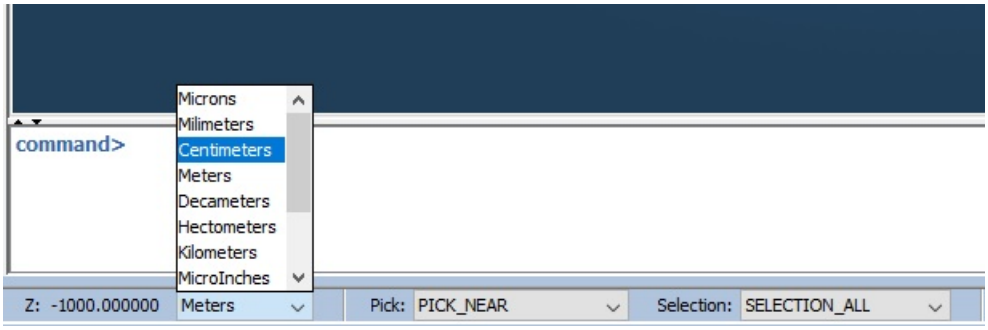


Figure 1625: Scale settings

Step 3 Open the function

Select ' **Tools --> User Functions**' option on the menu bar and open the function "SlottedWaveguideArray **.java**"

```
import java.io.BufferedWriter;
import java.io.File;
import java.io.FileWriter;
import java.io.IOException;
import java.util.logging.Level;
import java.util.logging.Logger;


    void waveguideFunction(int N,int M,double F) {

        // Parameters that can be modified

        double b = 1.016;     // waveguide heigth
        double f = F*1.0E9;    // working frequency
        double c = 3.0E10;    // light speed(cm)


        // Static parameters
        double lambda = c / f;
        double a = lambda/ Math.sqrt(2);   // waveguide width
        double fc = c / (2 * a);
        double lambdag = lambda / Math.sqrt(1 - Math.pow(fc / f, 2.0));
        double Length = (N - 1) * (lambdag / 2) + (6 * lambdag / 4);
        double Length_UP=(M - 1) * (lambdag / 2) + (6 * lambdag / 4);
        double L = lambdag / 2.0;
```

Figure 1626: User Function Code

Look at these lines, where you can change the parameters of the waveguide structure

```
    void waveguideFunction(int N,int M,double F) {

        // Parameters that can be modified

        double b = 1.016;     // waveguide heigth
        double f = F*1.0E9;    // working frequency
        double c = 3.0E10;    // light speed(cm)


        // Static parameters
        double lambda = c / f;
        double a = lambda/ Math.sqrt(2);   // waveguide width
        double fc = c / (2 * a);
        double lambdag = lambda / Math.sqrt(1 - Math.pow(fc / f, 2.0));
        double Length = (N - 1) * (lambdag / 2) + (6 * lambdag / 4);
        double Length_UP=(M - 1) * (lambdag / 2) + (6 * lambdag / 4);
        double L = lambdag / 2.0;


        double SlotLength = 0.81 * (lambda / 2);
        double SlotThin = 0.05 * lambda;
        double dStart = 3 * (lambdag / 4.0);   //distance from the end of the waveguide to the first slot center
        double SlotGap = 0.243953;
```

Figure 1627: User Function Code

The parameter "a" represents the width of the guides and is calculated to be equal to lambdag / 2 so that for any working frequency the side walls of the upper guides are in contact and there is no gap.

Step 4 Create the geometry of the waveguide array

Select ' **Tools --> Calculator**' option on the menu bar and write "SlottedWaveguideArray(N,M,F)".

In our case, as we want a 12-slot feeding waveguide, N= 12, M=11 and F is the working frequency value in GHz.

The script file, called " **script_waveguide.nfs**", will be automatically generated in the mydatafiles folder in the **newFASANT** directory.

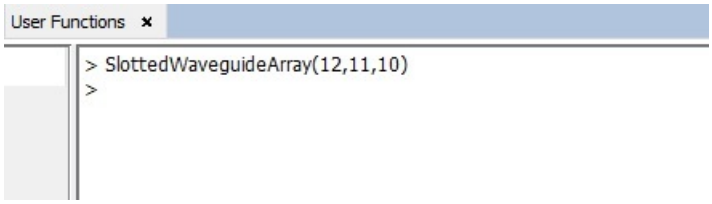


Figure 1628: Calculator panel

So our array will have a feeding waveguide with 12 slots, 12 upper guides with 11 slots each one and the working frequency is 10 GHz.

Step 5 Load the generated Script

The next step is to execute the generated script file. For that, click on **Tools – Script - Load** and open the script "**script_waveguide.nfs**".

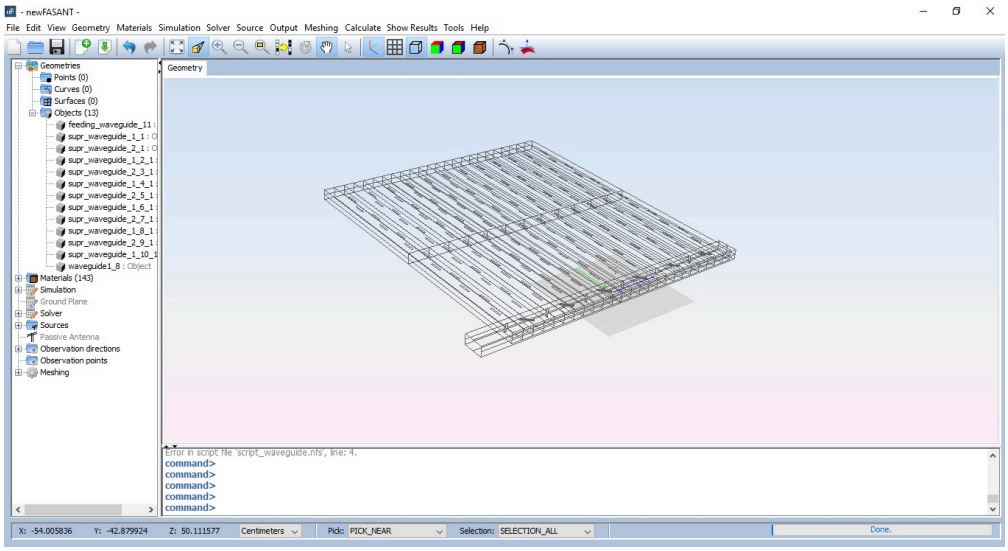


Figure 1629: Geometry

Step 6 Set Simulation Parameters

Select ' **Simulation --> Parameters**' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

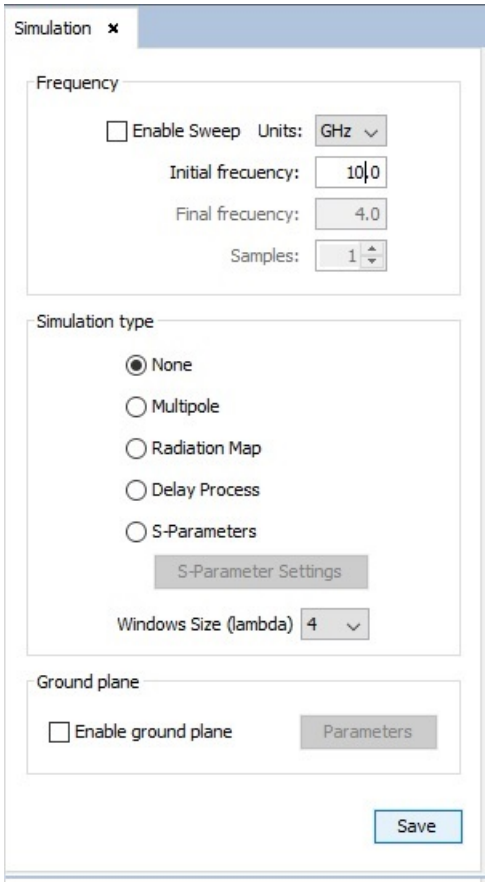


Figure 1630: Simulation Parameters panel

If the frequency to which you are going to simulate is equal to the frequency at which the parameters of the array were calculated, the radiation diagram will have the shape of a brush pointing along the z axis. On the other hand slightly varying the frequency of simulation will get the brush to oscillate around the z-axis.

Step 7 Add a port to the feeding waveguide.

Click on the waveguide and Select ' **Source --> Waveguides--> Add Waveguide Port**' option on the menu bar and add the port (see Add Waveguide Port).

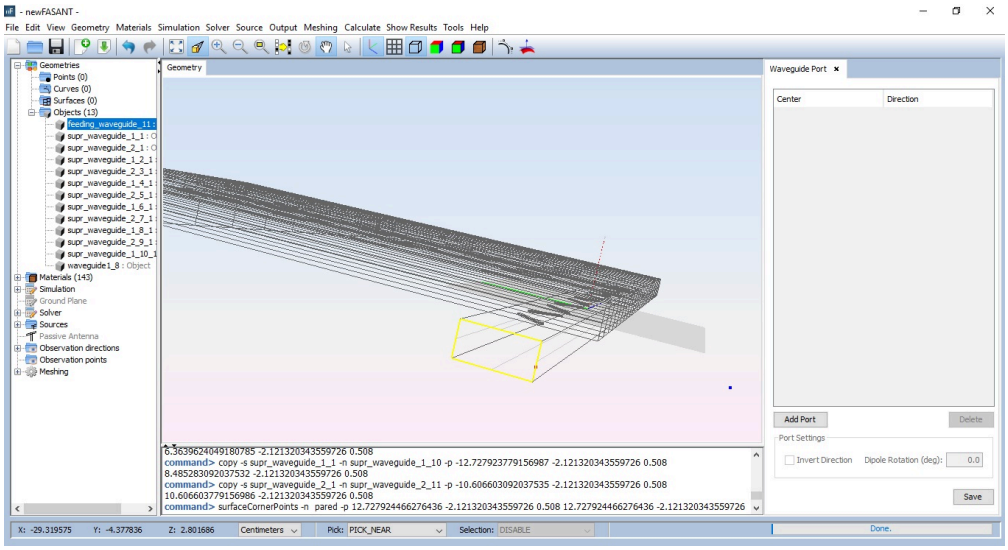


Figure 1631: Add Waveguide Port Panel

Step 8 Set the solver parameters.

Click on **Solver --> Parameters** option on the menu bar. Verify that all the parameters are defined by default, as shown in next figure. Click on Save button before going to next step

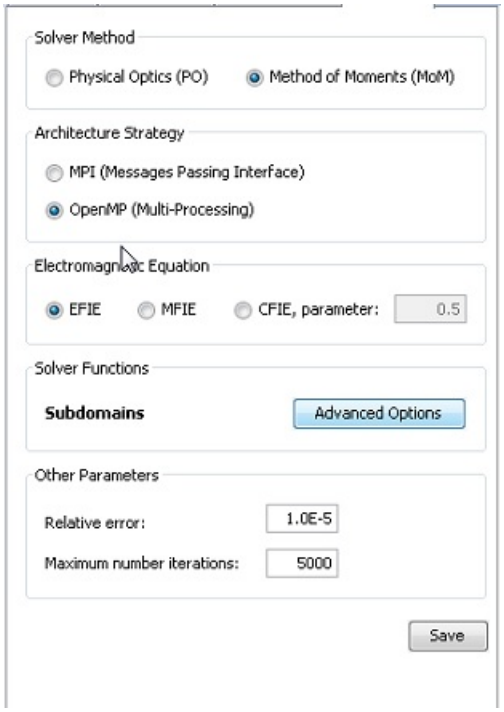


Figure 1632: Solver panel

Select ' **Advanced Options**' and activate Preconditioner as shown.

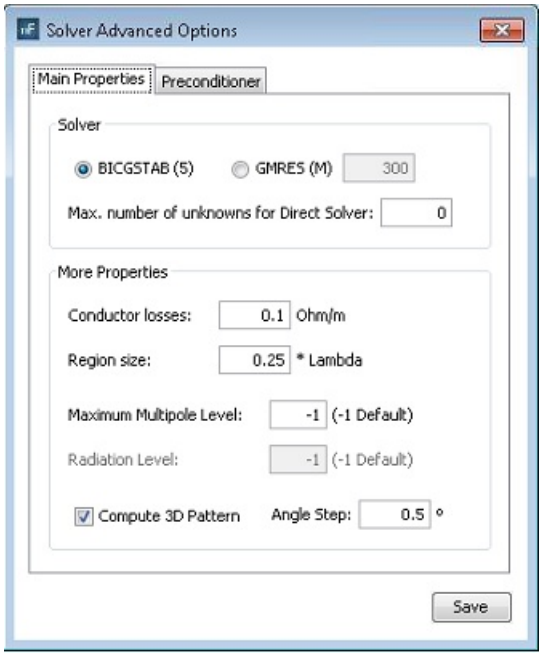


Figure 1633: Main Properties Solver Panel

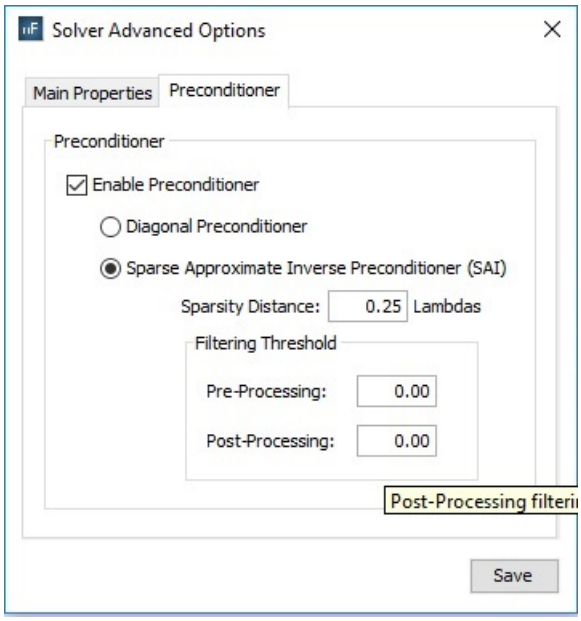


Figure 1634: Preconditioner Solver Panel

Step 9 Meshing the geometry model.

Select ' **Meshing --> Parameters**' to open the meshing configuration panel and then set the parameters as show the next figure. In order to obtain the shortest possible time for meshing, it is recommended to run the process of meshing with the number of physical processors available to the machine. As it is a very complicated design requires 20 divisions in the mesh and a high computing capacity

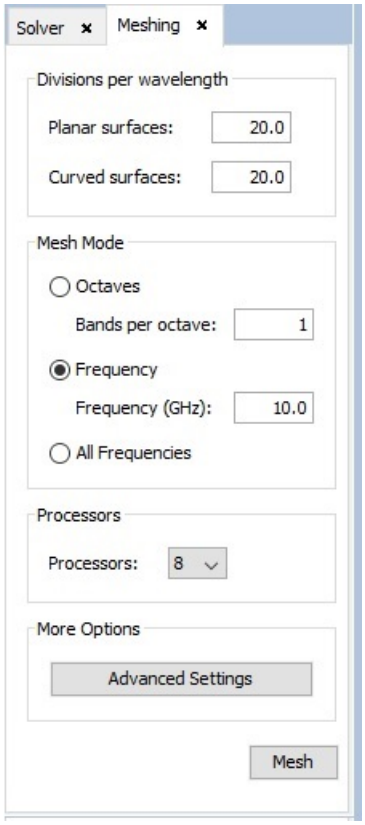


Figure 1635: Meshing Panel

Step 10 Execute the simulation.

Select ' **Calculate --> Execute**' option to open simulation parameters. Then select the number of processors as the next figure show. In order to obtain the shortest possible time for calculating the results, it is recommended to run the process with the number of physical processors available to the machine.

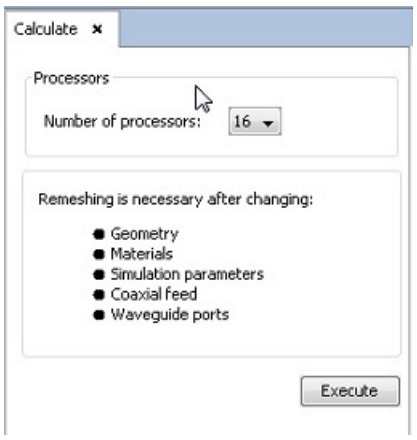


Figure 1636: Execute panel

Then click on ' **Execute**' button to starting the simulation.

Step 11 Show Results.

To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select ' **Show Results --> Radiation Pattern --> View 3D Pattern**' option to show the radiation diagram in three dimensions.

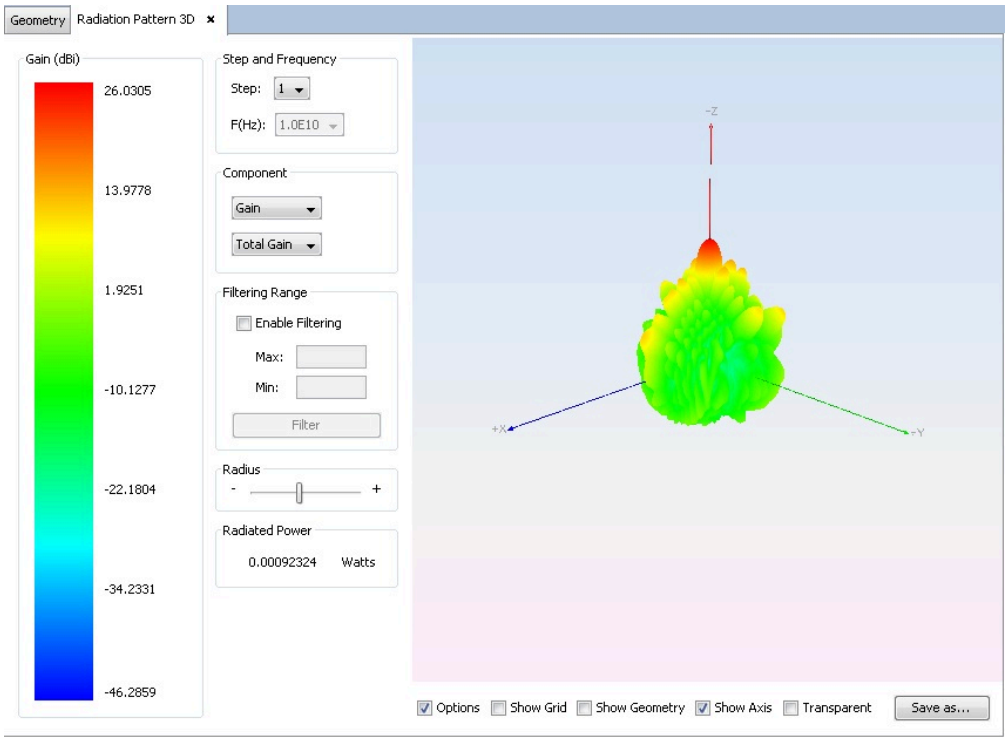
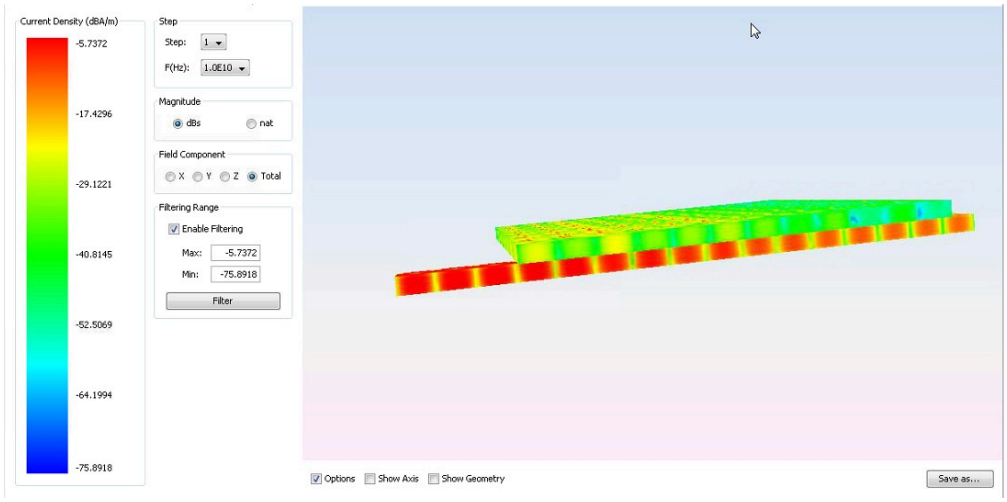


Figure 1637: 3D Radiation Pattern

Select ' **Show Results --> View Currents** ' option to show the current density on the surfaces of the array.



11.15.13 Reflectarrays

Example 1: Analysis of a Rectangular Reflectarray

This case explains how to analyze a reflectarray layout. The database used on step 4 to create the reflectarray layout is the database for 37 steps and 20 divisions per wavelength (see Annex 2: Analysis of a Reflectarray Database Creation).

Step 1: Create a new MOM Project.

Open newFASANT and select 'File --> New' option.

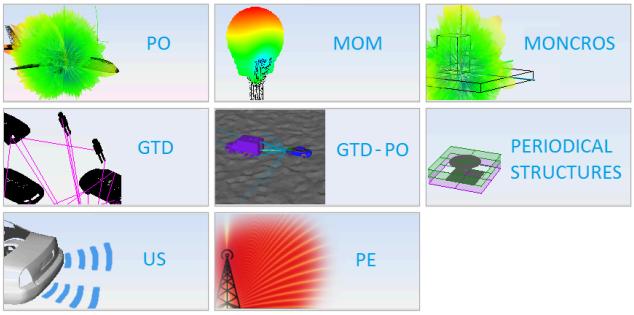


Figure 1638: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model.

First, select 'millimeters' on units list on the bar at the bottom of the main window.

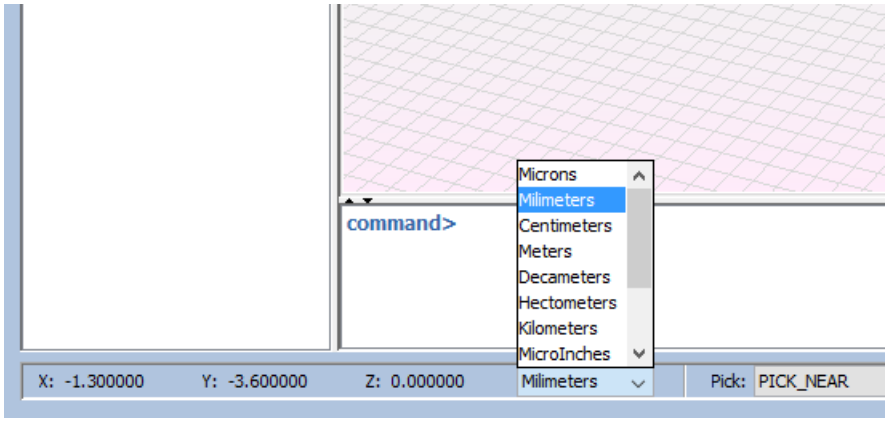


Figure 1639: Units Selection

Add these parameters to have the reference for the reflectarray layout generation. On this tab, all the parameters to be used in the construction of geometry will be set (see Define Parameters). - 'length' contains the length value for the plane of the reflectarray layout. In order to follow the reference article, this parameter must be the length of the unitary cell of the database (x = 11.43mm) multiplied by 37 unitary cells that the reflectarray contains in 'X' axis. - 'width' contains the width value for the plane of the reflectarray layout. In order to follow the reference article, this parameter must be the width of the

unitary cell of the database (y = 10.16mm) multiplied by 29 unitary cells that the reflectarray contains in ‘Y’ axis. - ‘focus’ contains the value for the position of the focus on the ‘Z’ axis. This value will be used for positioning a relative plane to locate the radiation pattern and its value is 340mm.

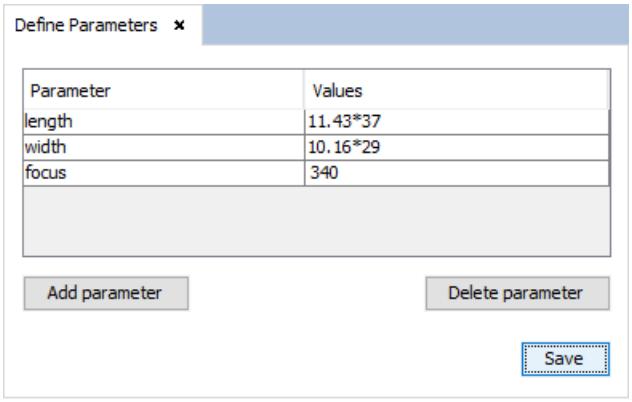


Figure 1640: Define Parameters panel

Generate a plane with a width of ‘length’ and a depth of ‘width’, with the first corner on -‘length’/2, -‘width’/2, 0; executing the 'plane' command writing it on the command line and sets the parameters when command line asks for it.

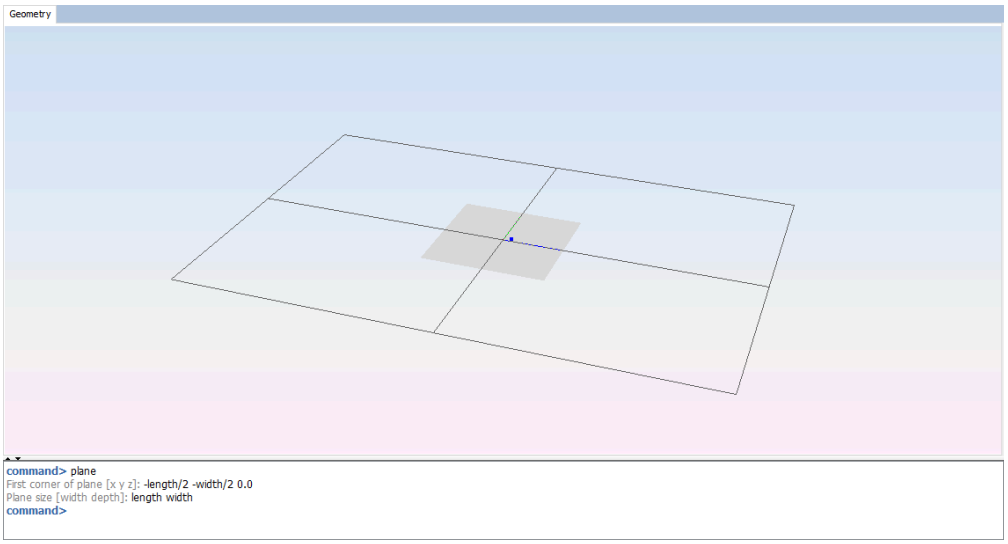


Figure 1641: Plane parameters

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

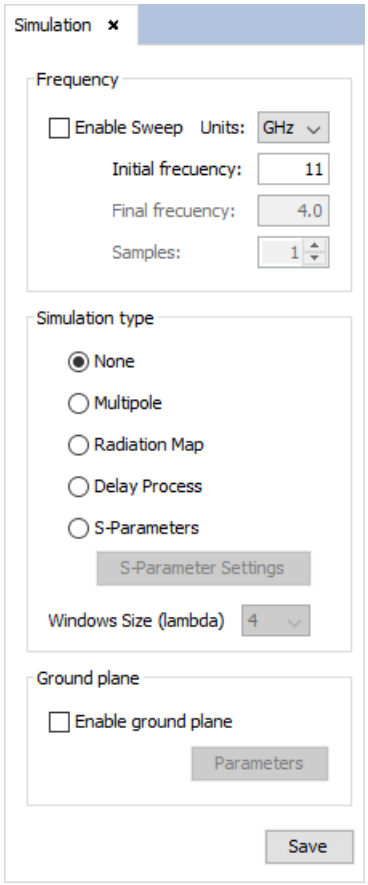


Figure 1642: Simulation Parameters panel

Step 4: Create a reflectarray layout.

Select 'Source --> Reflectarray --> New Layout' option. The following panel will appear. First load de database previously created with Periodical Structures module. Then select the parameters as the next figure shows

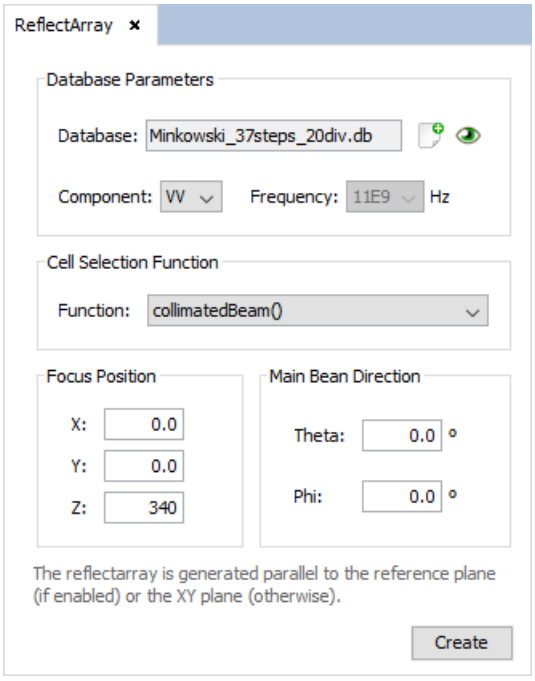


Figure 1643: Reflectarray New Layout panel

Clicking on the 'eye' icon, the parameters of the database will be shown on a new tab.

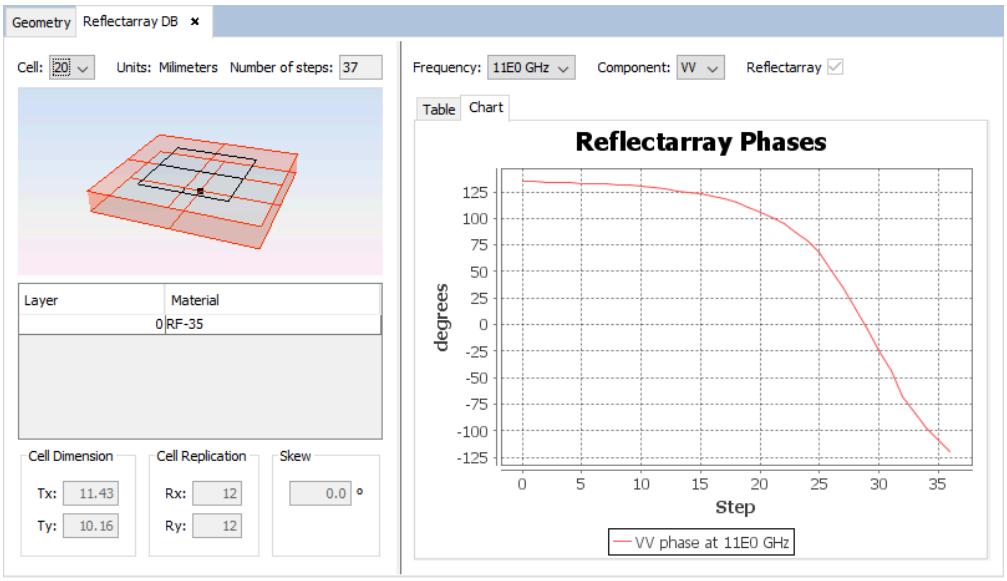


Figure 1644: Reflectarray Database Information panel

Select the plane previously created to generate the reflectarray on it. Then, click on 'Create' button and wait until the reflectarray layout is created.

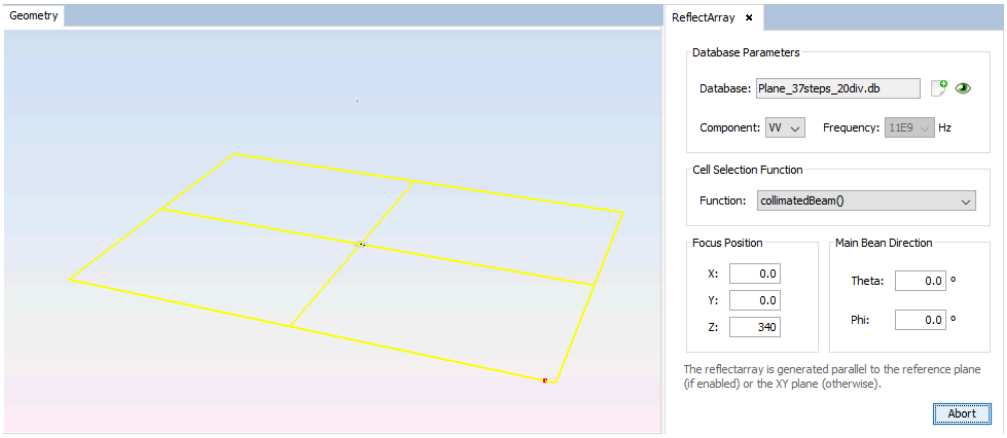


Figure 1645: Reflectarray creation process

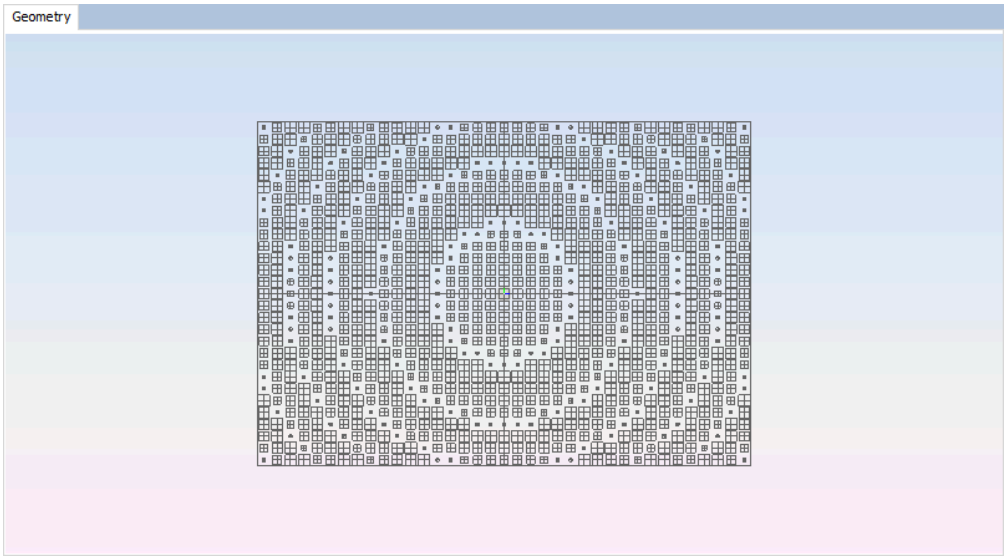


Figure 1646: Reflectarray Layout

Step 5: Set the antenna parameters.

First, select 'View--> Reference Plane' and set the parameters as the next figure shows, then click on 'Update' button to put the reference plane as the figure:

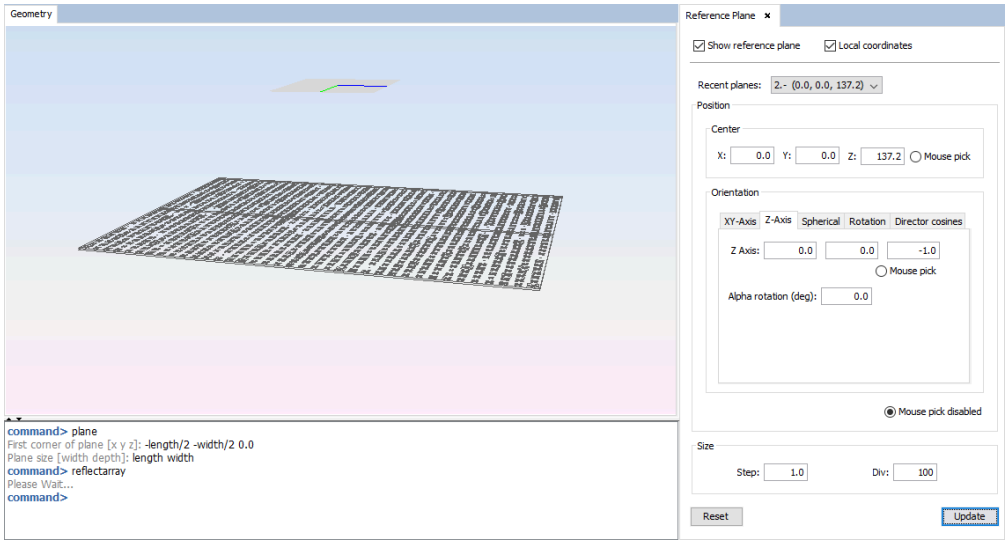


Figure 1647: Reference Plane panel

Select 'Source --> Import Pattern File' option and select 'BOC0677.dia' file from 'myDataFiles' folder on the installation path of newFASANT software.

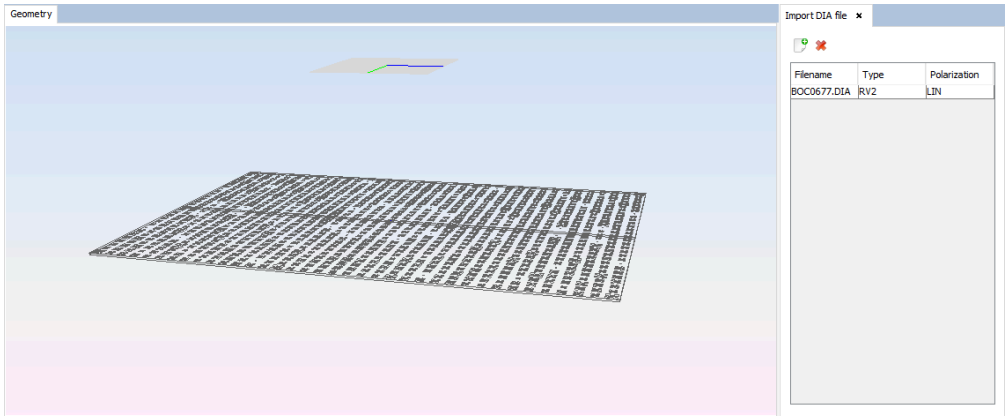


Figure 1648: Pattern File Import panel

Select 'Source --> Pattern File --> Pattern File Antenna' option and set the parameters as show the next figure. Then save the parameters and the antenna appears.

Important select the check box "Relative to reference plane" to set the position and orientation relative to the local reference plane.

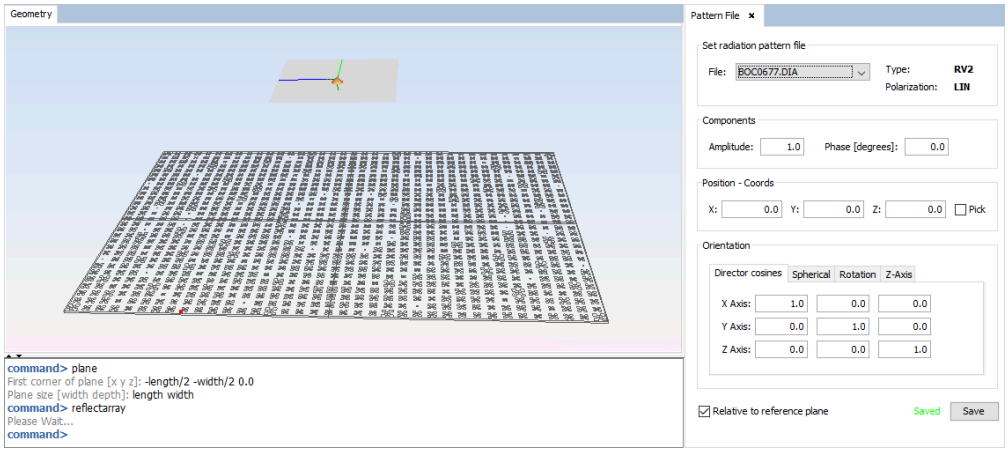
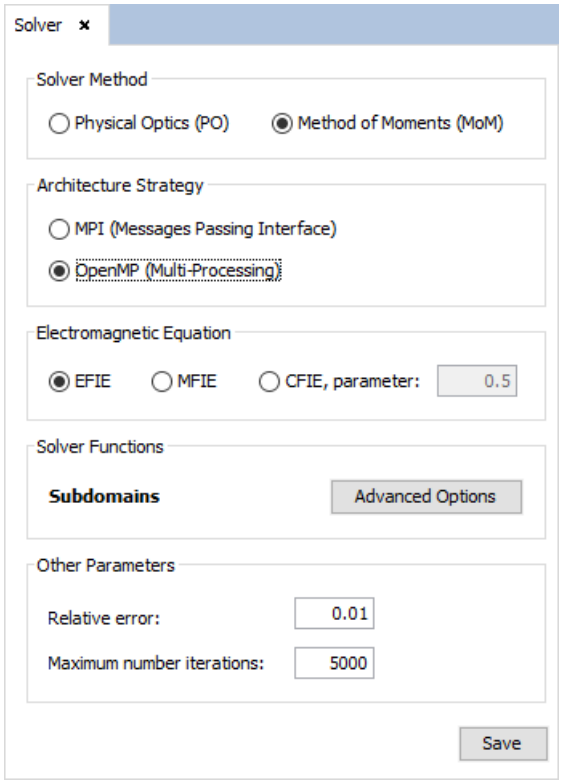


Figure 1649: Pattern File Antenna panel

Step 6: Set solver parameters.

Select Solver --> Parameters' to open the solver configuration panel and then set the following parameters in order to obtain the most possible efficiency and the 3D results: - OpenMP option for the Architecture Strategy, on Solver parameters panel.



Then click on 'Advanced Options' button to configure the advanced properties as the next figure shows:

- Check the “Compute 3D Pattern” box and set the angle step to 0.5, on ‘Main Properties’ tab.
- Check the “Enable Preconditioner” box and select Diagonal Preconditioner option on ‘Preconditioner’ tab.

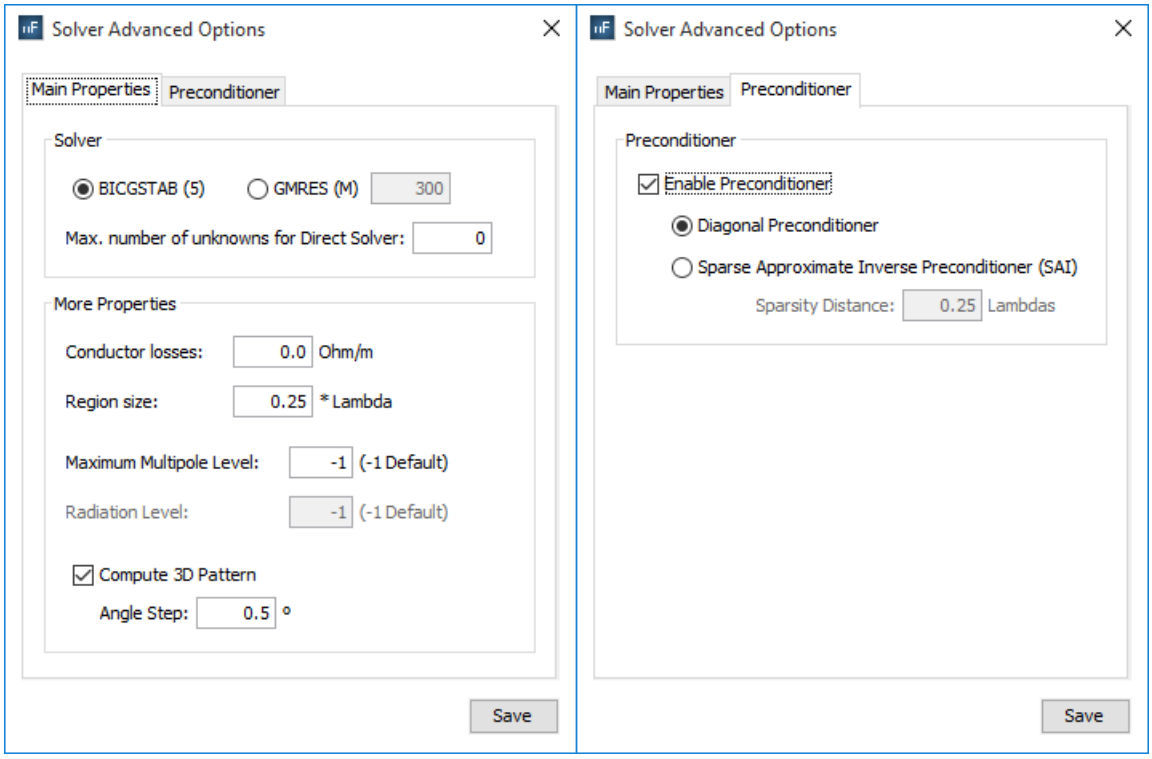


Figure 1650: Solver Advanced Options panel

Step 7: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure.In this case, the number of division is selected by the case with the most accuracy of the databases compared to section Periodical Structures. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed.

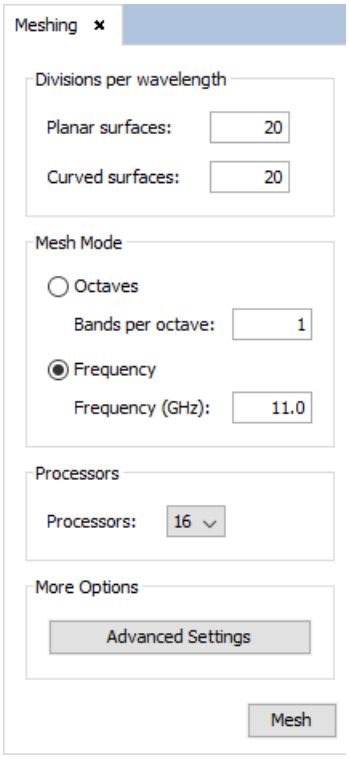


Figure 1651: Meshing Parameters panel

Step 8: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed.

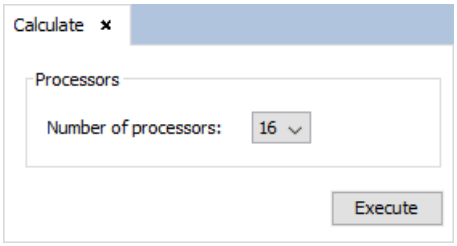


Figure 1652: Execute Parameters panel

Step 9: Show Results. To get more information about the graphics panel advanced options (clicking on right button of the mouse over the panel) see Annex 1: Graphics Advanced Options.

Select 'Show Results --> Far Field --> View Cuts' option to show the cuts of the observation directions options. Delete all series with the red cross button and select 'Gain Lineal' and 'ETotal Gain (dBi)' on 'Component' panel, then click on 'Add Series' button.

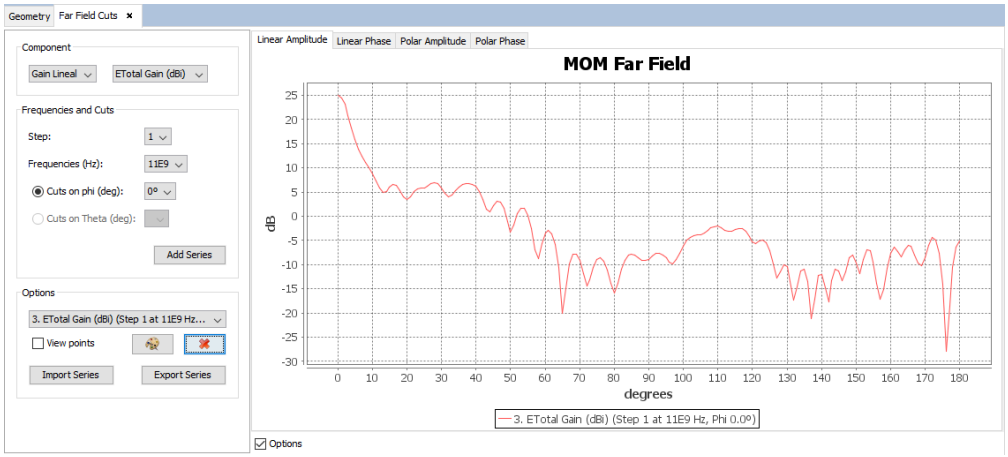


Figure 1653: Far Field cuts

Selecting other values for the component, step, frequency or cut parameters and clicking on 'Add Series' button a new cut will be added to the selected parameters. On 'Show Results --> Far Field' menu, other results are present such as 'View Cuts by Step' and 'View Cuts by Frequency' and this option display the values for one selected point for each step or frequency.

Select 'Show Results --> Radiation Pattern --> View 3D Pattern' option to show the cuts of the radiation pattern options.

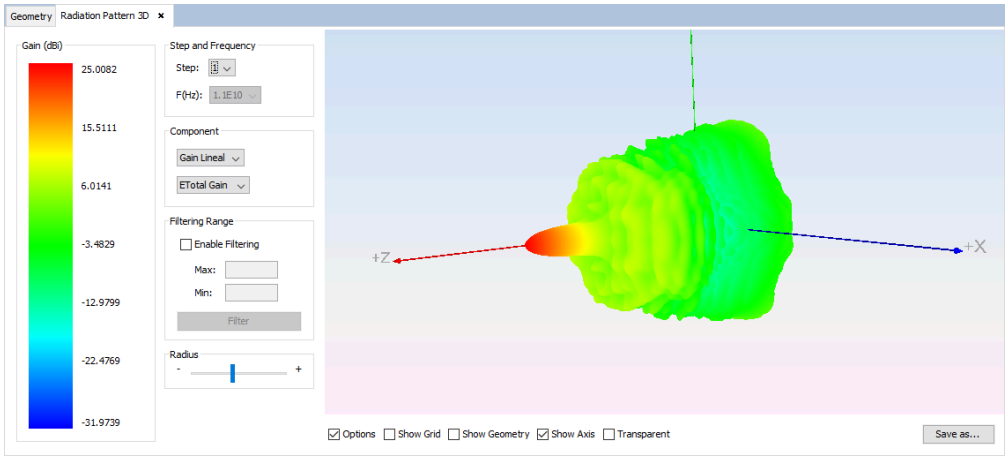


Figure 1654: Radiation Pattern 3D

Changing values for step, frequency, component or filtering parameters the visualization for the new parameters will be shown.

11.15.14 Radomes

Example 1: Parametric Radomes

Benchmark Description

This document shows several examples for studying the thickness of a radome by using the parametric simulations. In particular, we are interested in studying the behavior of a radome with three layers, where the external ones are identical but the middle one has a different material and its thickness depends on the parameter t. This idea is shown in the bi-dimensional example of the figure below.

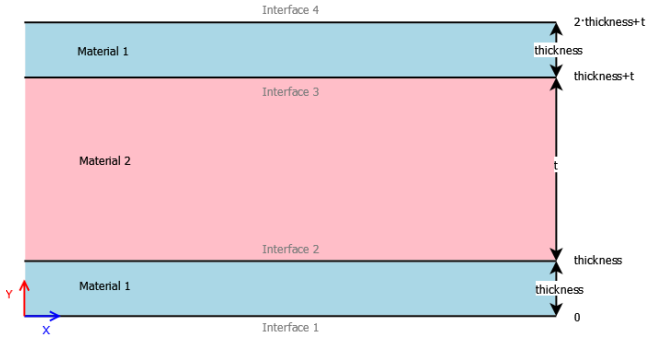


Figure 1655: Parametric radome design.

It has been analyzed by using several radome shapes and also different excitation methods.

Setting-Up Common Parameters

The table below resumes the common parameters taken into account in the following examples:

Table 8:

Data	Definition	Values
Frequency	Simulation frequency	5 GHz
Material 1	Dielectric constant of Material 1	4 - 0.08·j
Material 2	Dielectric constant of Material 2	1.1 - 0.0055·j
Thickness	Constant thickness of material 1	0.25 mm
t	Parametric thickness of material 2	3 , 3.5 , 4, 4.5, 5 mm
Units	Units for working in newFASANT	Millimeters

Now, we set-up the parameters of this table within a project that will be copied and renamed for all the configurations shown.

1. **Create a new MOM project**. To do that, click on **New Project** button and select the **MOM** module.

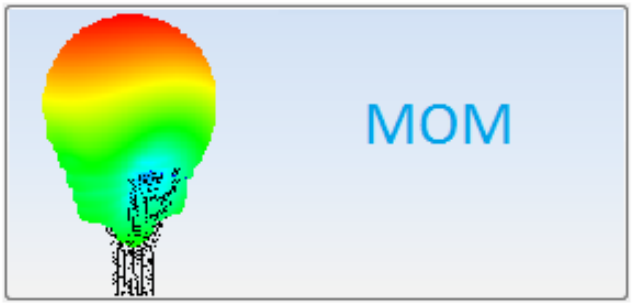


Figure 1656: Choose the MOM module

2. **Set the units**. Click on the current units and select **Millimeters** to make easier working with the thickness.

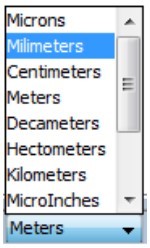


Figure 1657: Set units to Milimeters unit

3. **Set the frequency.** Click on **Simulation – Parameters** menu and then the Simulation panel is open on right side of screen. Set the frequency to 5 GHz and click on **Save** button before closing the panel.

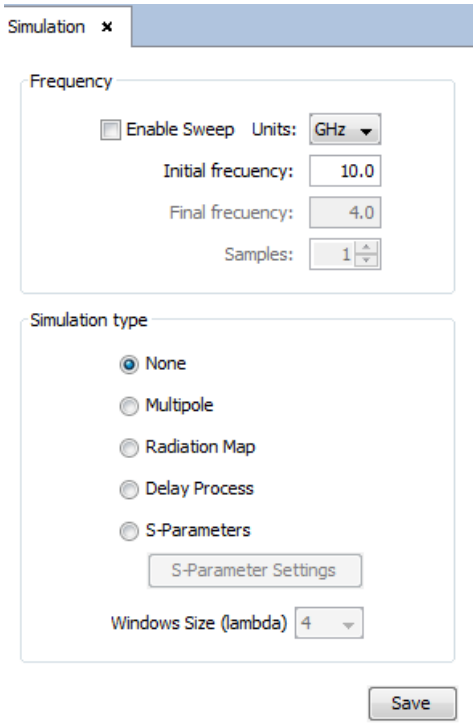


Figure 1658: Set the frequency to 5 GHz

4. **Define the materials.** Click on **Materials – Add** menu and then the **Add** Material panel is open on right side of screen. Set the **Name** and Color of the materials to be defined, keep the default Material Definition option, that is **Material defined by geometry**, and then click on Set Parameters button to specify the dielectric values of each material (click on **OK** button within the Geometry Material window to confirm the dielectric constant assigned). Remember clicking on **Save** button after defining every material.

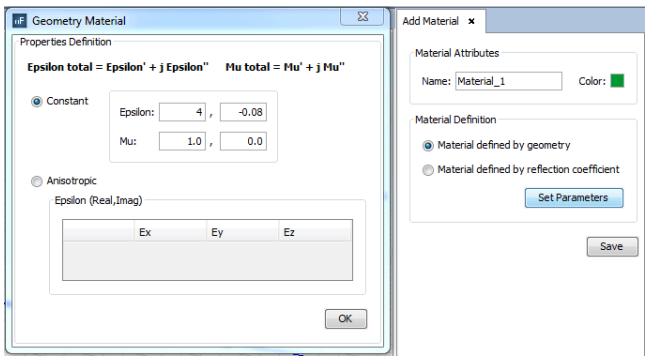


Figure 1659: Material 1 definition

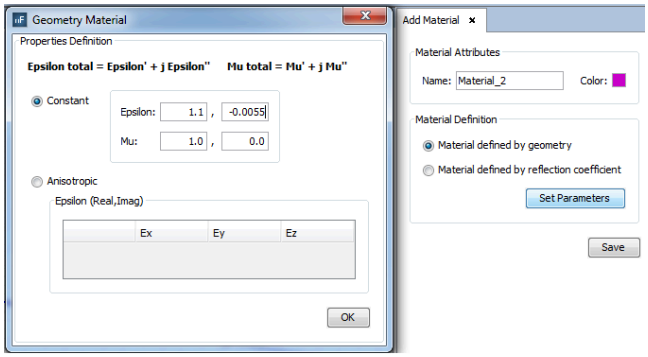


Figure 1660: Material 2 definition

5. **Define the parameters.** Click on **Geometry – Parameters – Define Parameters**, and then the Define Parameters panel is open on right side of screen. Unless it is not necessary to add the thickness of the Material 1 as a parameter because it has a fixed value, we will define its parameter to create the geometry also as a full parametric model. To assign a fixed value to a parameter, the value must be inserted between "{ }" characters. The interesting parameter is the thickness of the Material 2, which we have named t, and it must be ranging between 3 and 5 millimeters. To assign a range of values to a parameter, the initial and end values are inserted between "[ ]" characters and separated by spaces, and then the number of samples must be specified. Remember click on ***Save*** button before closing this panel.

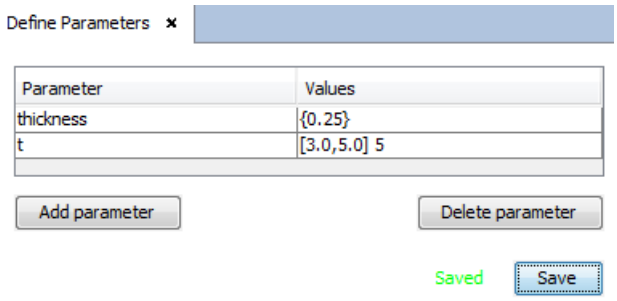


Figure 1661: Parameters definition

6. **Save the project.** Click on **File – Save As** menu and specify where you want to save this project, which will be used as base project for the next example. We have named it as `baseProject.nfp`. Then, for every example we will copy and rename this base project and only will have to set-up the antenna and the radome.
7. **Solver parameters.** Click on **Solver – Parameters** menu to open the Solver panel on right side. The only parameter to be modified in this document may be the **Preconditioner** one that may be enabled or disabled for speeding-up the solver convergence. Click on **Advanced Options** button to open the Solver Advanced Options window, then select the **Preconditioner** tab and enable or disable the option **Enable Precontioner** as desired.

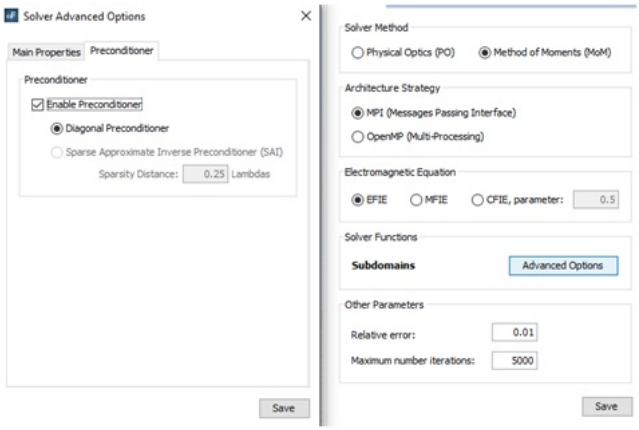


Figure 1662: Preconditioner Solver options

Feeding with a Circular Horn

Create a copy of `baseProject.nfp` project and rename it as `baseProjectHorn.nfp`. Then open it with newFASANT by clicking on File – Open menu or clicking twice on the project icon.

Insert a conical horn by using the existing primitives. To do that, click on **Source – Primitive Antenna – Horns – Conical** Horn menu and then the Conical Horn panel is open on right side of screen.

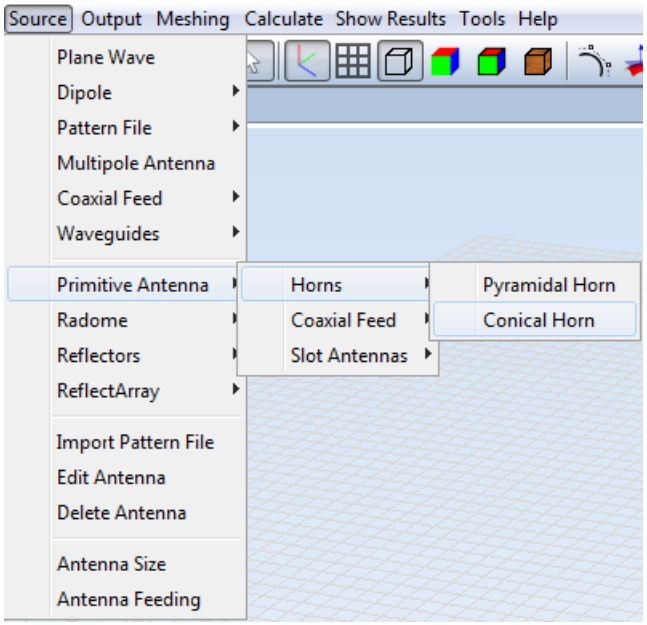


Figure 1663: 9 Conical Horn menu.

The radomes will be placed on the origin of coordinates to make easier its geometrical definition, so the horn is created 10 centimeters below it on the Z axis. Keep the most of parameters in the Conical Horn panel by default except the **Z-Position** that must be set to **-100**. Click on **Add** button to insert the horn before closing the panel.

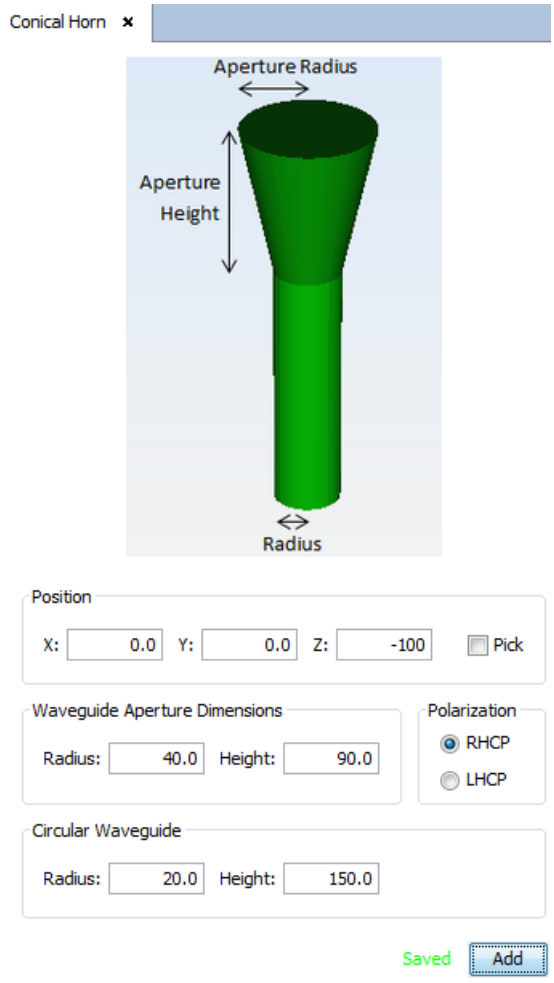


Figure 1664: 10 Conical Horn definition.

Save the current project, as it will be used again as base project in the following subsections.

Square Radome

Create a copy of `baseProjectHorn.nfp` project and rename it as `Horn_Square.nfp`. Then open it with newFASANT by clicking on File – Open menu or clicking twice on the project icon.

The radome considered in this section is a square of 50 centimeters of side with the base centered at the origin of coordinates.

Geometric Design

Four square planes will be generated to model the radome as a thin layer approximation. It may be achieved by using the plane command, which requires the first corner (the lower one), the width (X size) and the depth (Y size). The table below shows the parameters to be used for every plane:

Plane	X (mm)	Y (mm)	Z (mm)	Width (mm)	Depth (mm)
1	-250	-250	0	500	500
2	-250	-250	thickness	500	500
3	-250	-250	thickness+t	500	500
4	-250	-250	(2*thickness)+t	500	500

```
command> plane
First corner of plane [x y z]: -250 -250 0
Plane size [width depth]: 500 500
command> plane
First corner of plane [x y z]: -250 -250 thickness
Plane size [width depth]: 500 500
command> plane
First corner of plane [x y z]: -250 -250 thickness+t
Plane size [width depth]: 500 500
command> plane
First corner of plane [x y z]: -250 -250 (2*thickness)+t
Plane size [width depth]: 500 500
command>
```

Figure 1665: Command line planes definition

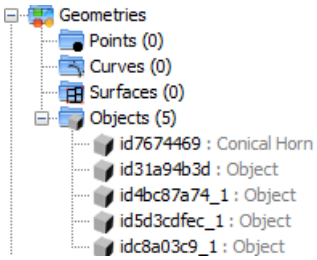


Figure 1666: List of geometric entities in the tree

After creating the four planes, in the tree we can see that they have been added in order as Objects, and each one has a different name.

Radome Definition

Having defined the geometric model, the radome must be created and assigned. To do that, click on **Source – Radome – Define Radome** menu and the Radome panel is open on right side of screen.

In the Radome panel, no radomes are created so click on Add Radome button to insert a new radome. Then, the table of interfaces of the selected radome (1) is loaded below. As we have shown in Figure 1655, the radome is made of 4 interfaces that define the 3 layers, so click on **New Interface** button twice to get the 4 interfaces.

Each plane must be assigned to every radome interface in the same order than specified in Figure 1655. Remember that the normal vectors of every interface must be pointing to the next one. To assign a plane to an interface, select the geometrical object from the tree or in the Geometry panel, then select

the corresponding row in the Interfaces table, and click on Set Objects button. Then, assign the Material to its interface, which is defined to the current layer.

The material (or layers) is considered between one interface and its next one, so the last interface does not required a material.

Remember clicking on Save button to confirm the radome definition.

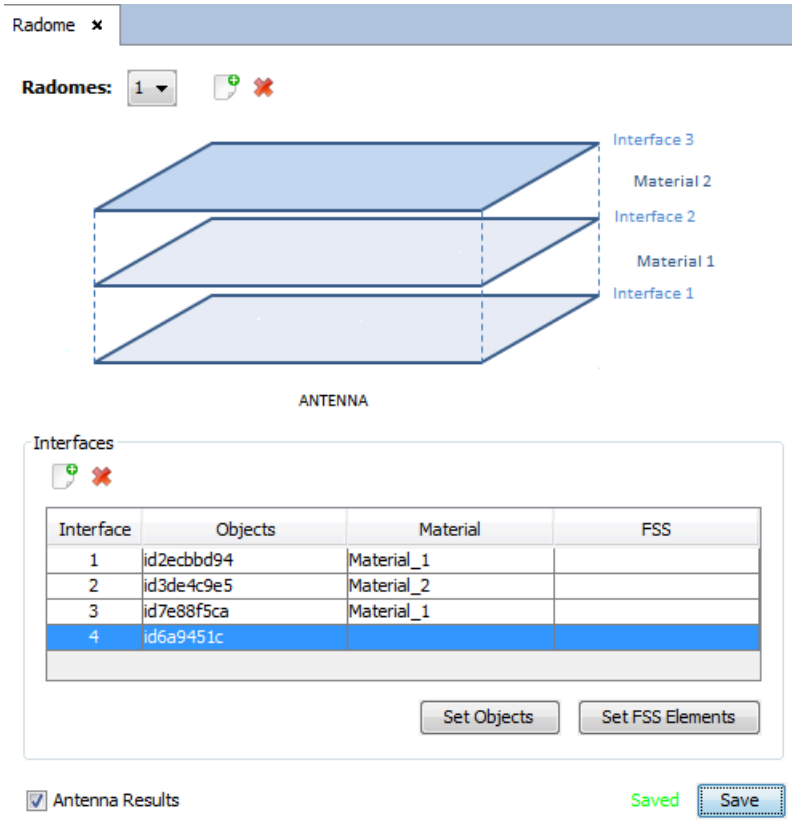


Figure 1667: 13 Radome definition.

The next figure shows the final design of the radome with the horn, with the Material Shaded view. In this representation, we verify that the layers of the radome have the correct materials.

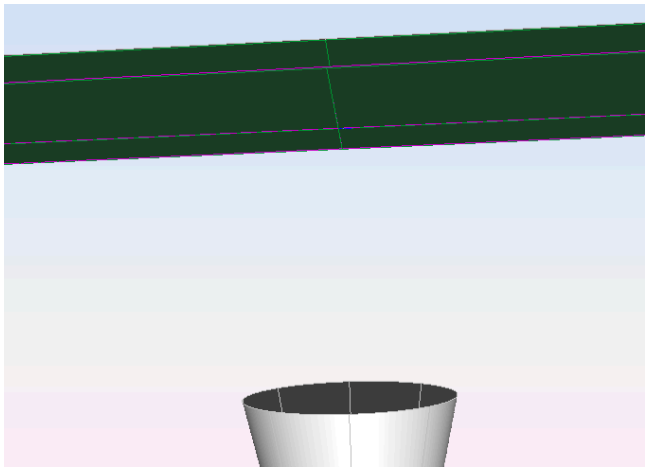


Figure 1668: 14 Shaded view with materials of the final design

Mesh and Run

The only parameters modified in the meshing process are the number of **Processors** and the **Multilevel Meshing Mode** (click on Advanced Settings to open the Meshing Advanced Parameters window), which has been **disabled**. Click on **OK** and **Mesh** buttons to confirm the changes and mesh.

The five parametric steps are meshed by requiring about 4 GB of RAM memory and 10 seconds per step.

Click on **Calculate – Execute** menu and launch the simulation with **8 processors** by clicking on Execute button.

The five parametric steps are simulated by requiring about 4 GB of RAM memory and 5 minutes per step.

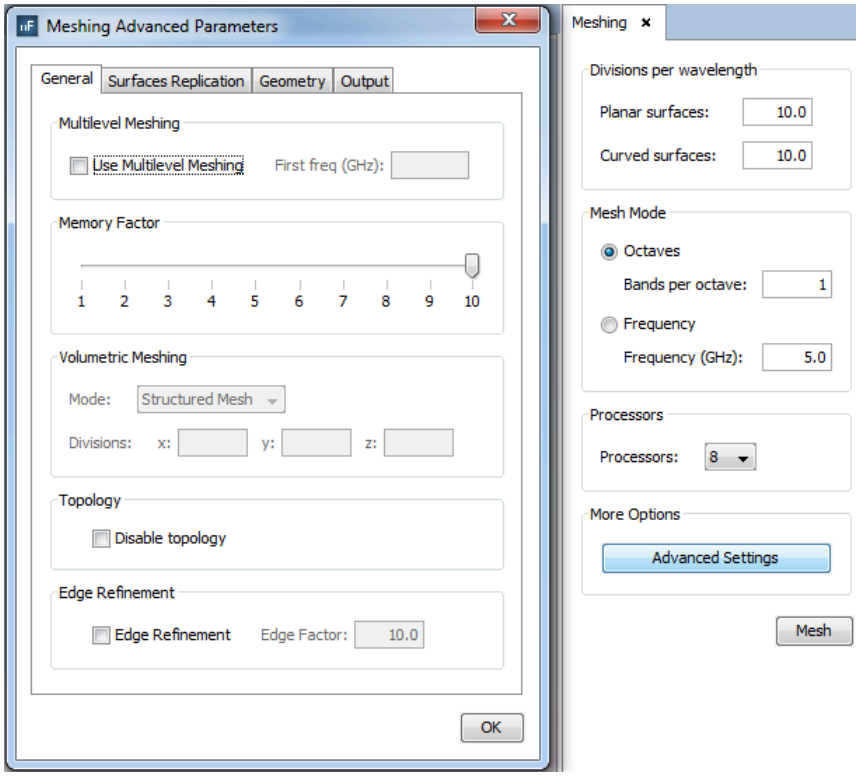


Figure 1669: 15 Meshing parameters

Computational Specifications

Step		1	2	3	4	5
Number of unknowns		43165	43165	43165	43165	43165
Required RAM (GB)		4.5	4.5	4.5	4.5	4.5
Without Preconditioner	**Number of iterations**	100	91	92	82	79
	Simulation Time (min : ss)	5 : 34	5 : 07	5 : 12	4 : 57	5 : 03
With Preconditioner	**Number of iterations**	23	24	23	22	22
	Simulation Time (min : ss)	3 : 42	3 : 39	3 : 36	3 : 40	3 : 31

Results

Radiation Pattern

- Step 1

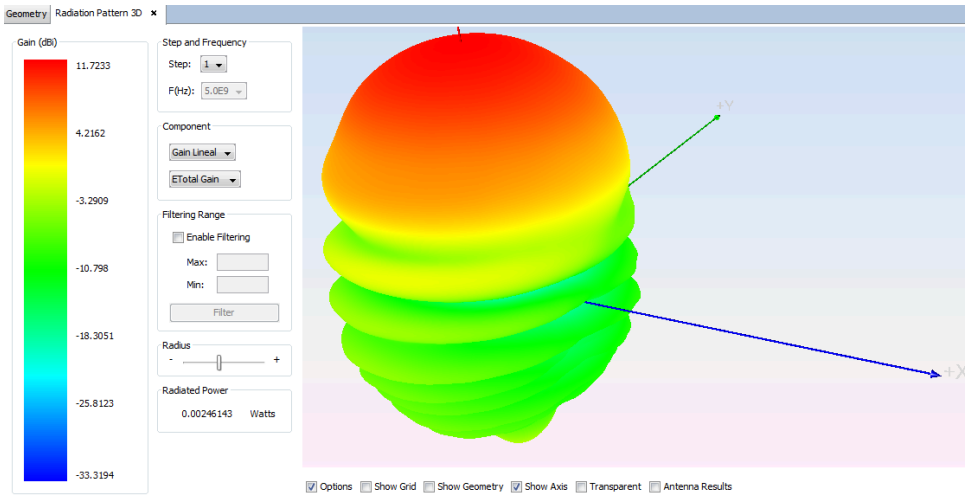


Figure 1670: 3D radiation pattern - Step 1: 5.0e9 Hz

- Step 2

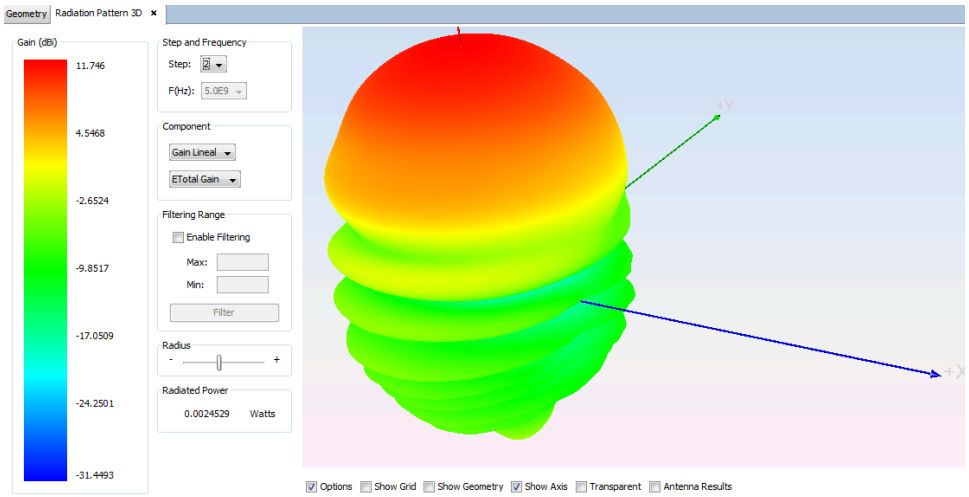


Figure 1671: 3D radiation pattern - Step 2: 5.0e9 Hz

- Step 3

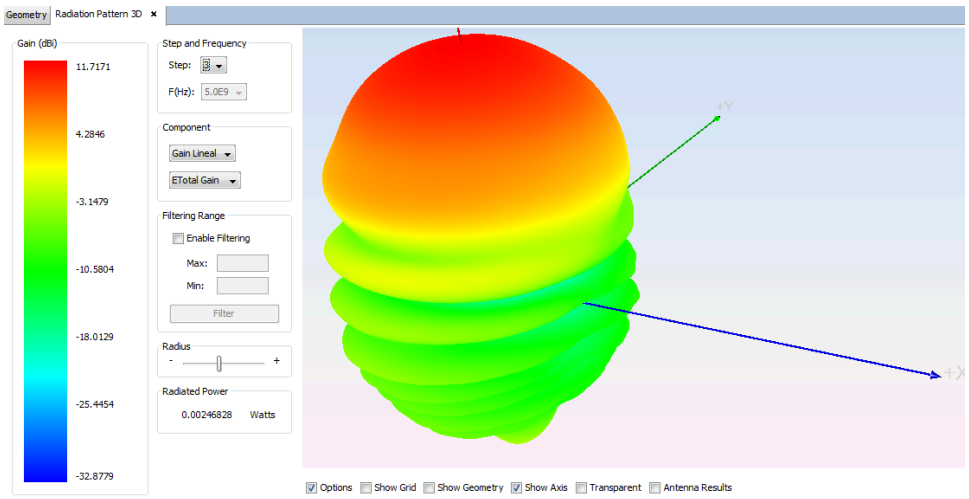


Figure 1672: 3D radiation pattern - Step 3: 5.0e9 Hz

- Step 4

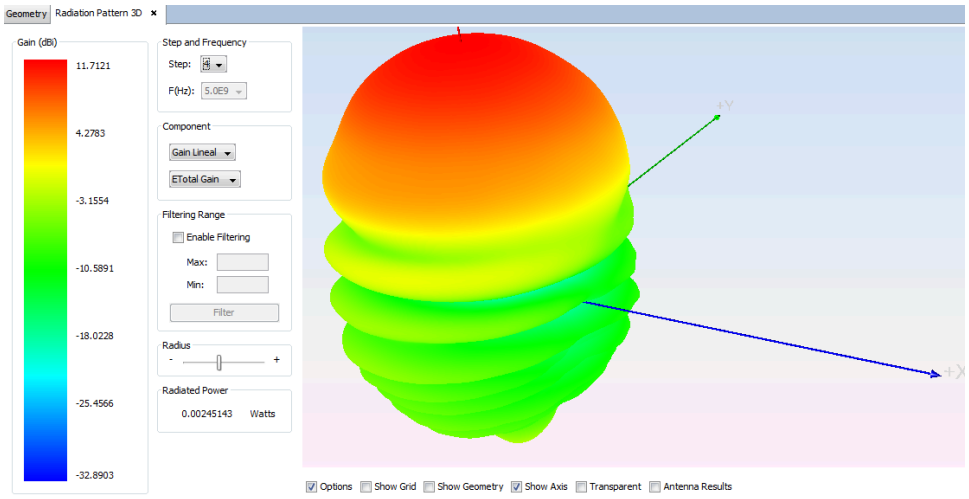


Figure 1673: 3D radiation pattern - Step 4: 5.0e9 Hz

- Step 5

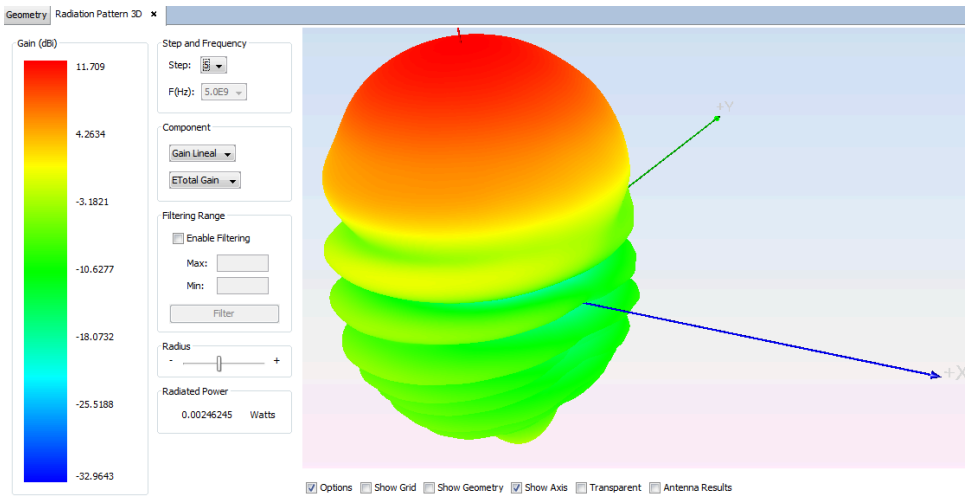


Figure 1674: 3D radiation pattern - Step 5: 5.0e9 Hz

Current Density

- Step 1

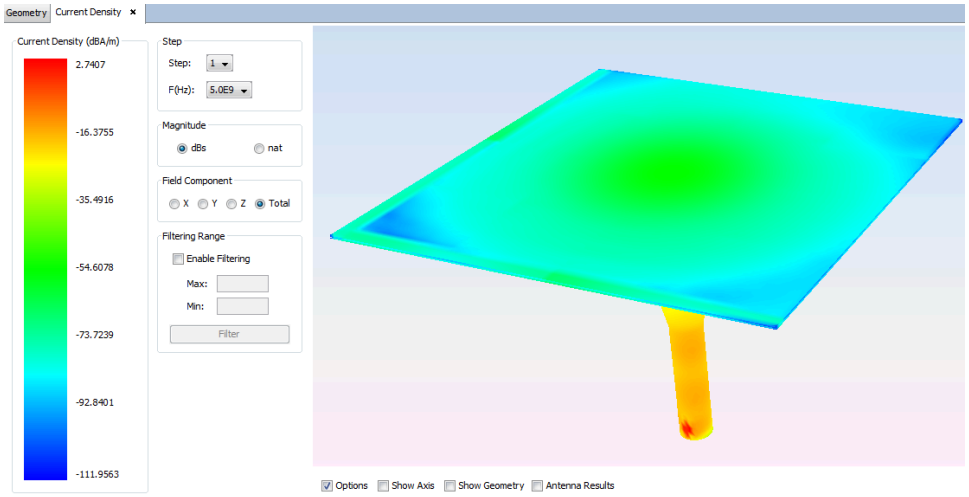


Figure 1675: Current density - Step 1: 5.0e9 Hz

- Step 2

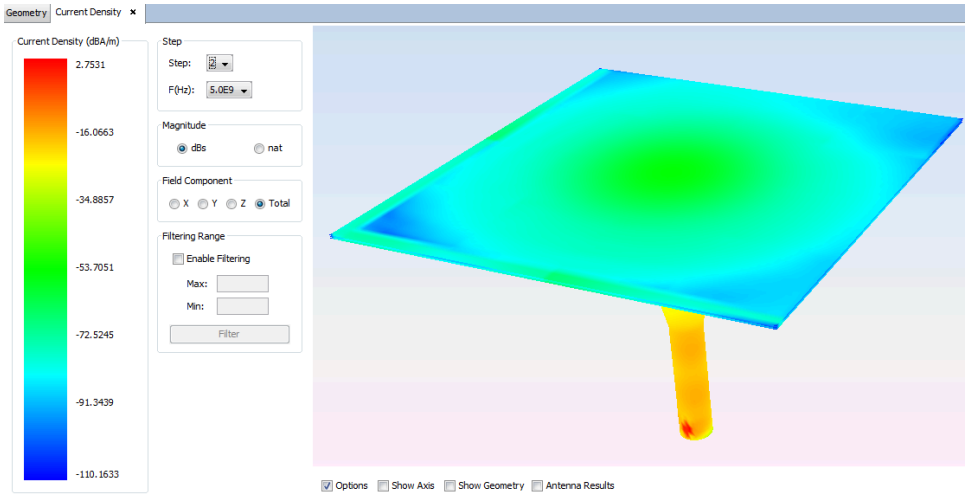


Figure 1676: Current density - Step 2: 5.0e9 Hz

- Step 3

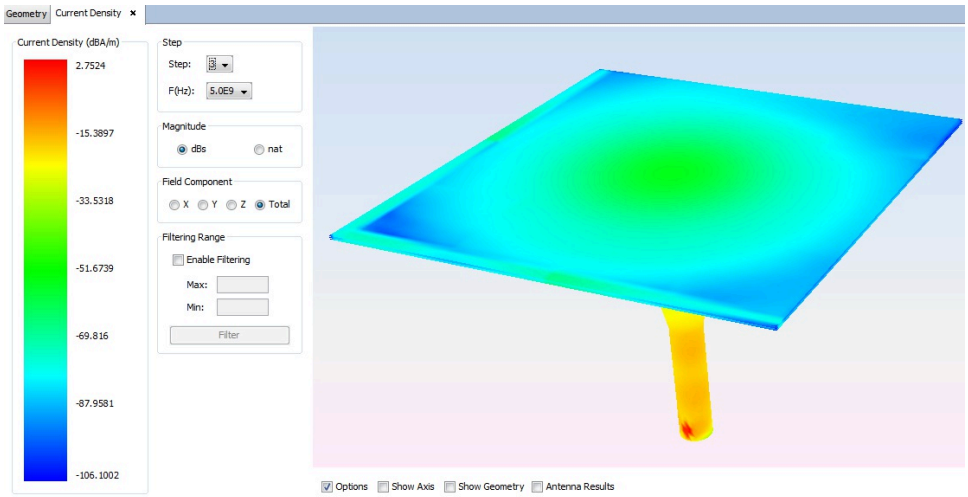


Figure 1677: Current density - Step 3: 5.0e9 Hz

- Step 4

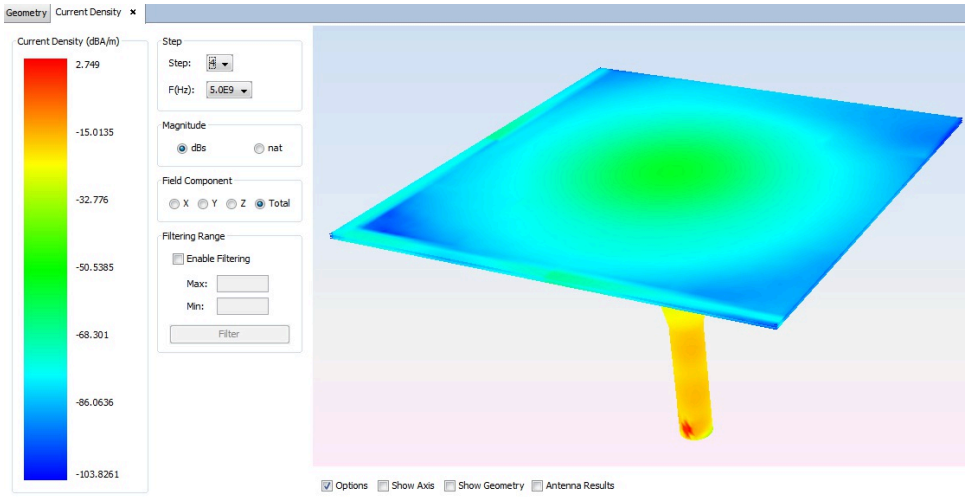


Figure 1678: Current density - Step 4: 5.0e9 Hz

- Step 5

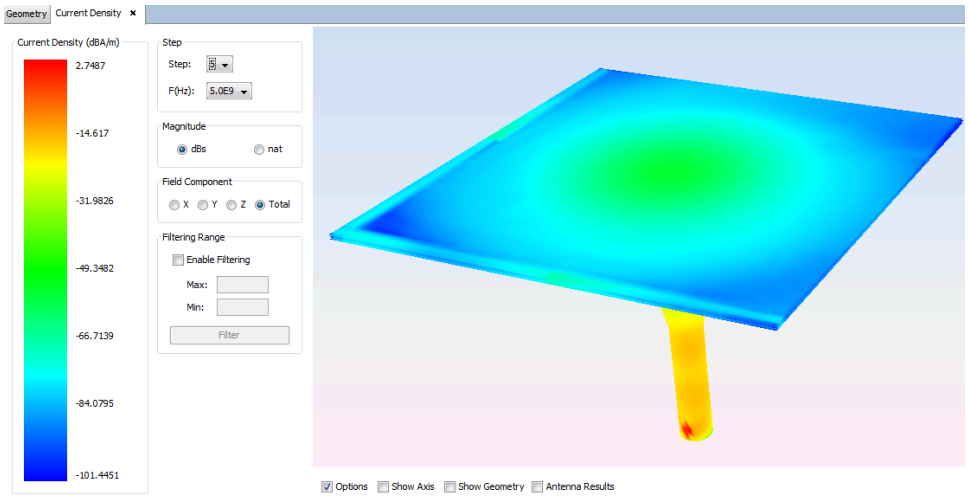


Figure 1679: Current density - Step 5: 5.0e9 Hz

Step	1	2	3	4	5
Insertion Losses (dB)	-0.006747	-0.028946	-0.000466	0.005269	0.007828
Radiated Power – Antenna (RPA) (Watts)	0.00263137				

Step	1	2	3	4	5
Radiated Power with radome (RPR) (Watts)	0.00246143	0.0024529	0.00246828	0.00245143	0.00246245
RPA/RPR (dB)	0.579888591	0.61004152	0.5557499	0.61524845	0.57628996

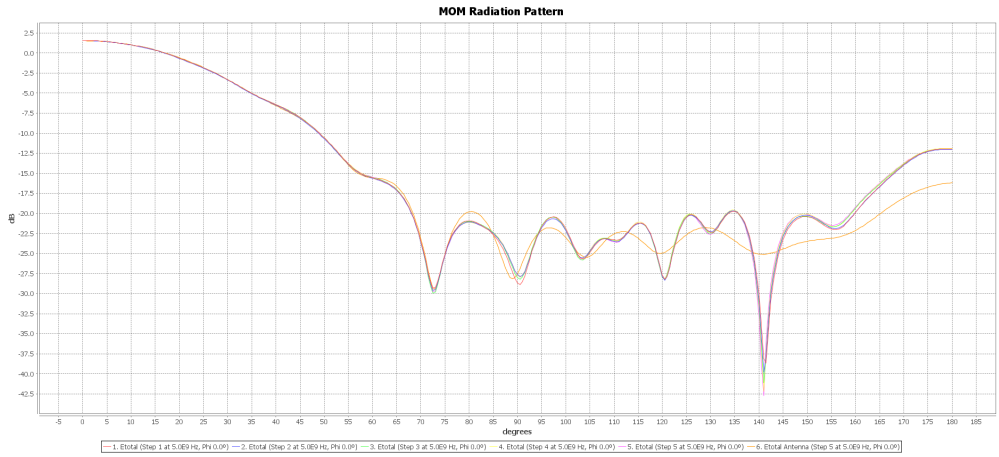


Figure 1680: Cut Phi = 0 comparison - radome steps and isolated antenna

Semi Spherical Radome

Create a copy of **baseProjectHorn.nfp** project and rename it as **Horn_SemiSphere.nfp**. Then open it with newFASANT by clicking on **File** > **Open** menu or clicking twice on the project icon.

The radome considered in this section is a semisphere of 25 centimeters of radius with the base centered at the top of the horn.

Geometric Design

Four semispheres will be generated to model the radome as a thin layer approximation. It may be achieved by using the **sphere** command, which requires both the center and the radius. The table below shows the parameters to be used for every sphere:

Sphere	X (mm)	Y (mm)	Z (mm)	Radius (mm)
1	0	0	-100	250
2	0	0	-100	250+thickness
3	0	0	-100	250+ thickness+t

Sphere	X (mm)	Y (mm)	Z (mm)	Radius (mm)
4	0	0	-100	250+(2*thickness)+t

Note that the sphere command creates the full closed sphere and we need just the upper semisphere. Then, after creating each sphere, we use the explode command to separate it in several surfaces, then remove the lower semisphere with the **delete** command, and finally we use the group command on the upper remaining surfaces to have one object for every semisphere.

```
command> sphere
Select center [x y z]: 0 0 -100
Radius [double]: 250
command> explode
command> delete
command> group
command> sphere
Select center [x y z]: 0 0 -100
Radius [double]: 250+thickness
command> explode
command> delete
command> group
command> sphere
Select center [x y z]: 0 0 -100
Radius [double]: 250+thickness+t
command> explode
command> delete
command> group
command> sphere
Select center [x y z]: 0 0 -100
Radius [double]: 250+(2*thickness)+t
command> explode
command> delete
command> group
command>
```

Figure 1681: Command line semispheres definition

After creating the four semispheres, in the tree we can see that they have been added in order as Objects, and each one has a different name.

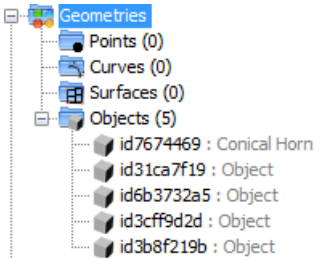


Figure 1682: List of geometric entities in the tree

Radome Definition

Having defined the geometric model, the radome must be created and assigned. To do that, click on **Source – Radome – Define Radome** menu and the Radome panel is open on right side of screen.

In the Radome panel, no radomes are created so click on **Add Radome** button to insert a new radome. Then, the table of interfaces of the selected radome (1) is loaded below. As we have shown in Figure 1655, the radome is made of **4 interfaces** that define the **3 layers**, so click on **New Interface** button **twice** to get the **4 interfaces**.

Each semisphere must be assigned to every radome interface in the same order than specified in Figure 1655. Remember that the normal vectors of every interface must be pointing to the next one. To assign a semisphere to an interface, select the geometrical object from the tree or in the Geometry panel, then select the corresponding row in the Interfaces table, and click on **Set Objects** button. Then, assign the **Material** to its interface, which is defined to the current layer.

The material (or layers) is considered between one interface and its next one, so the last interface does not required a material.

Remember clicking on **Save** button to confirm the radome definition.

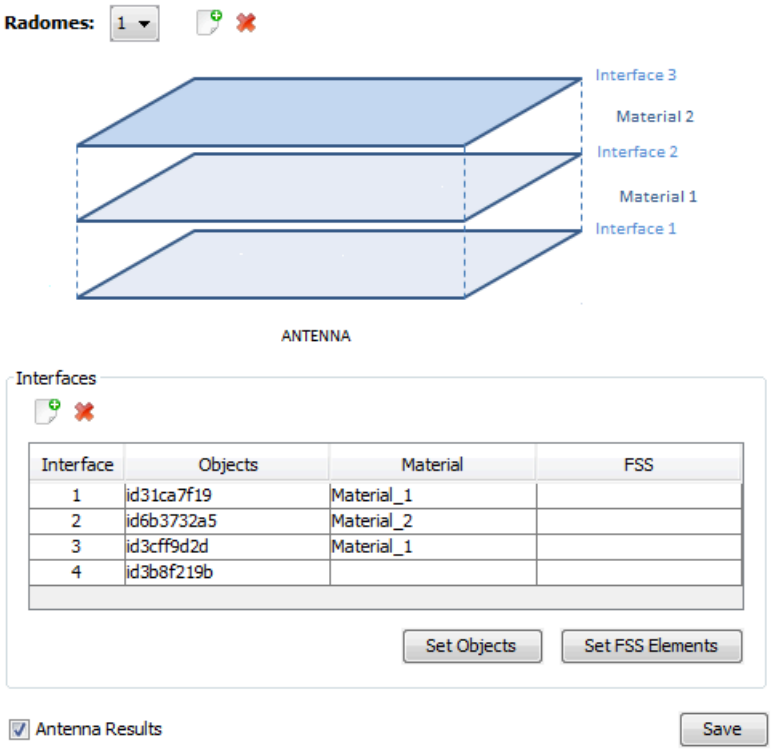


Figure 1683: 19 Radome definition.

The next figure shows the final design of the radome with the horn, with the Material Shaded view. In this representation, we verify that the layers of the radome have the correct materials.

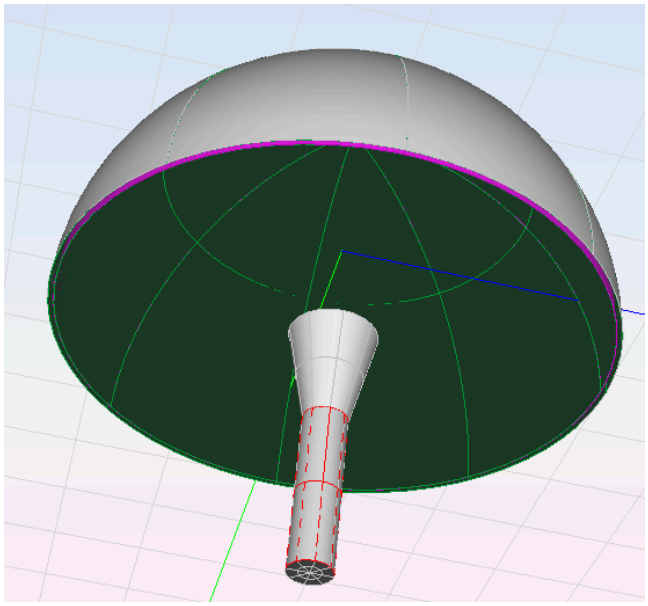


Figure 1684: 20 Shaded view with materials of the final design.

Mesh and Run

The only parameter modified in the meshing process is the number of **Processors**, which is set to 8. Specify the number of processors and click on **Mesh** buttons to confirm the changes and mesh.

The five parametric steps are meshed by requiring about 4 GB of RAM memory and 10 seconds per step.

Click on **Calculate – Execute** menu and launch the simulation with **8 processors** by clicking on **Execute** button.

The five parametric steps are simulated by requiring about 4.5 GB of RAM memory and 10 seconds per step.

Computational Specifications

Step		**1**	**2**	**3**	**4**	**5**
Number of unknowns		68816	68818	68834	68906	68906
Required RAM (GB)		4.5	4.5	4.5	4.5	4.5
Without Preconditioner	**Number of iterations**	337	331	295	248	245
	Simulation Time (min : ss)	26 : 25	25 : 46	21 : 30	19 : 31	18 : 33

Step		1	2	3	4	5
With Preconditioner	**Number of iterations**	30	27	27	29	20
	Simulation Time (min : ss)	7 : 49	7 : 29	7 : 20	7 : 09	6 : 37

Results

Radiation Pattern

- Step 1

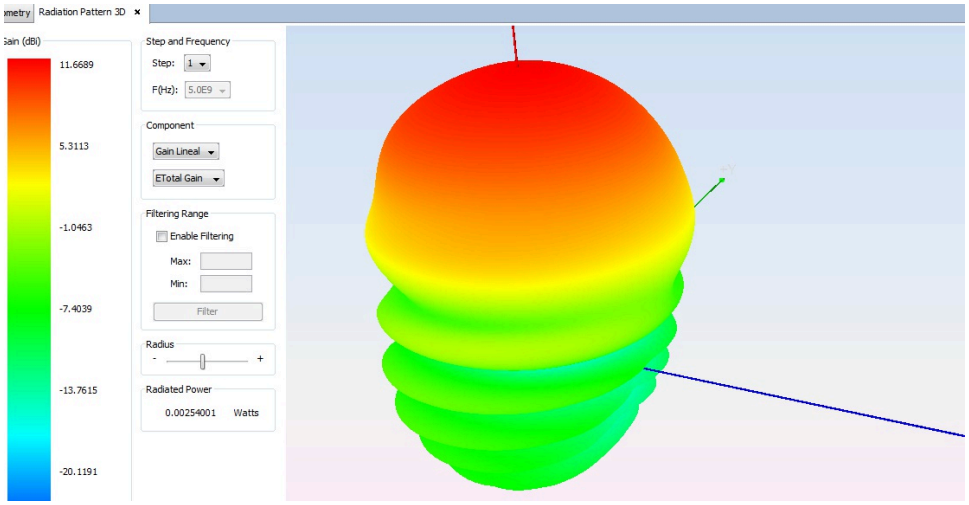


Figure 1685: 3D radiation pattern - Step 1: 5.0e9 Hz

- Step 2

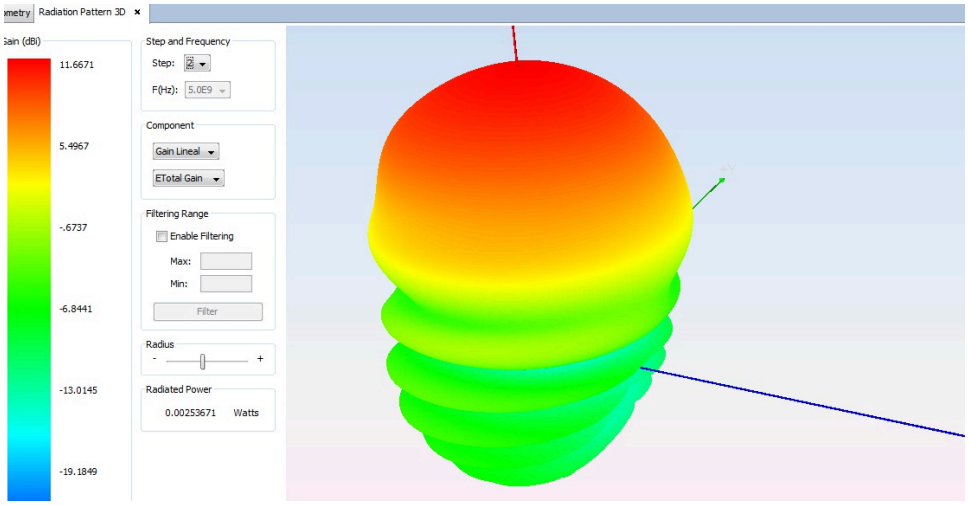


Figure 1686: 3D radiation pattern - Step 2: 5.0e9 Hz

- Step 3

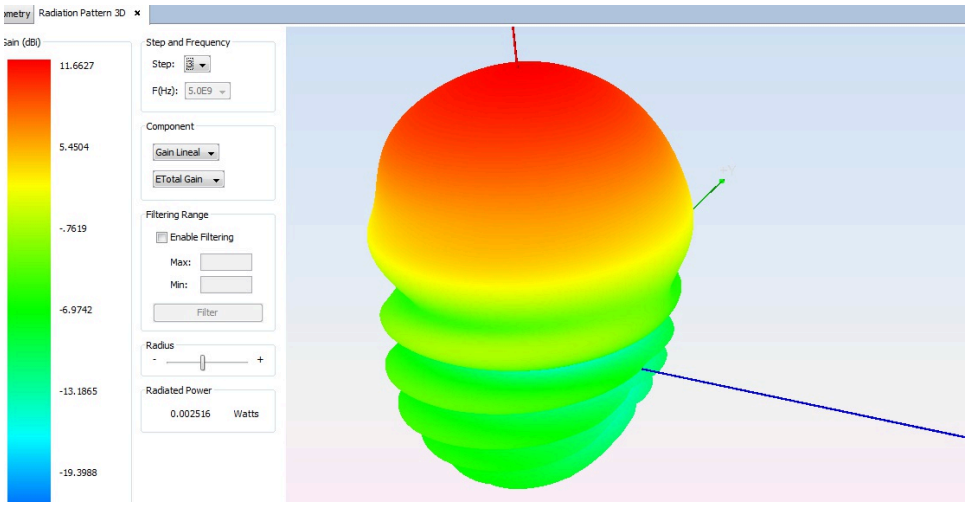


Figure 1687: 3D radiation pattern - Step 3: 5.0e9 Hz

- Step 4

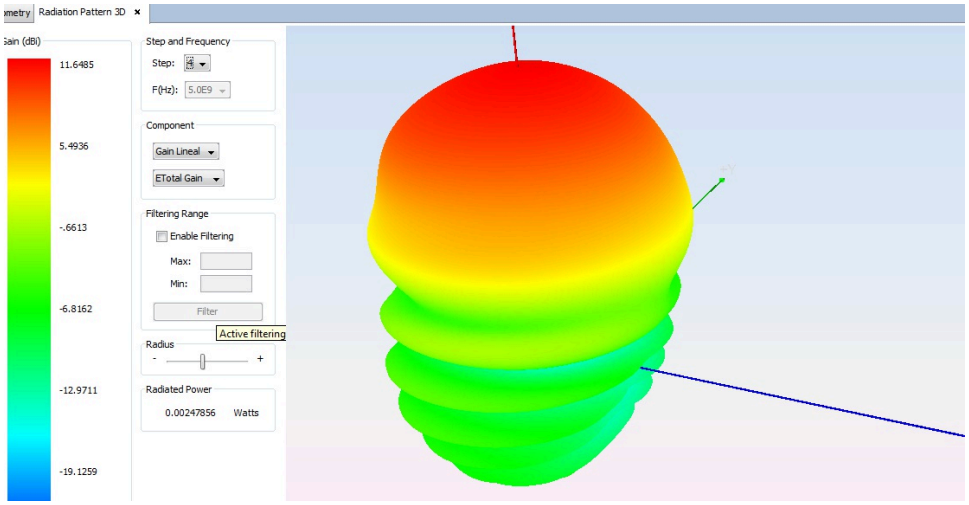


Figure 1688: 3D radiation pattern - Step 4: 5.0e9 Hz

- Step 5

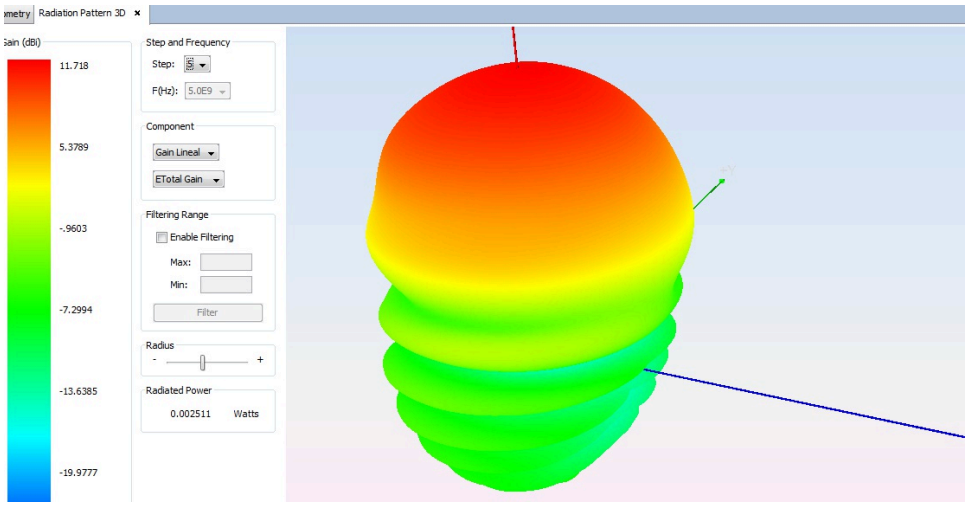


Figure 1689: 3D radiation pattern - Step 5: 5.0e9 Hz

Current Density

- Step 1

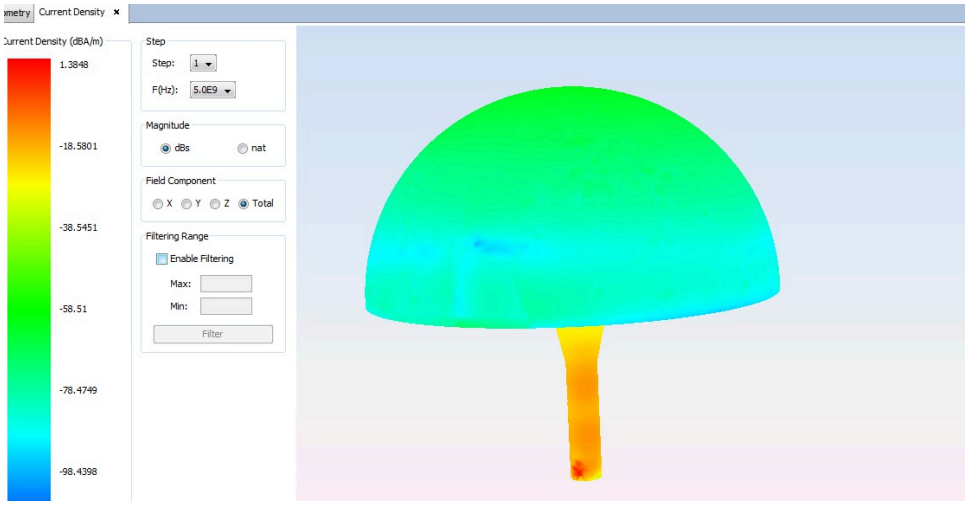


Figure 1690: Current density - Step 1: 5.0e9 Hz

- Step 2

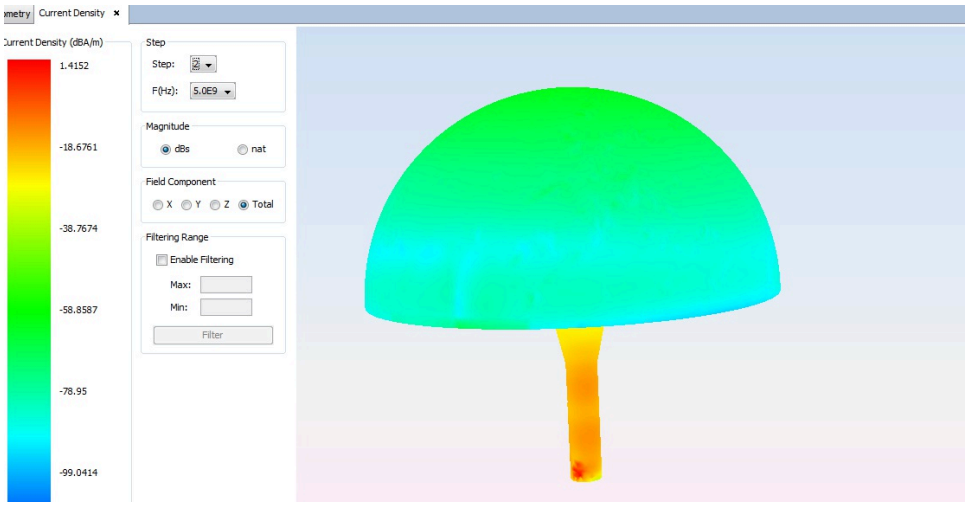


Figure 1691: Current density - Step 2: 5.0e9 Hz

- Step 3

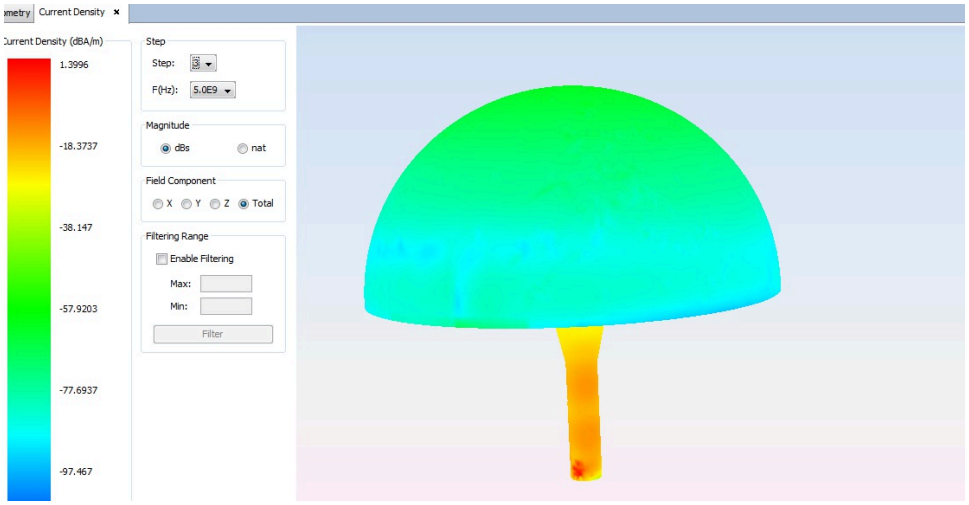


Figure 1692: Current density - Step 3: 5.0e9 Hz

- Step 4

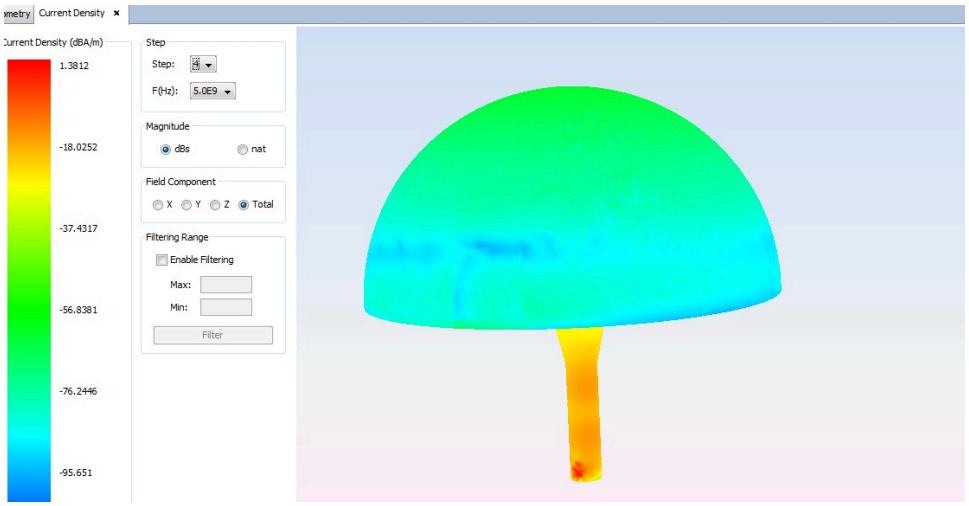


Figure 1693: Current density - Step 4: 5.0e9 Hz

- Step 5

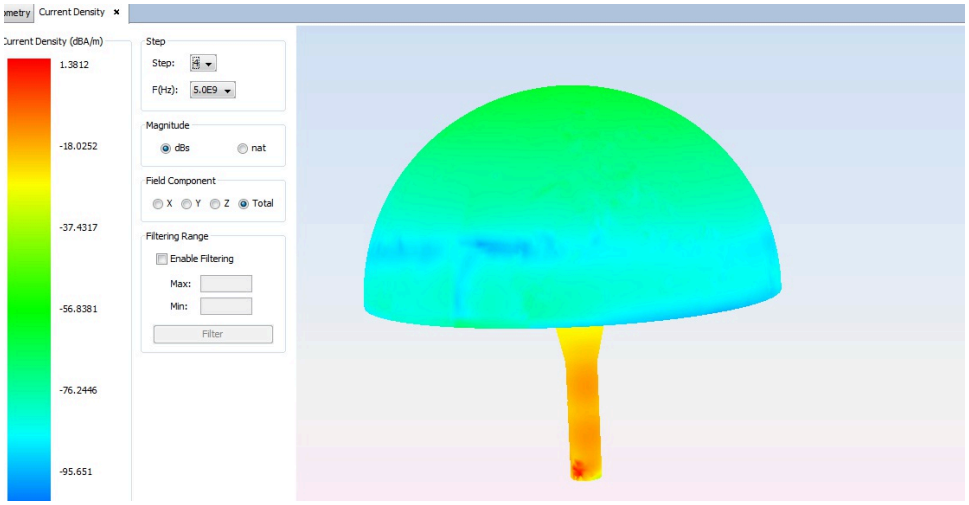


Figure 1694: Current density - Step 5: 5.0e9 Hz

Step	1	2	3	4	5
Insertion Losses (dB)	0.01443	0.049481	0.053826	0.033855	-0.017957
Radiated Power – Antenna (RPA) (Watts)	0.00263137				
Radiated Power with radome (RPR) (Watts)	0.00254001	0.00253671	0.002516	0.00247856	0.002511
RPA/RPR (dB)	0.306929852	0.31822196	0.38942565	0.51964965	0.40670413

Figure 1695: Cut Phi = 0 comparison - radome steps and isolated antenna

Custom Nose-Airplane Shape Radome

Create a copy of `baseProjectHorn.nfp` project and rename it as `Horn_Nose.nfp`. Then open it with newFASANT by clicking on **File** > **Open** or clicking twice on the project icon.

The radome considered in this section is modified ogive which resembles to the noise of an airplane. We have created its geometry by using the *userFunctions* utility included in newFASANT tools that allow the user to create geometries with its own Java programmed functions.

The geometry shape may be defined in two parts according to the next parameters and functions:

Parameter	**a**	**b**	**c**	**d**	**e**
Value	0.5	2	1.2	1.4	a*1000

Positive Y-axis half geometry definition, see Equation 11.

$$\begin{matrix} \$u \in [0,1] \\ \$v \in [0,1] \end{matrix} \begin{cases} x = a \cdot \cos\left(\frac{\pi \cdot \$v}{2}\right) \cdot \cos(\pi \cdot \$u) - \left(\frac{\$v}{b}\right)^2 \\ y = \frac{a \cdot \cos\left(\frac{\pi \cdot \$v}{2}\right) \cdot \sin(\pi \cdot \$u)}{c} \\ z = d \cdot \$v \end{cases} \tag{11}$$

Negative Y-axis half geometry definition, see Equation 12.

$$\begin{matrix} \$u \in [0,1] \\ \$v \in [0,1] \end{matrix} \begin{cases} x = a \cdot \cos\left(\frac{\pi \cdot \$v}{2}\right) \cdot \cos(\pi \cdot \$u) - \left(\frac{\$v}{b}\right)^2 \\ y = -\frac{a \cdot \cos\left(\frac{\pi \cdot \$v}{2}\right) \cdot \sin(\pi \cdot \$u)}{c} \\ z = d \cdot \$v \end{cases} \tag{12}$$

These functions may be used to interpolate a surface defined by its u and v parametric coordinates and using the **surfaceFunction** command. We recommend using it to build the same geometry.

Anyway, the next content may be copied to a text file to be imported as a newFASANT script, for example, `nose_radome.nfp`.

```
#
#   newFASANT script file to generate a Nose-Airplane shape, in milimeters
#

#required parameters
set a {0.5}
set b {2.0}
set c {1.2}
set d {1.400}
set e = 1000.0*a
#first half nose (Y-axis positive)
surfaceFunction -n id2a4dc855 -p \ #command start with main parameters
 0.0 1.0 19 \ # $u definition
 0.0 1.0 11 \ # $v definition
 "$x=a*cos(Math.PI*$v/2.0)*cos(Math.PI*$u)-Math.pow($v/b,2)" \ # x definition
 "$y=a*cos(Math.PI*$v/2.0)*sin(Math.PI*$u)/c" \ # y definition
 "$z=d*$v" \ #z definition
#second half nose (Y-axis negative)
surfaceFunction -n id5a4dc856 -p \ #command start with main parameters
 0.0 1.0 19 \ # $u definition
 0.0 1.0 11 \ # $v definition
 "$x=a*cos(Math.PI*$v/2.0)*cos(Math.PI*$u)-Math.pow($v/b,2)" \ # x definition
 "$y=-a*cos(Math.PI*$v/2.0)*sin(Math.PI*$u)/c" \ # y definition
 "$z=d*$v" #z definition
#set the second half normal vectors pointing to -Y
invertNormals -s id5a4dc856
#group the two halfs in the same object
group -s id2a4dc855 id5a4dc856 -n id31a94b3d
#scale the objects from meters to millimetres
scale -s id31a94b3d -p 1000.0
```

This script contains several commands:

- The first section defines the required values for creating the geometry.
- The second section generates two surfaces according to the formulas given previously.
- The last section processes the normal vectors orientation, group the surfaces in an only object and scale it from meters to millimeters.

Geometric Design

Four nose surfaces are generated to model the radome as a thin layer approximation. To do that, click on **Tools – Script – Load** menu and load the `nose_radome.nfp` script that provides the internal layer of the radome. Not that the script is ready for loading it in millimetres because of its last command.

As it is not a canonical shape, we can approximate the next interfaces by scaling the original one. Use three times the command *duplicate* to create three additional copies of the generated shape.

The selected dimension as reference is the X-one, which has a distance to the origin of 500 millimetre, given by the e parameter (it is the a parameter, in meters, scaled to millimetres). As we need that the reference dimensions varies according to the thickness and *t***parameters**, the scale factor is computed as the desired size divided by the original one.

Use the **scale** command to scale the three copies of the nose shape, each one by the corresponding factor:

Table 9:

Plane	Scale Factor
2	(e+thickness)/e
3	(e+thickness+t)/e
4	(e+(2*thickness)+t)/e

```
command> script -load "C:\nose_mm.nfs"
command> set a {0.5}
command> set b {2.0}
command> set c {1.2}
command> set d {1.400}
command> set e = 1000.0*a
e = 500.0
command> surfaceFunction -n id2a4dc855 -p  0.0 1.0 19  0.0 1.0 11  "$x=a*cos(Math.PI*$v/2.0)*cos(Math.PI*$u)-Math.pow($v/b,2)"  "$y=a*cos(Math.PI*$v/2.0)*sin(Math.PI*$u)/c"  "$z=d*$v"
command> surfaceFunction -n id5a4dc856 -p  0.0 1.0 19  0.0 1.0 11  "$x=a*cos(Math.PI*$v/2.0)*cos(Math.PI*$u)-Math.pow($v/b,2)"  "$y=-a*cos(Math.PI*$v/2.0)*sin(Math.PI*$u)/c"  "$z=d*$v"
command> invertNormals -s id5a4dc856
command> group -s id2a4dc855 id5a4dc856 -n id31a94b3d
command> scale -s id31a94b3d -p 1000.0
command> duplicate
command> duplicate
command> duplicate
command> scale
Scale factor [double]: (e+thickness)/e
command> scale
Scale factor [double]: (e+thickness+t)/e
command> scale
Scale factor [double]: (e+(2*thickness)+t)/e
command>
```

Figure 1696: Command line semispheres definition

After creating the four noses, note that the base is centred at the origin of coordinates, so they must be moved to the horn.

Use the **move** command to move the four noses from *0 0 0* to *0 0 -100* point.

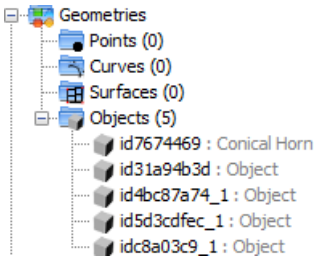


Figure 1697: List of geometric entities in the tree

Radome Definition

Having defined the geometric model, the radome must be created and assigned. To do that, click on **Source – Radome – Define Radome** menu and the Radome panel is open on right side of screen.

In the Radome panel, no radomes are created so click on **Add Radome** button to insert a new radome. Then, the table of interfaces of the selected radome (1) is loaded below. As we have shown in Figure 1698, the radome is made of 4 interfaces that define the **3 layers**, so click on **New Interface** button **twice** to get the 4 interfaces.

Each nose must be assigned to every radome interface in the same order than specified in Figure 1698. Remember that the normal vectors of every interface must be pointing to the next one. To assign a nose to an interface, select the geometrical object from the tree or in the Geometry panel, then select

the corresponding row in the Interfaces table, and click on **Set Objects** button. Then, assign the **Material** to its interface, which is defined to the current layer.

The material (or layers) is considered between one interface and its next one, so the last interface does not required a material.

Remember clicking on **Save** button to confirm the radome definition.

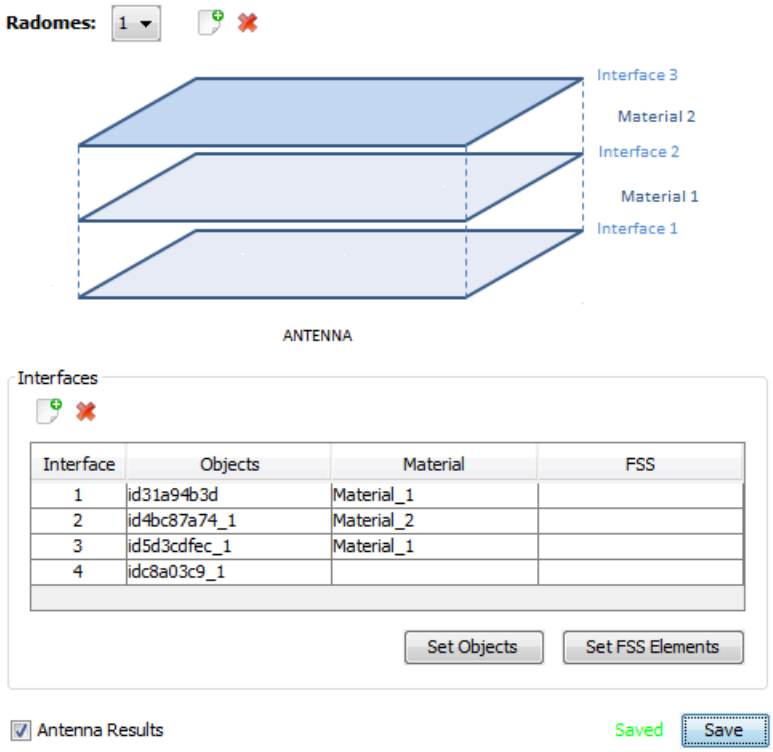


Figure 1698: 27 Radome definition.

The next figure shows the final design of the radome with the horn, with the Material Lines view. In this representation, we verify that the layers of the radome have the correct materials.

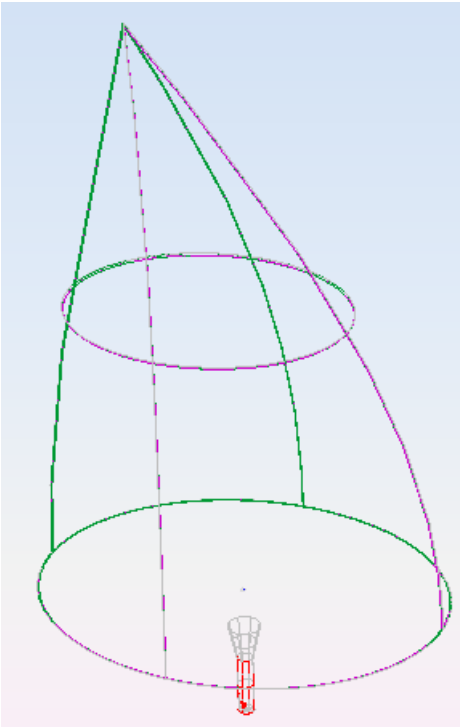


Figure 1699: 28 Lines view with materials of the final design.

Mesh and Run

Note that it is an electrically large case, so we have run it in a workstation

The only parameter modified in the meshing process is the number of **Processors**, which is set to 16. Specify the number of processors and click on **Mesh** buttons to confirm the changes and mesh.

The five parametric steps are meshed by requiring about 7 GB of RAM memory and 1 minute per step.

Click on **Calculate – Execute** menu and launch the simulation with **16 processors** by clicking on **Execute** button.

The five parametric steps are simulated by requiring about 4.5 GB of RAM memory and 10 seconds per step.

Computational Specifications

Step		**1**	**2**	**3**	**4**	**5**
Number of Unknowns		454,780	454,983	455,342	455,780	456,010
Required RAM (GB)		10	10	10	10	10
With Preconditioner	**Number of Iterations**	28	28	30	29	29

Step		1	2	3	4	5
	Simulation Time (min : ss)	31 : 36	30 : 15	29 : 53	31 : 23	30 : 40

Results

Radiation Pattern

Step 1

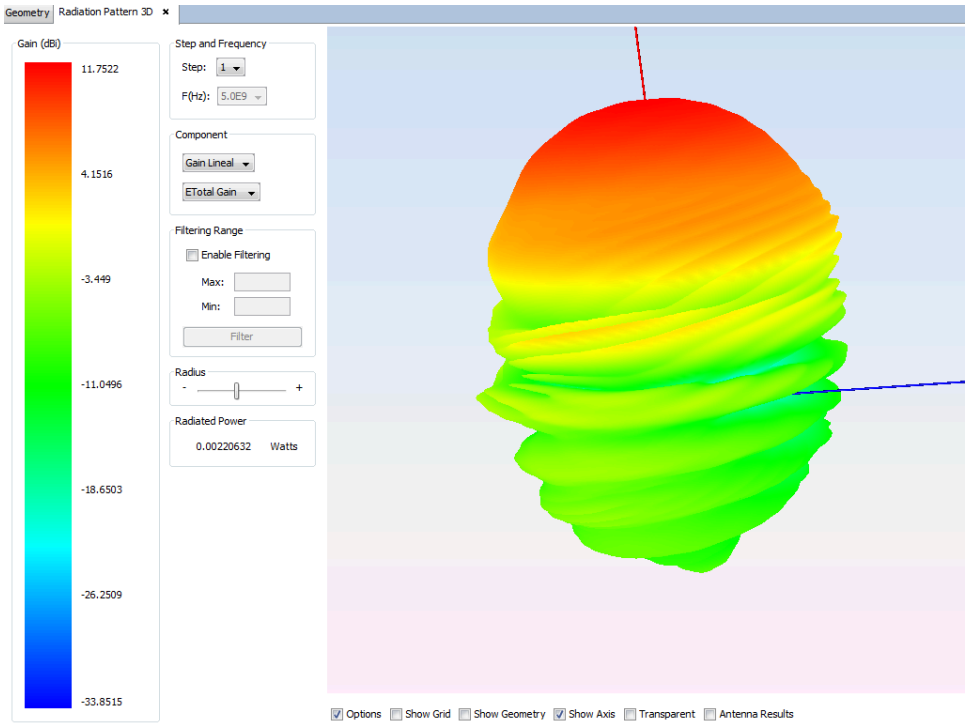


Figure 1700: 3D radiation pattern - Step 1: 5.0e9 Hz

Step 2

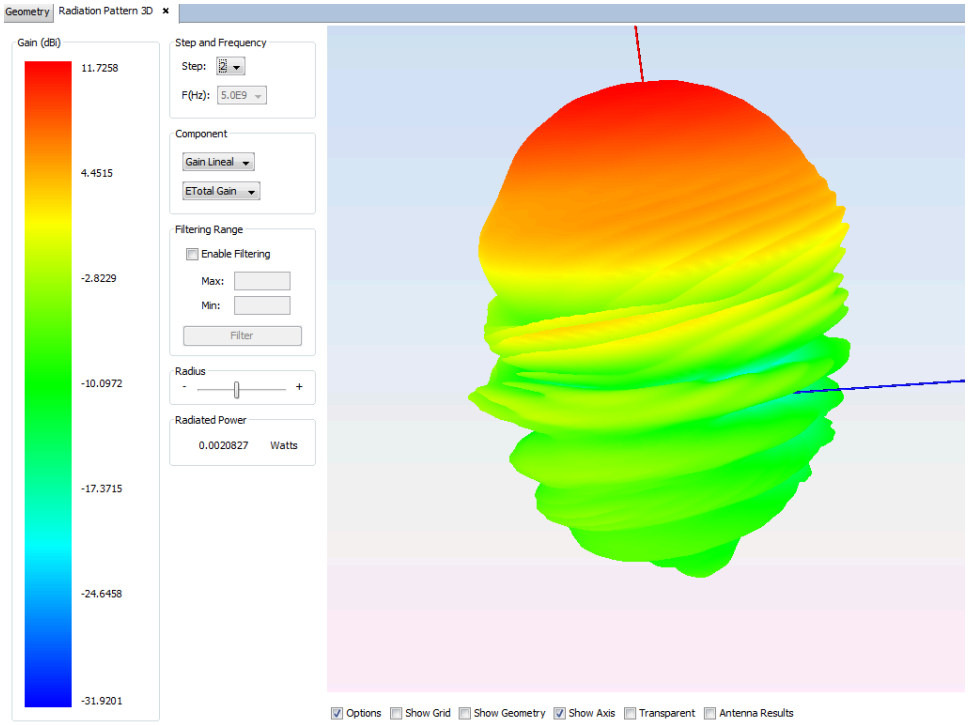


Figure 1701: 3D radiation pattern - Step 2: 5.0e9 Hz

Step 3

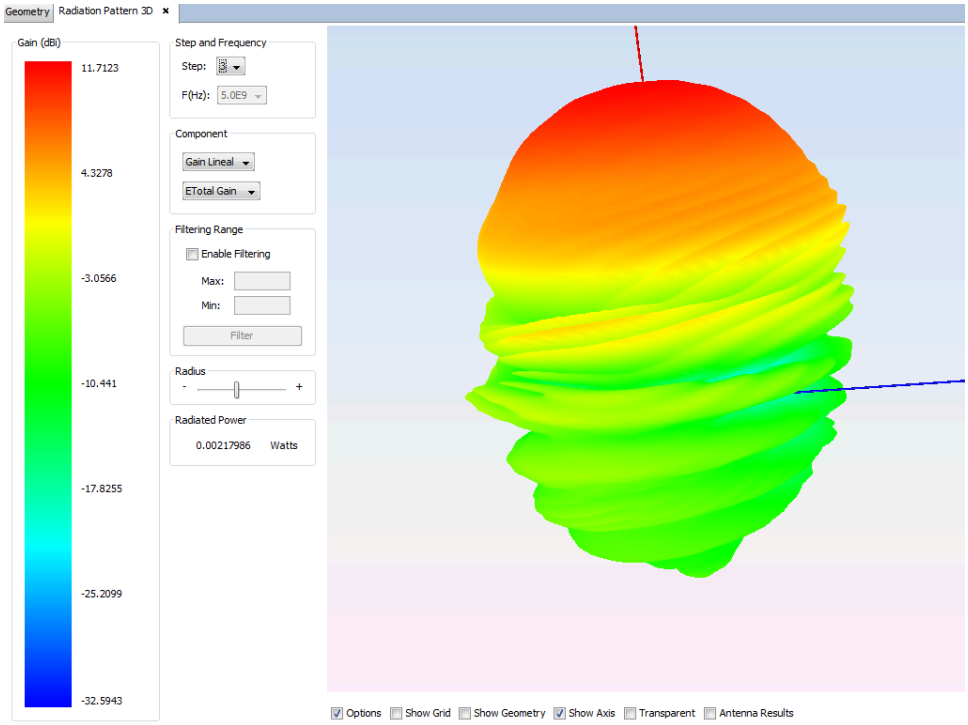


Figure 1702: 3D radiation pattern - Step 3: 5.0e9 Hz

Step 4

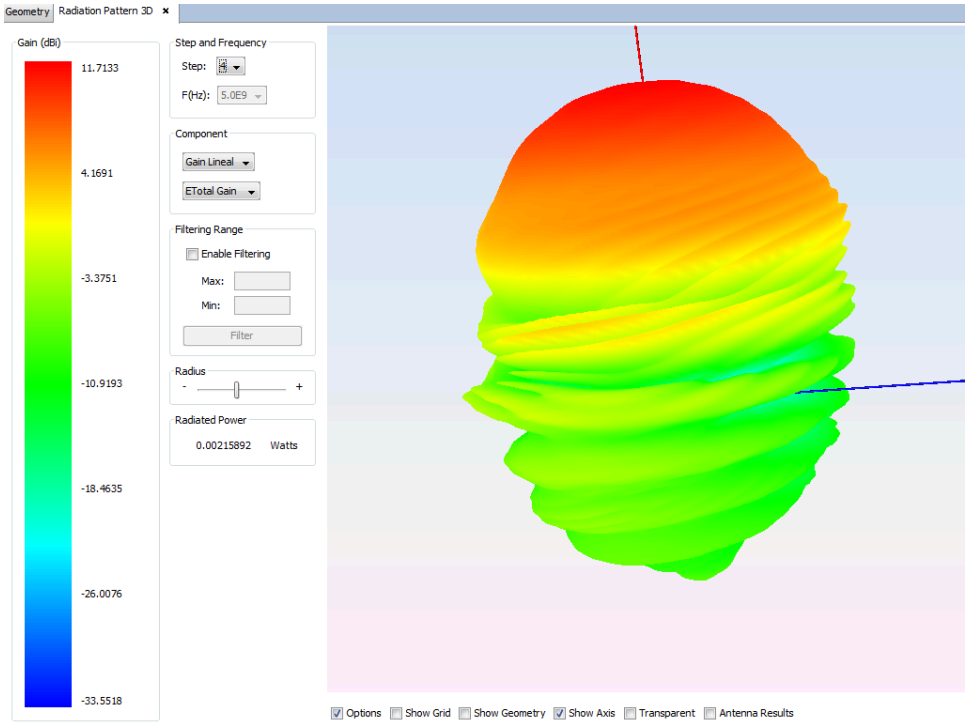


Figure 1703: 3D radiation pattern - Step 4: 5.0e9 Hz

Step 5

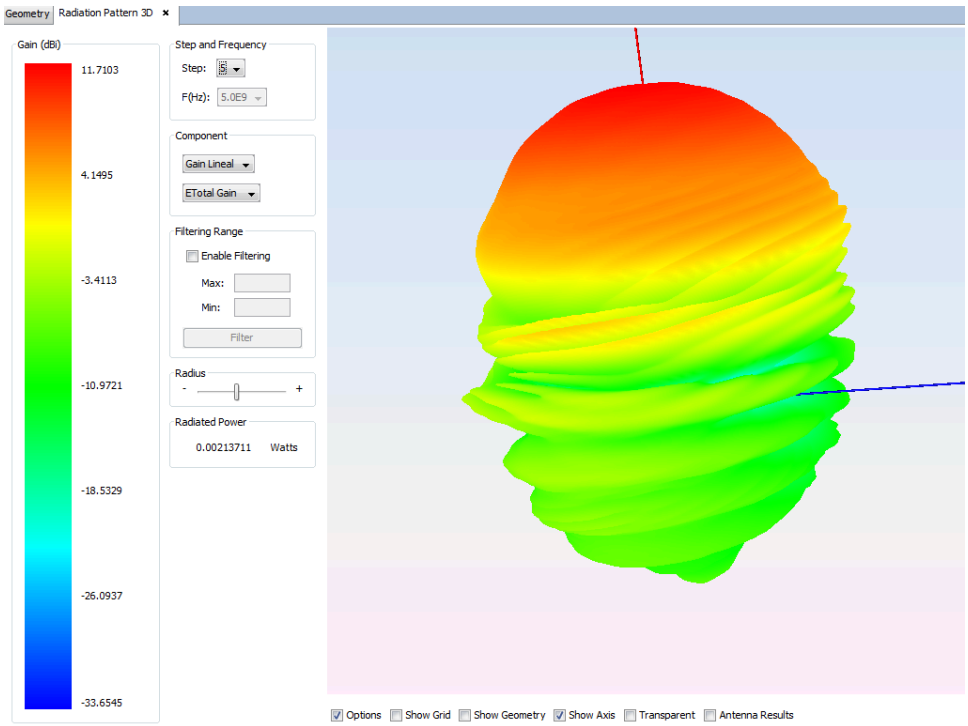


Figure 1704: 3D radiation pattern - Step 5: 5.0e9 Hz

Current Density

Step 1

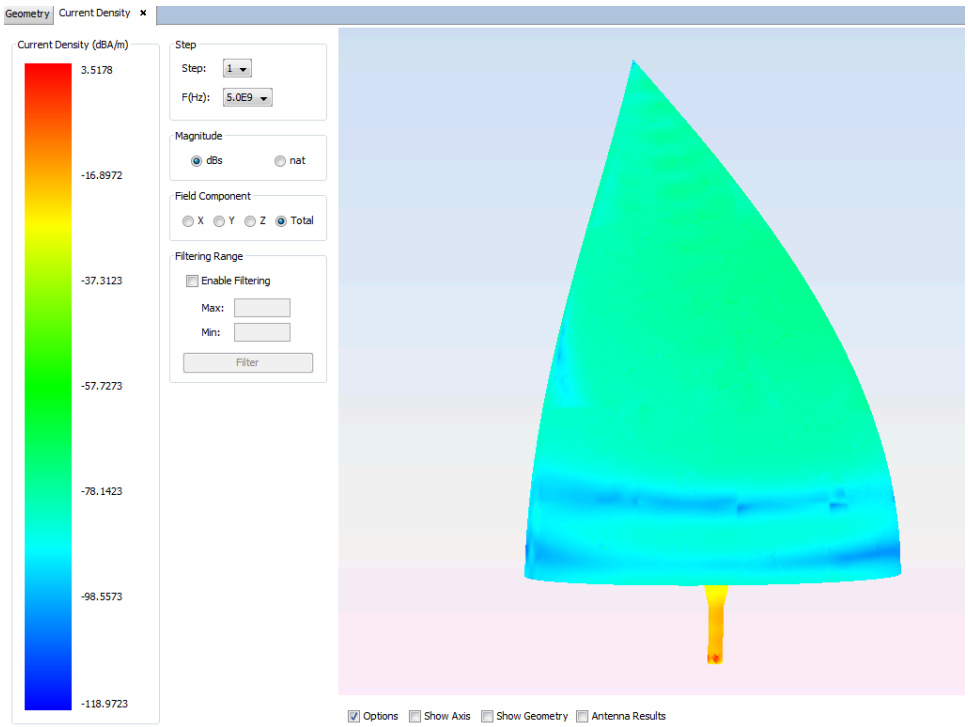


Figure 1705: Current density - Step 1: 5.0e9 Hz

Step 2

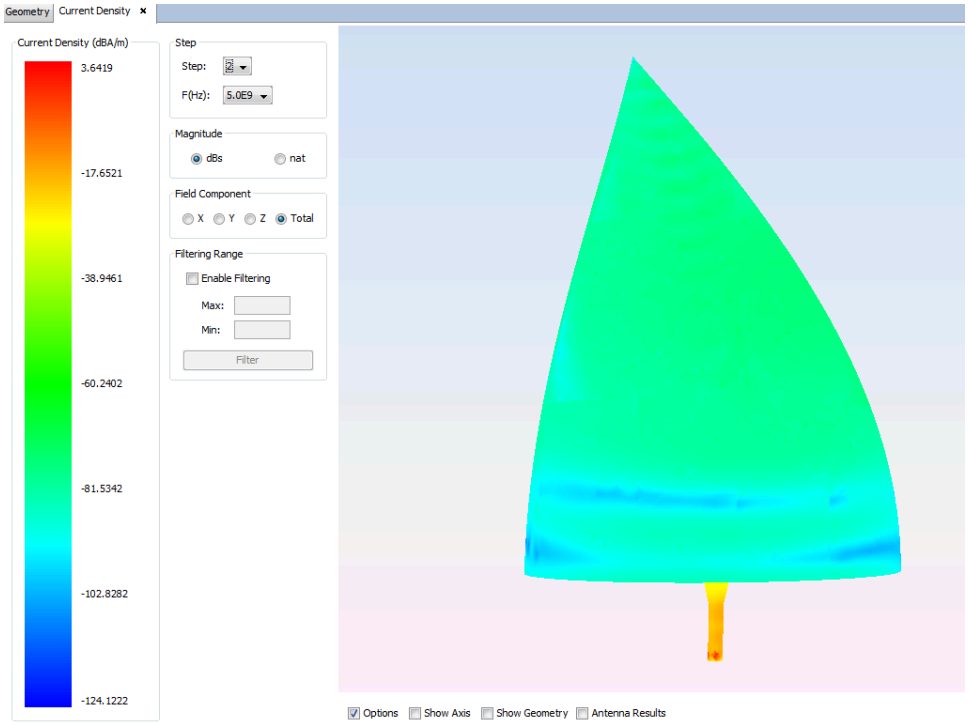


Figure 1706: Current density - Step 2: 5.0e9 Hz

Step 3

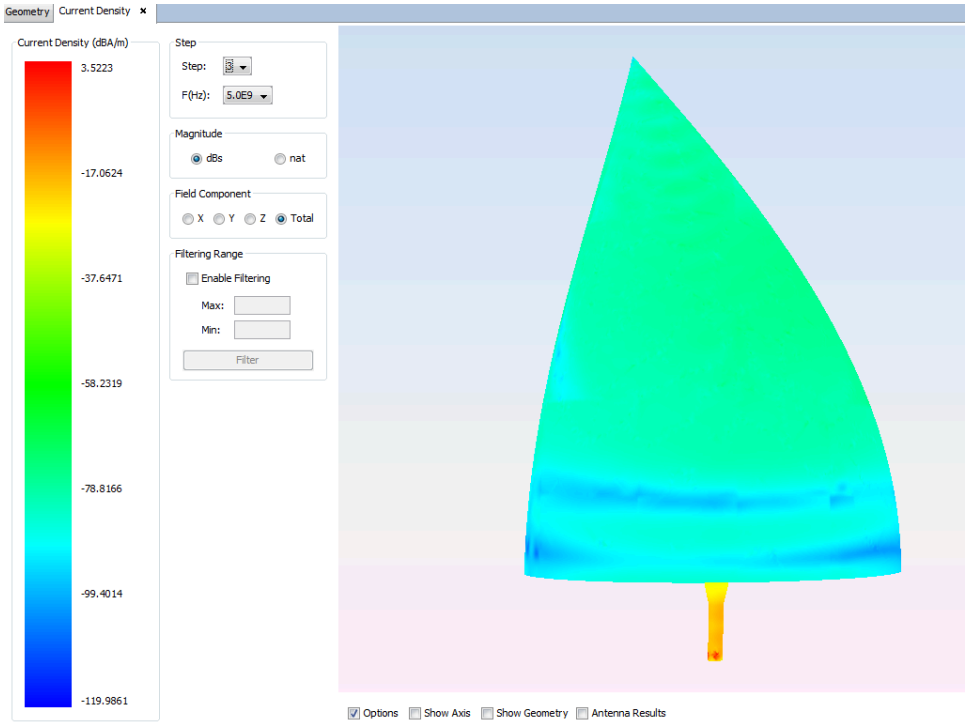


Figure 1707: Current density - Step 3: 5.0e9 Hz

Step 4

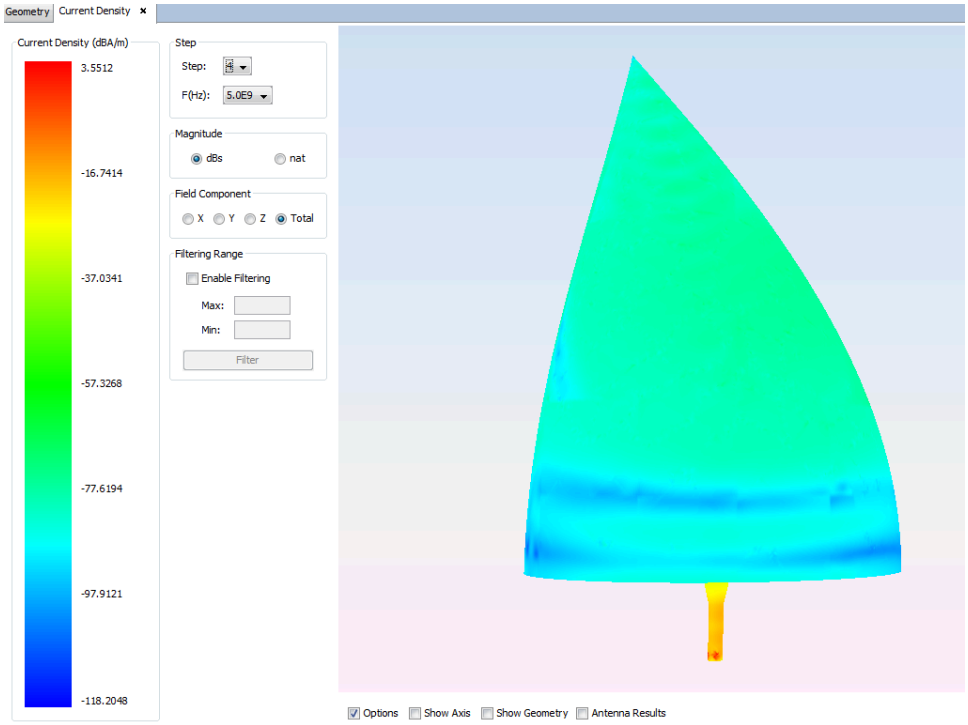


Figure 1708: Current density - Step 4: 5.0e9 Hz

Step 5

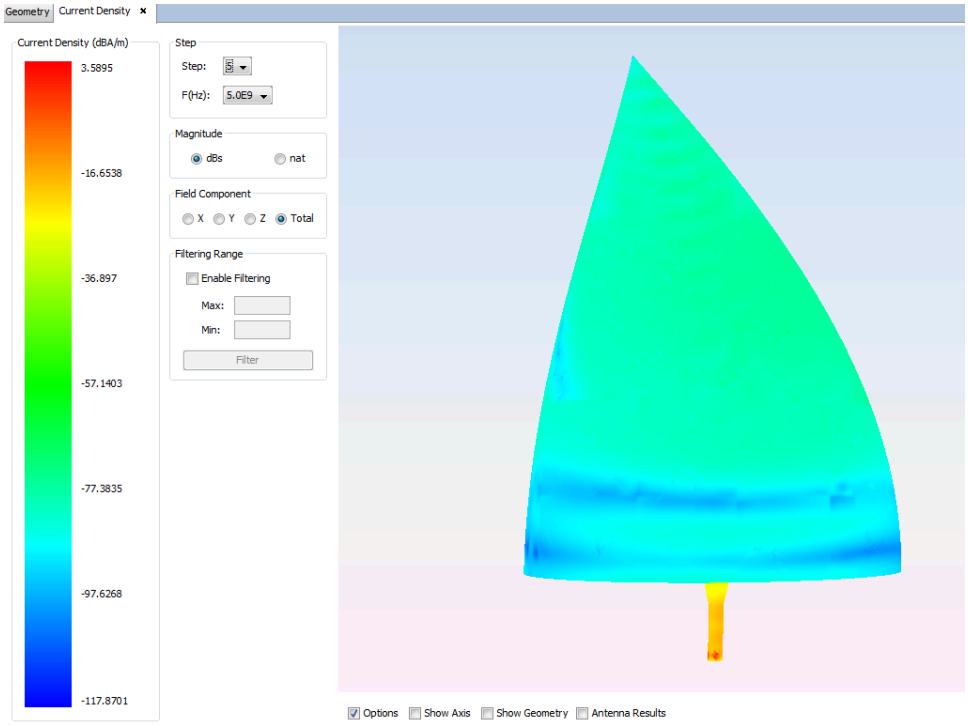


Figure 1709: Current density - Step 5: 5.0e9 Hz

Step	1	2	3	4	5
Insertion Losses (dB)	-0.15372	-0.14825	-0.1358	-0.13691	-0.13398
Radiated Power – Antenna (RPA) (Watts)	0.00263137				
Radiated Power with radome (RPR) (Watts)	0.00220632	0.0020827	0.00217986	0.00215892	0.00213711
RPA/RPR (dB)	1.530268344	2.03110404	1.63506634	1.71890739	1.80710085

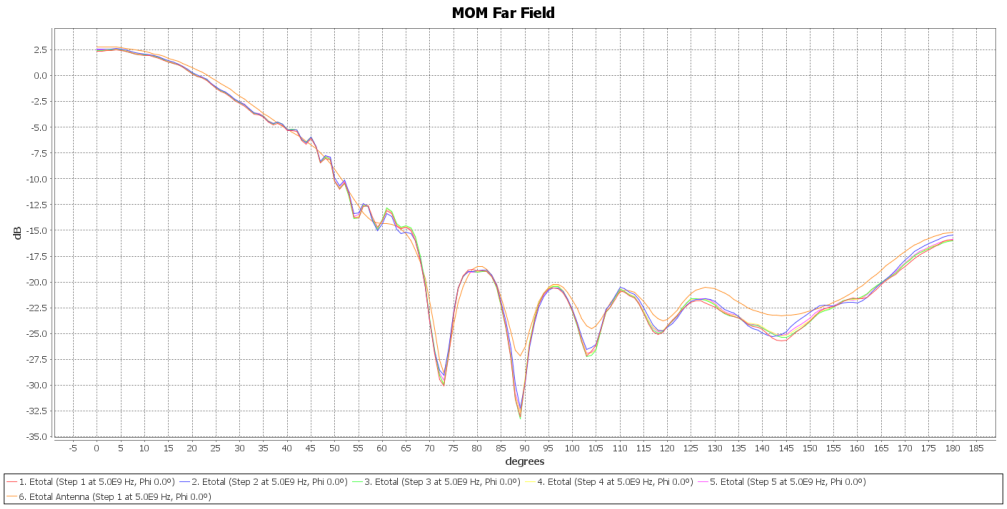


Figure 1710: Cut Phi = 0 comparison - radome steps and isolated antenna

Example 2: Radome with FSS

This document shows an example to create a radome with FSS. The table below resumes the parameters taken into account in the following example:

Data	Definition	Values
Frequency	Simulation Frequency	10 GHz

Data	Definition	Values
Material	Teflon	Er = 1.9
Thickness	Radome thickness	4 mm
FSS	Radome FSS	Cross

Create a new MOM project. To do that, click on **New Project** button and select the **MOM** module.

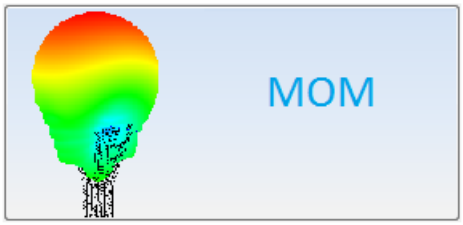


Figure 1711: 1 Choose the MOM module.

Set the frequency. Click on **Simulation –> Parameters** menu and then the Simulation panel is open on right side of screen. Set the frequency to 10 GHz and click on **Save** button before closing the panel.

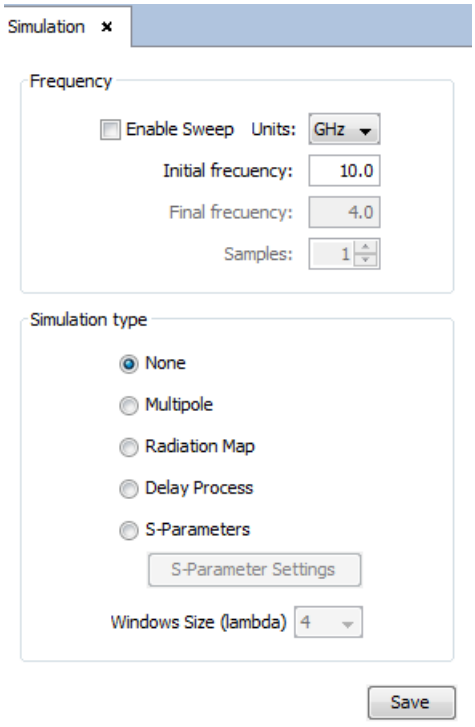


Figure 1712: 2 Simulation Parameters

Set the feeding. Click on **Source -> Primitive Antennas -> Horn -> Pyramidal Horn** menu and click on **Add** with default parameters. Note The default parameters of the horn are calculated automatically for the simulation frequency.

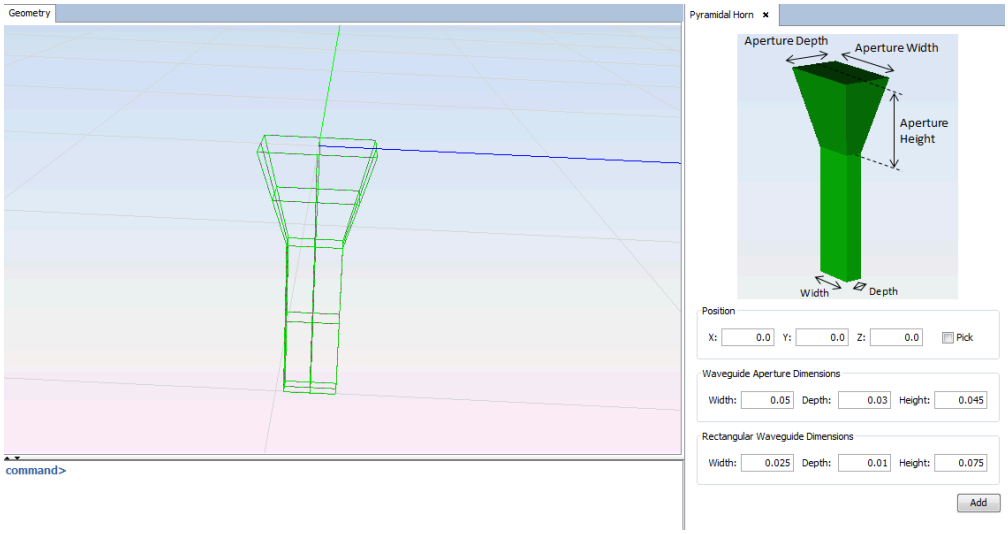


Figure 1713: 3 Set up a Pyramidal Horn

Set the Radome Geometry. To create the radome geometry follow this steps:

1. Change Y axis of **Reference Plane** to Z axis. Click on **View -> Reference Plane** and set Y-axis to (0.0, 0.0, 1.0) and then click on **Update** button.

2. Draw an **arc** with the following parameters. Click on **Geometry -> Curve -> Arc**.

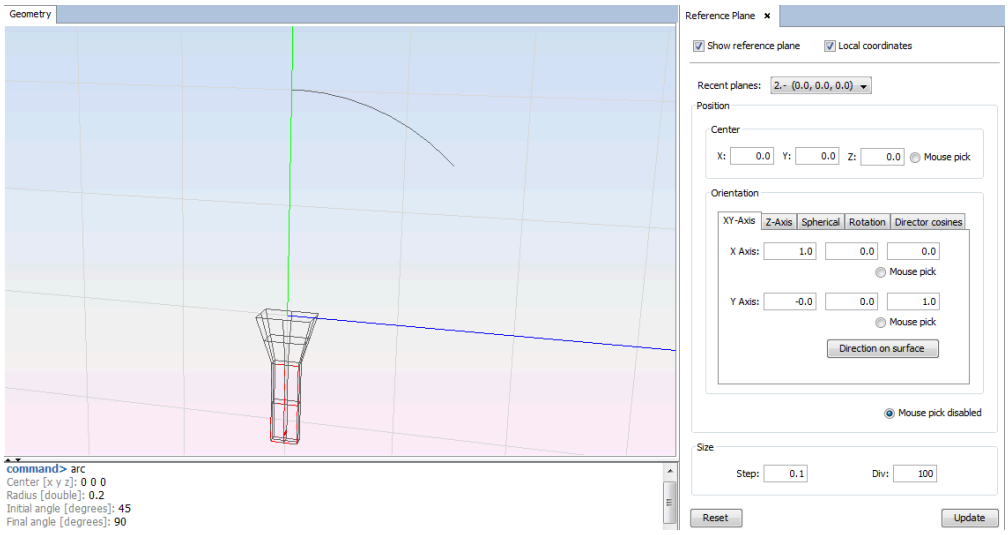


Figure 1714: 4 Change Reference plane and draw an arc

3. Draw another **arc** with the following parameters. The **thickness** between the two arcs is **4mm**.

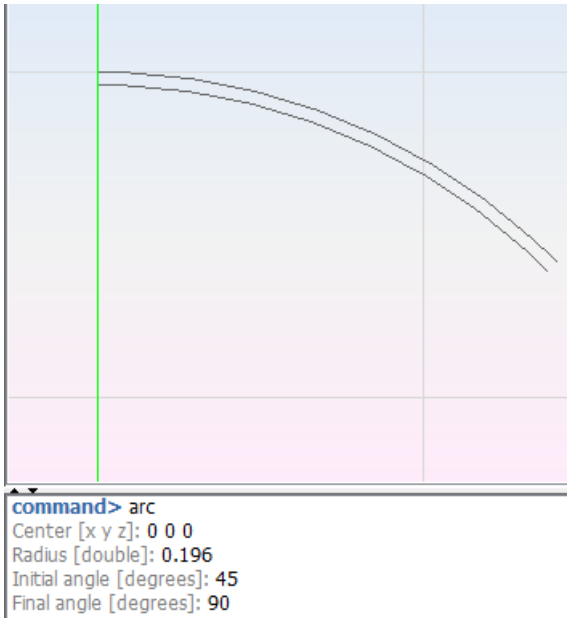
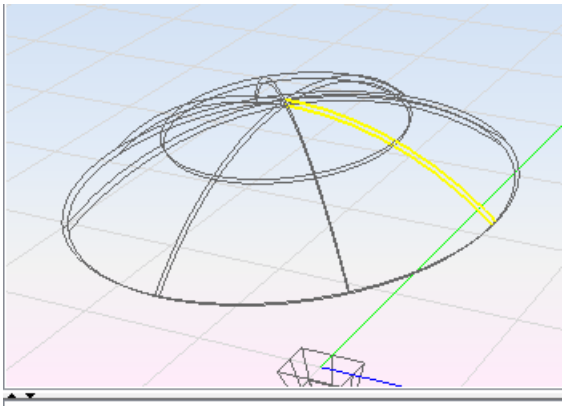


Figure 1715: 5 Second arc with 4mm of thickness

4. Click on **Reset** button of the **Reference Plane** panel.

5. **Revolve** arcs. Click on **Geometry -> Surface -> Revolve** and select the two arcs. For a better meshing create the surfaces with 90 degrees of revolution.



```
command> revolve
Axis point 1 [x y z]: 0 0 0
Axis point 2 [x y z]: 0 0 1
Initial Angle [degrees] (default=0.0):
Final Angle [degrees] (default=360.0): 90
command> revolve
Axis point 1 [x y z]: 0 0 0
Axis point 2 [x y z]: 0 0 1
Initial Angle [degrees] (default=0.0): 90
Final Angle [degrees] (default=360.0): 180
command> revolve
Axis point 1 [x y z]: 0 0 0
Axis point 2 [x y z]: 0 0 1
Initial Angle [degrees] (default=0.0): 180
Final Angle [degrees] (default=360.0): 270
command> revolve
Axis point 1 [x y z]: 0 0 0
Axis point 2 [x y z]: 0 0 1
Initial Angle [degrees] (default=0.0): 270
Final Angle [degrees] (default=360.0): 360
```

Figure 1716: 6 Revolve arcs

Set the FSS. To create a conformed FSS on the radome surface folow this steps:

1. Draw the **FSS Element**. Click on **Source -> Radome -> FSS Primitives -> Crosses -> Cross** and click on **Save** button with default parameters. The FSS Element must be on XY plane and centered on (0.0, 0.0, 0.0).

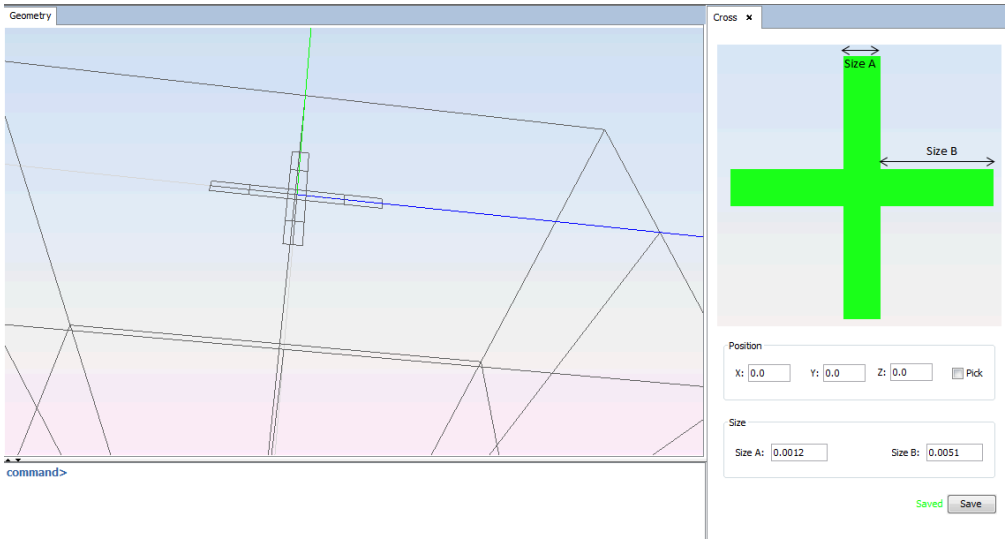


Figure 1717: 7 FSS Element

2. Select the upper surfaces of the radome and click on **Edit -> Geometric Operations -> Arrays -> Array On Surface**.

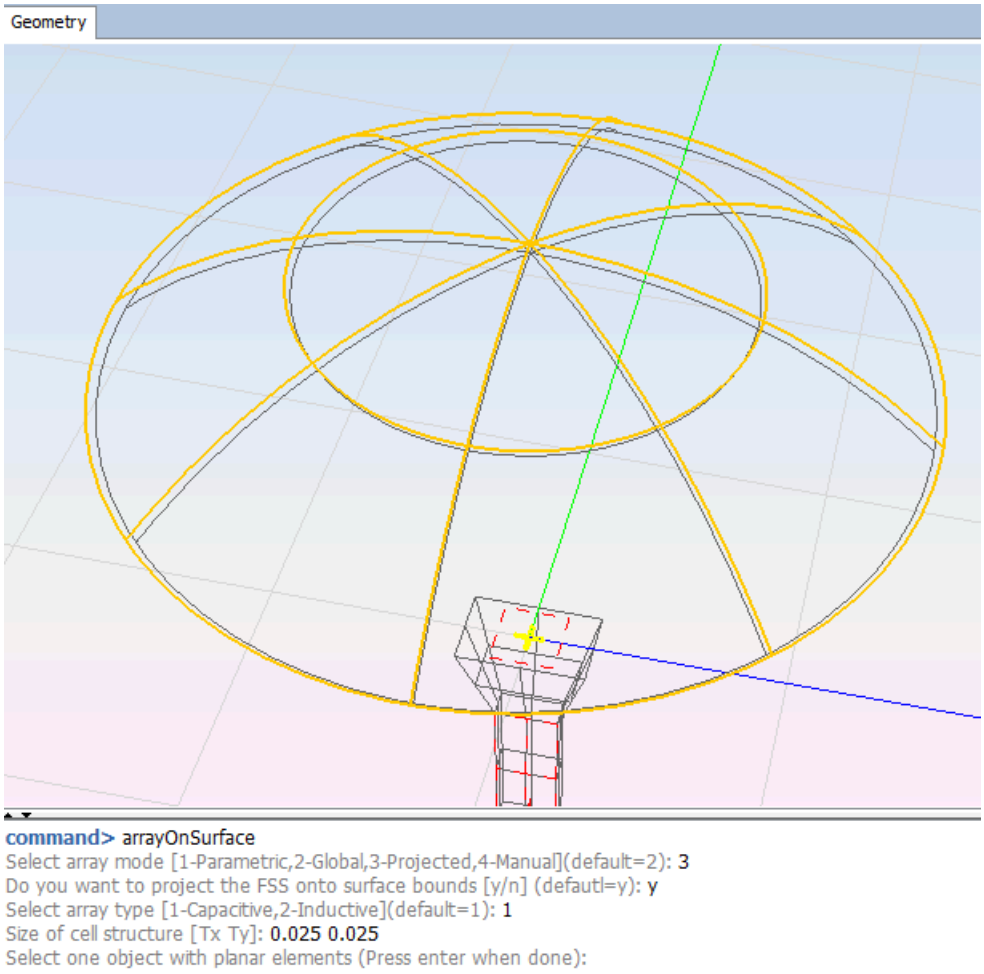


Figure 1718: 8 Set up a conformed FSS on Radome Surface

3. Group **radome interfaces**. Select the **upper** and **lower** surfaces of the radome and **group** them separately.

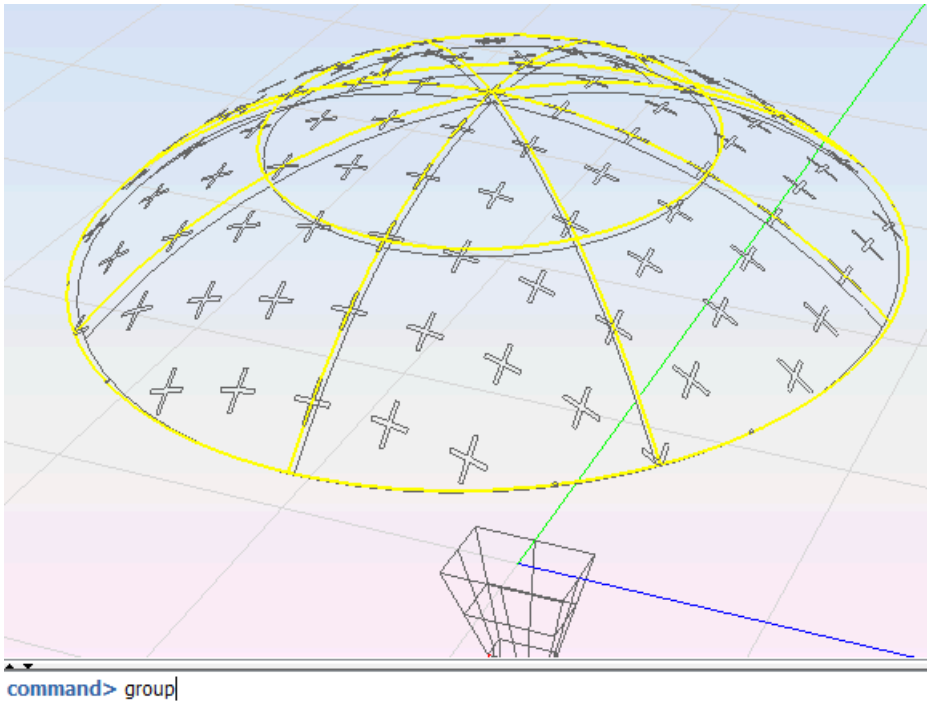


Figure 1719: 9 Group Radome Interfaces

4. **Remove** the **crosses** that are not closed and the **FSS Element**. Select crosses and click on **Edit -> Explode** and then select the individual cross and click on **Edit -> Delete**.

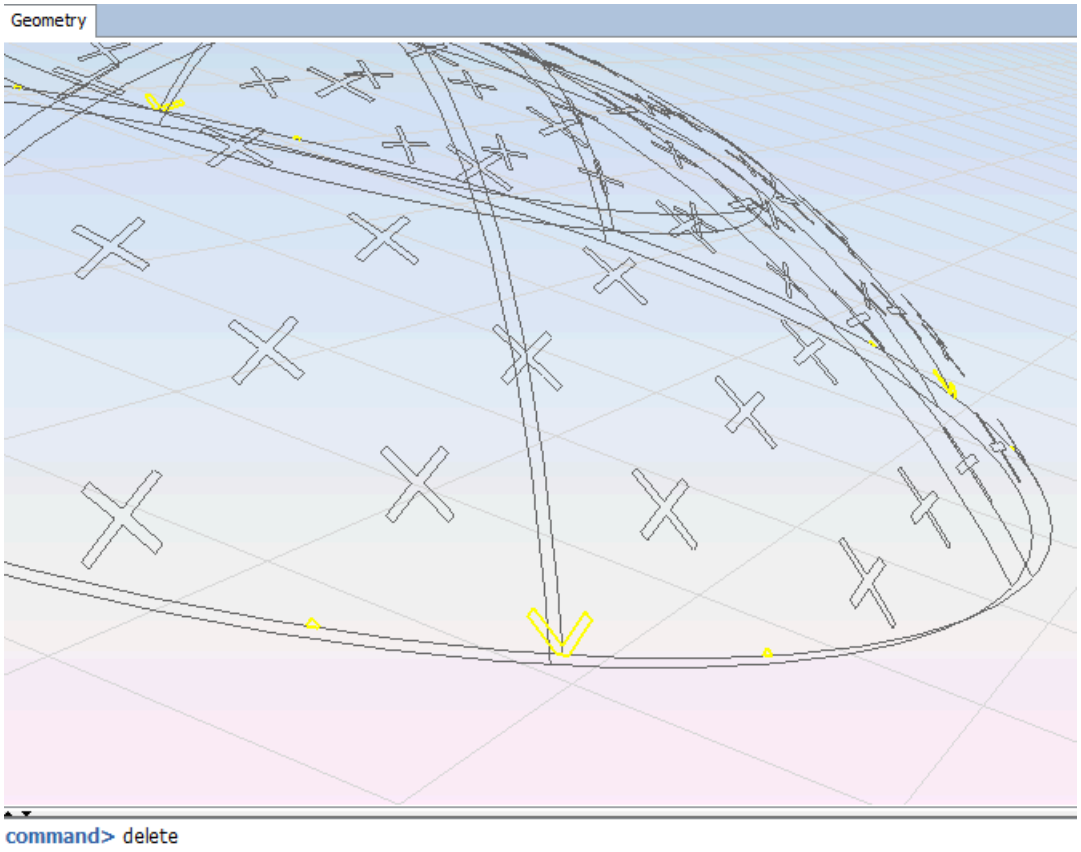


Figure 1720: 10 Remove FSS Element and individual crosses.

5. **Group** the **crosses** again. Crosses can be **selected** by clicking on the **tree** surfaces folder. Click on **Edit -> Group**.

6. It is possible to give a **thickness** to the **crosses**. In this case select the crosses and click on **Geometry -> Solid -> Extrude Normal**. Set the thickness to **1mm**.

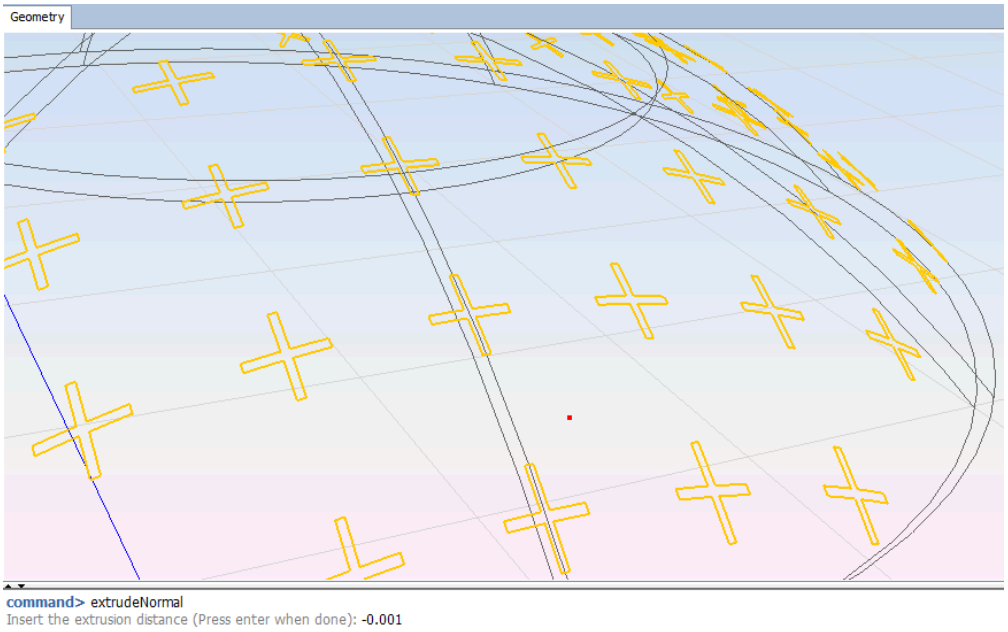


Figure 1721: 11 Set a thickness to the crosses

7. Group the crosses with their thickness and rename the objects as you can see in the following image.

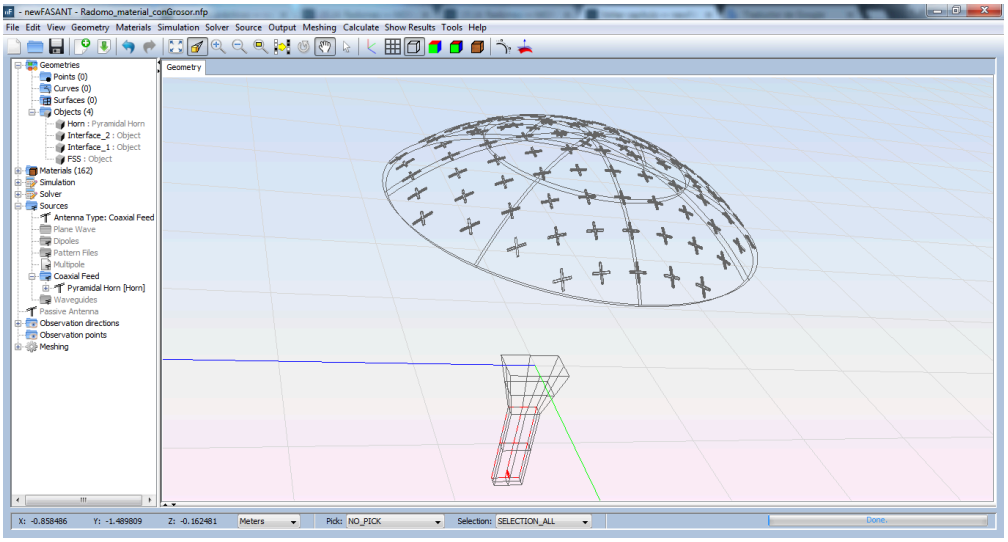


Figure 1722: 12 Geometry at this point

Configure **Radome Parameters**. To finish defining the radome follow the next steps:

1. Click on **Source -> Radome -> Define Radome** menu and add a radome.

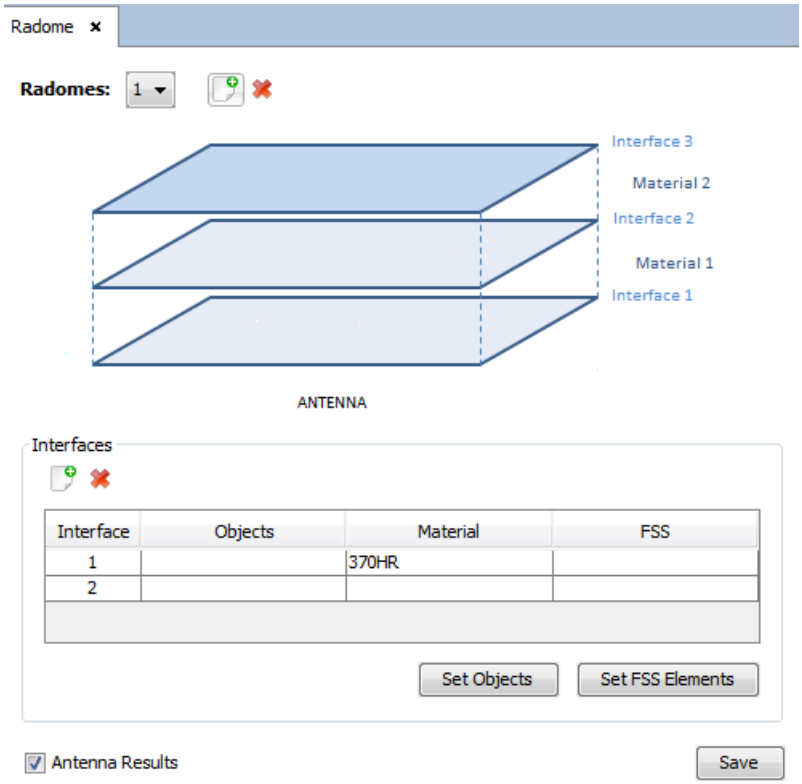


Figure 1723: 13 New Radome

2. Select the object of **Interface 1** (lower) and the first **row** in Radome panel and then click on **Set Objects** button. Repeat the process with the object of Interface 2.

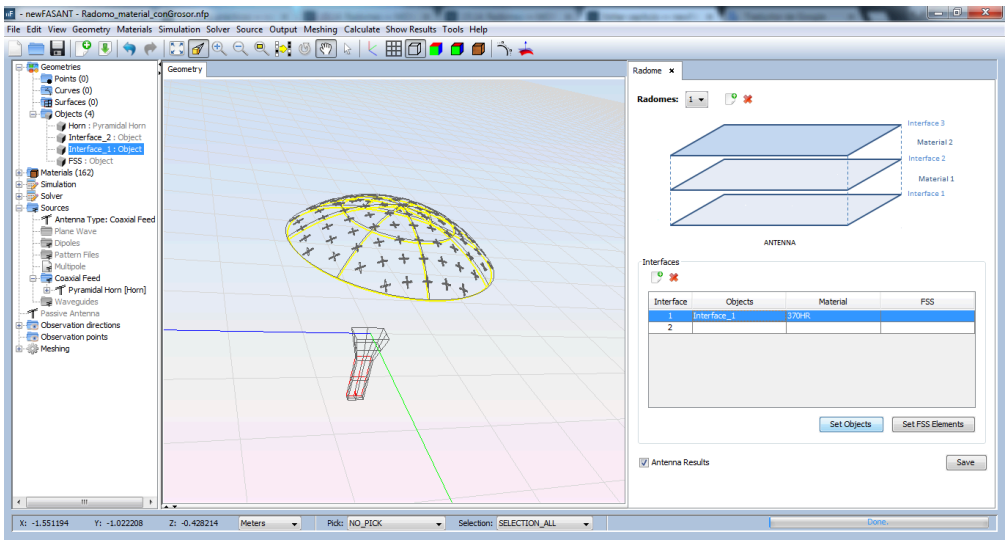


Figure 1724: 14 Add Interface Object

3. Selects the **FSS object** and the **second interface** in Radome Panel. Click on **Set FSS Elements** button.

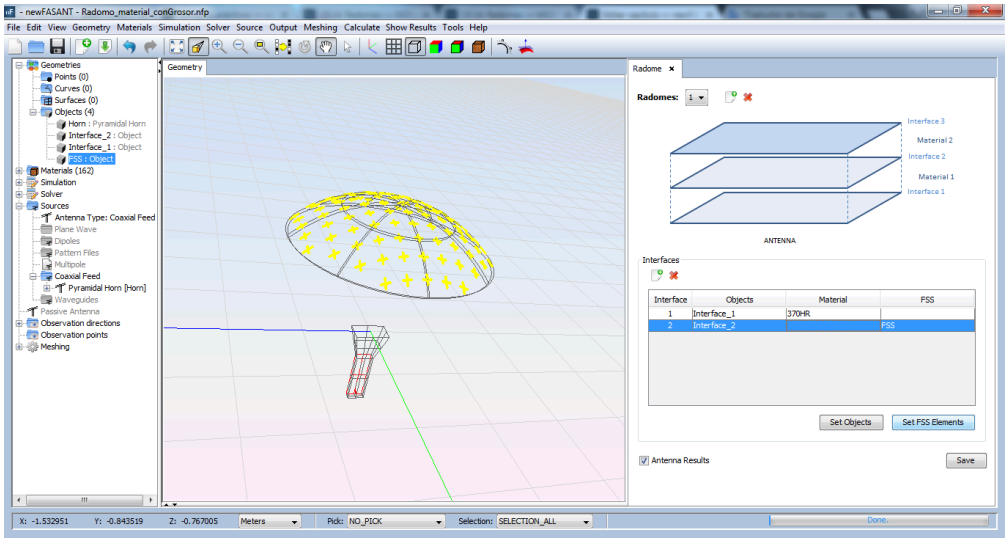


Figure 1725: 15 Set up a FSS in the Radome

4. Select the material **'Teflon'** on Interface 1. Click on **Save** button of Radome panel. Note The material of interface 1 corresponds to the material between the two interfaces.

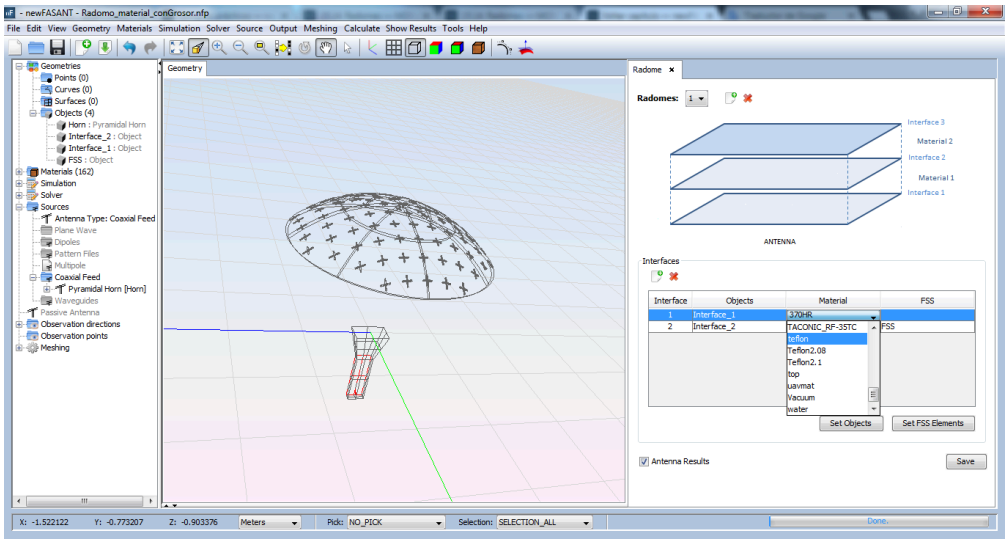


Figure 1726: 16 Select the material

Meshing Geometry. Click on **Meshing -> Create Mesh** and set planar and curved divisions to 20. Click on **Mesh** button.

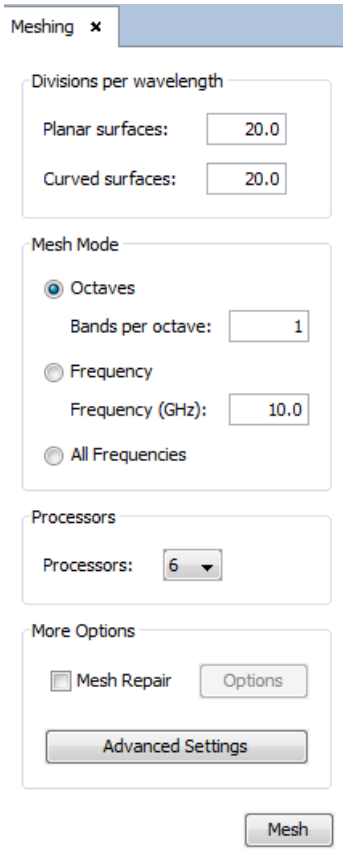


Figure 1727: 17 Meshing panel

Start Simulation. Click on **Calculate -> Execute**.

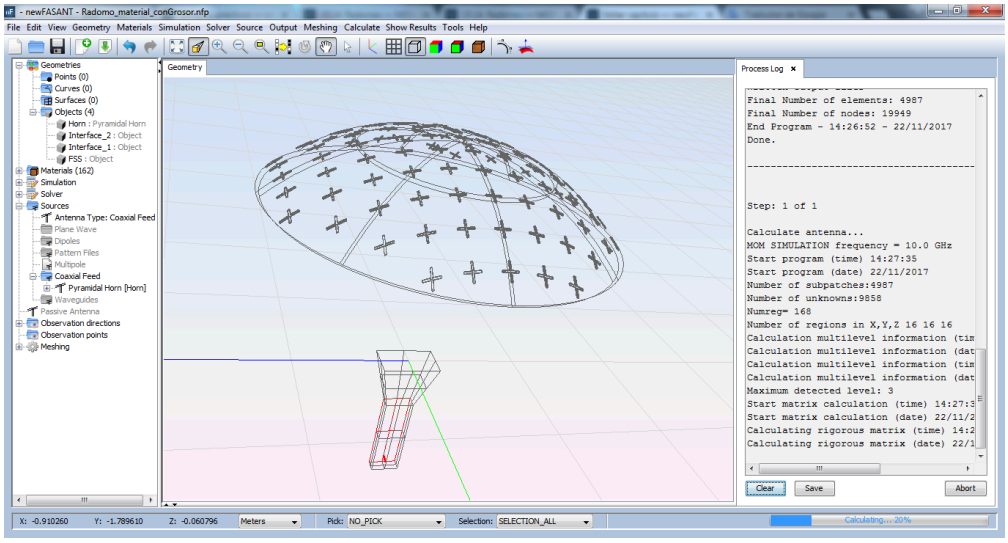


Figure 1728: 18 Simulation

Results

Radiation Pattern:

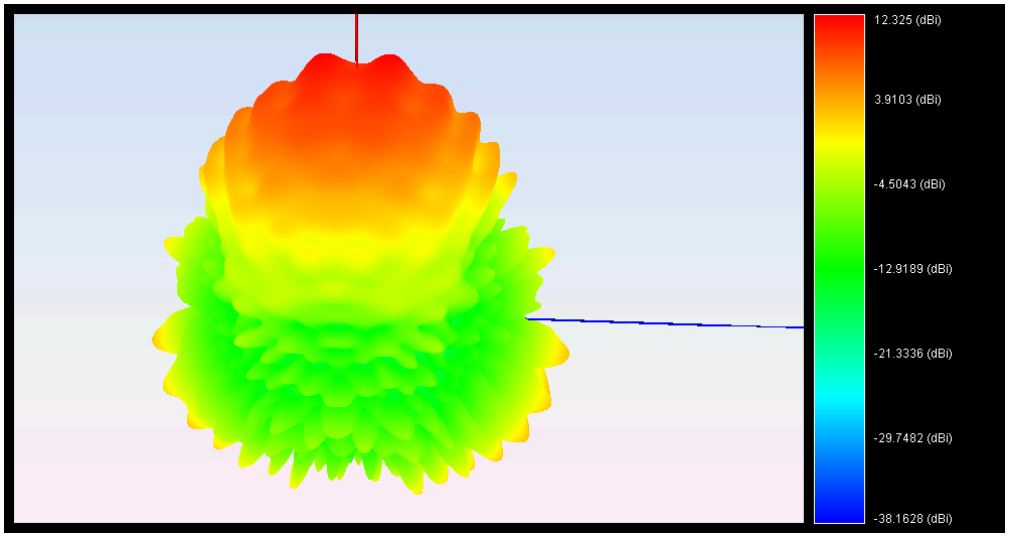
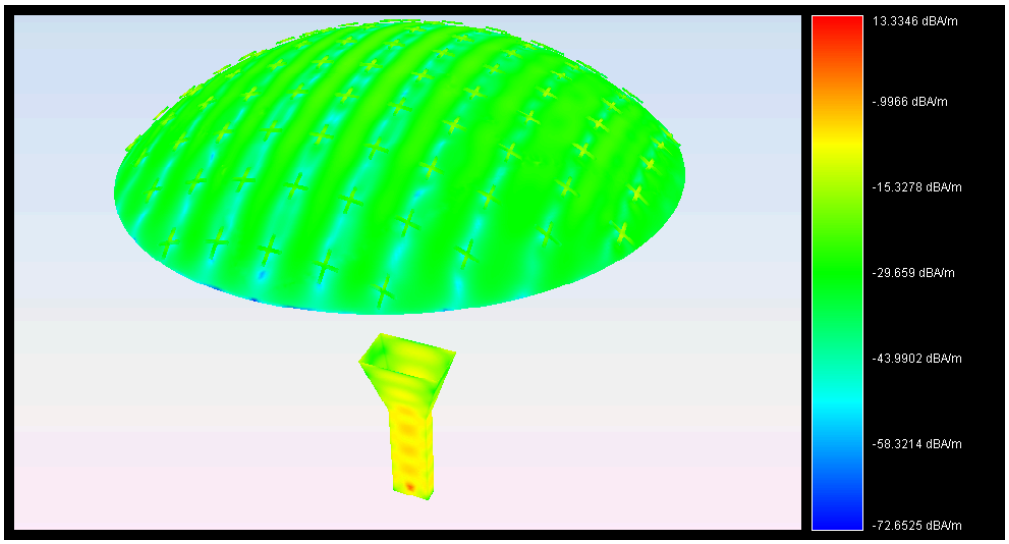


Figure 1729: Currents:



Example 3: Lightning Protection

This document shows an example to create a model of lightning protection for a radome:

1. Define two vertical segments close to the external radome surface.

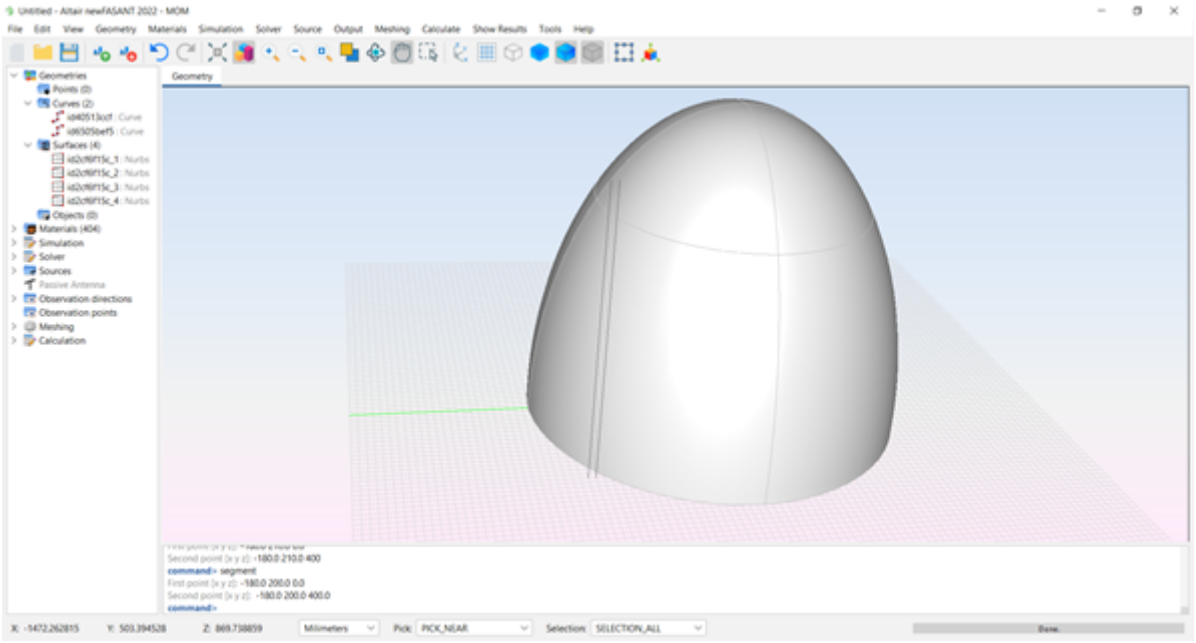


Figure 1730: Two vertical segments close to the external radome surface

2. Project one segment to the closest external surface.

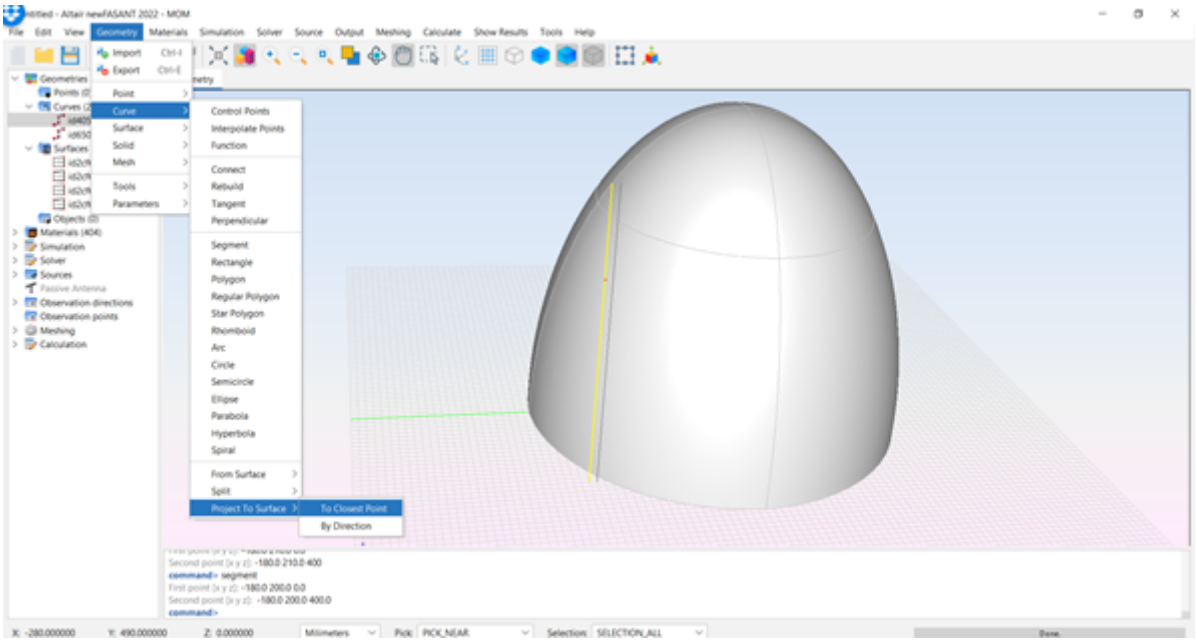


Figure 1731: Project one segment

3. Select the surface.

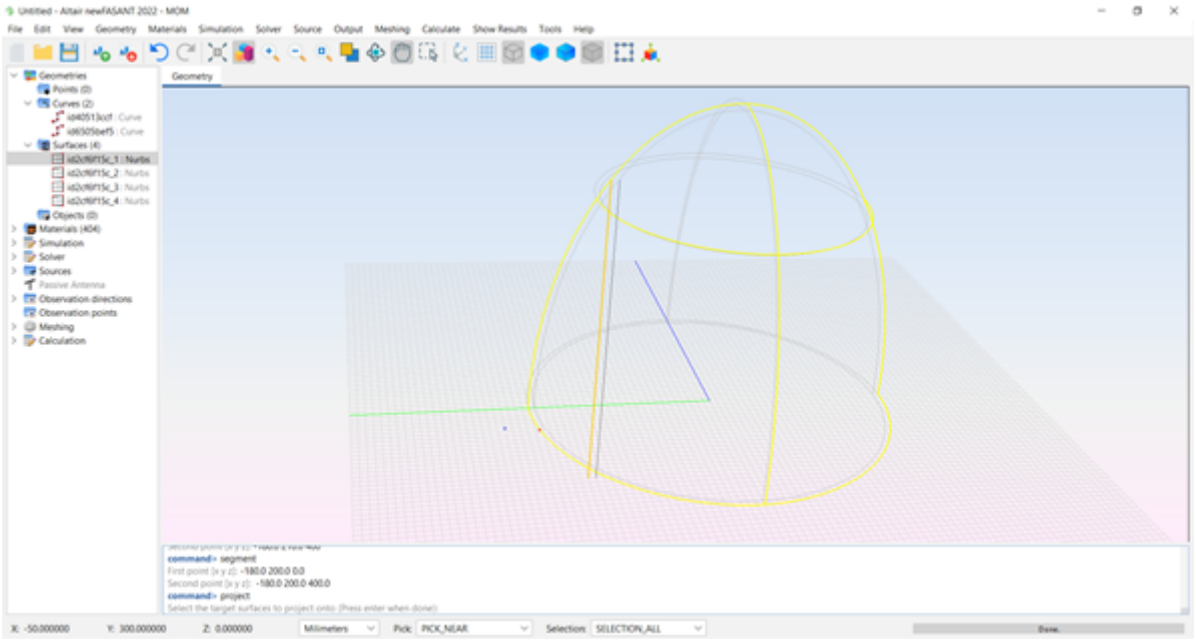


Figure 1732: Select the surface

4. Project the second segment on the surface.

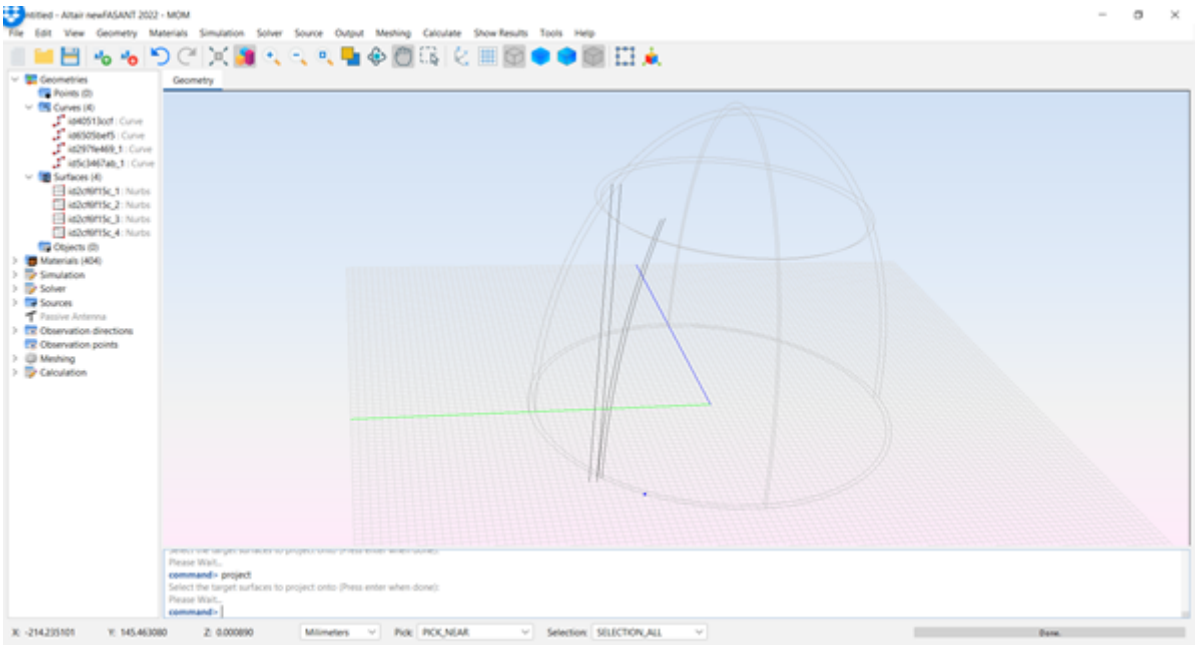


Figure 1733: Project the second segment

5. Create the surface of the lightning protection.

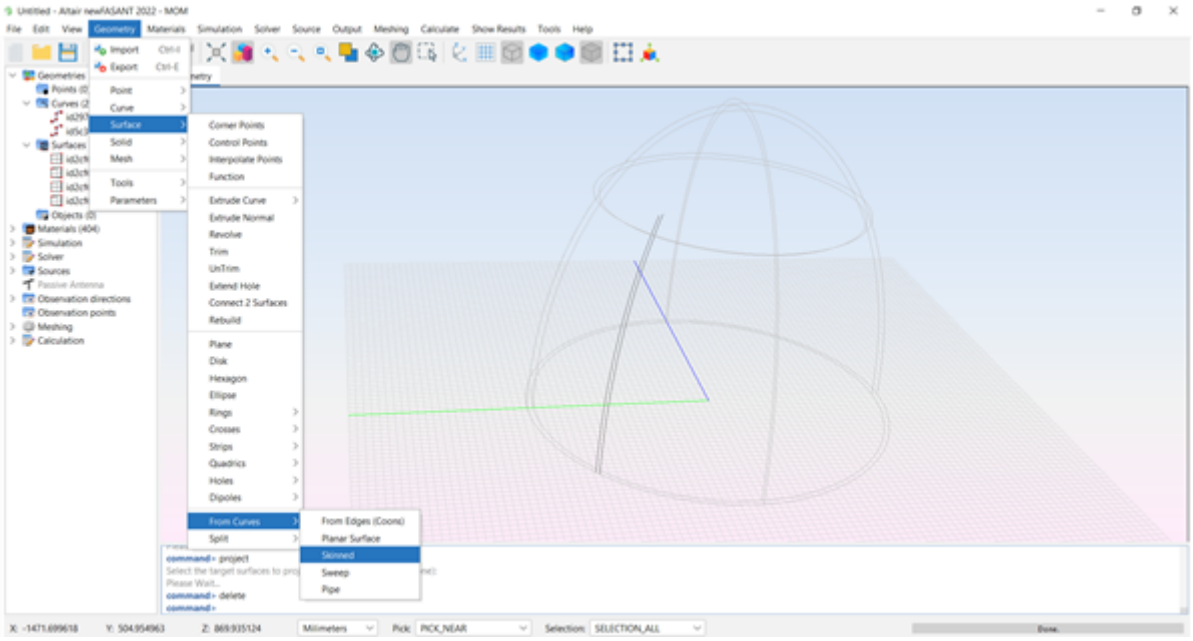


Figure 1734: Create the surface

6. Select the lines.

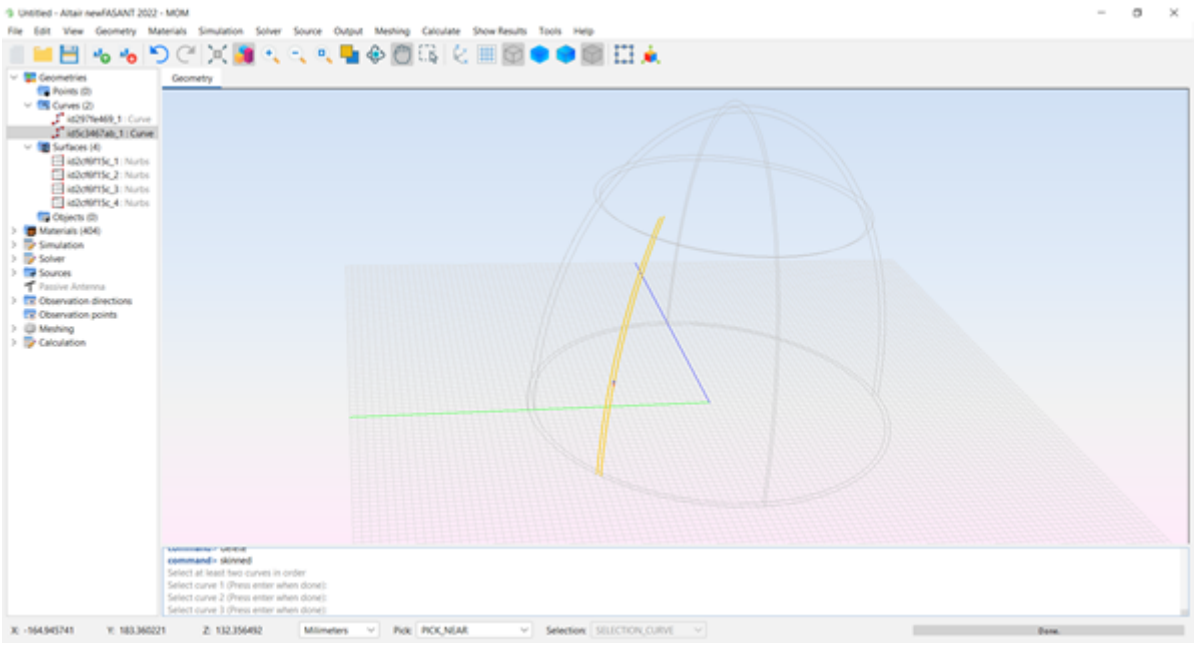


Figure 1735: Select the lines

7. Obtain the surface of the strip conformed to the surface radome.

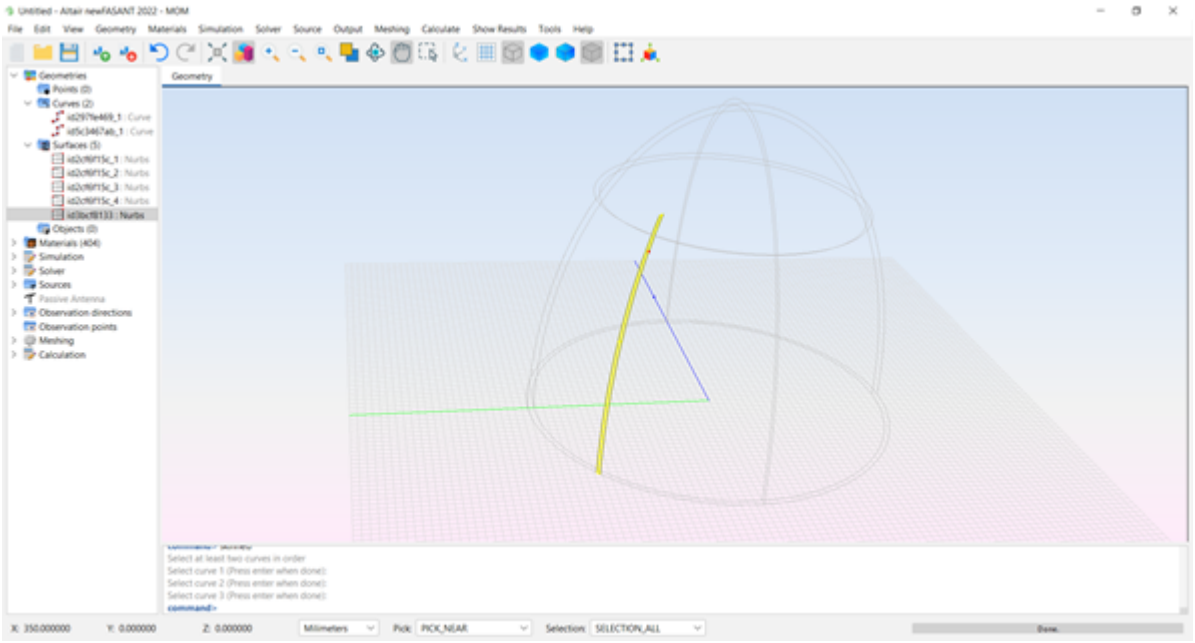


Figure 1736: Obtain the surface of the strip

8. Define a radome.

> **Note:** The lightning strip protection is external to the radome.

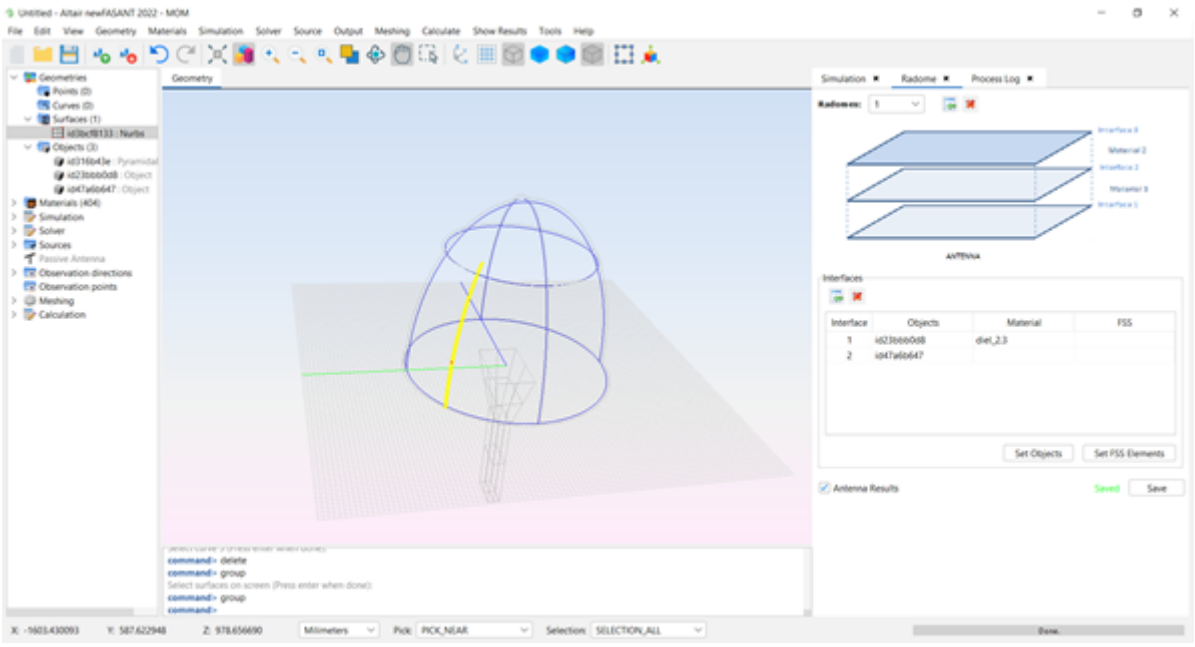


Figure 1737: Define a radome

9. Mesh and calculate.

10. View the radiation pattern of the radome plus strip.

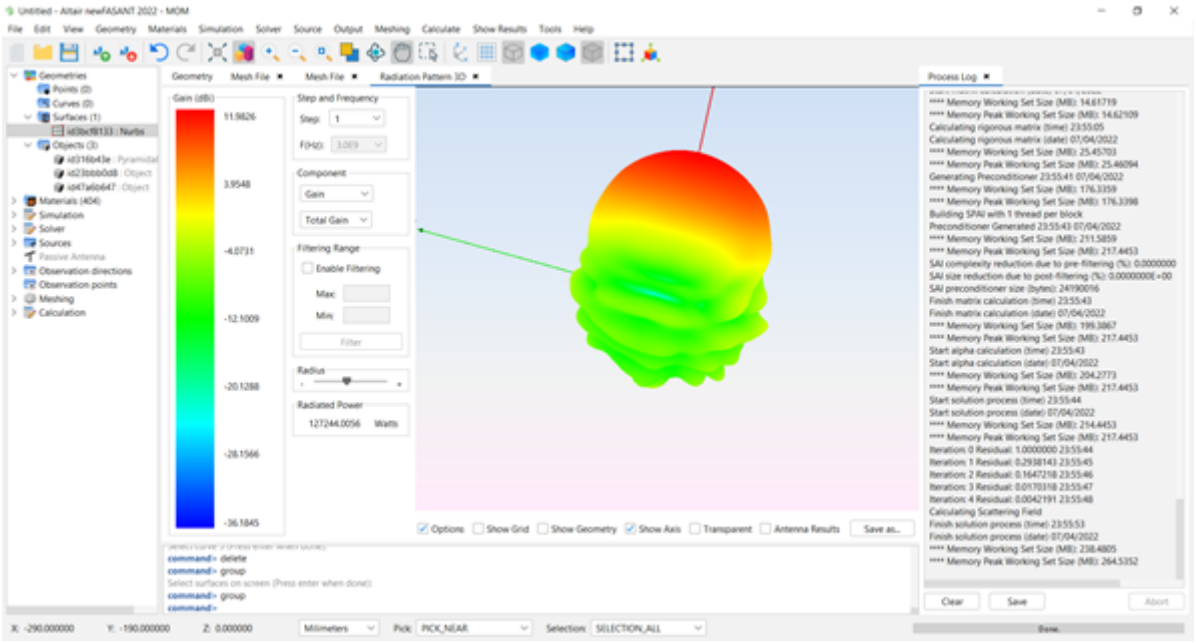


Figure 1738: Radiation pattern of the radome plus strip

11. Visualize the currents.

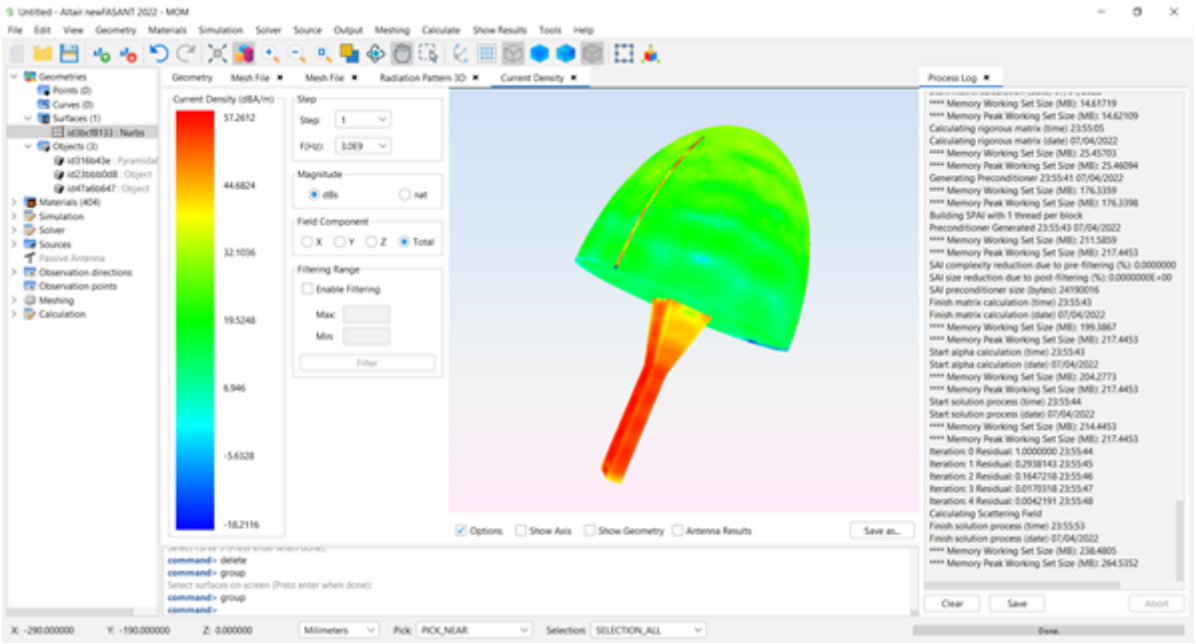


Figure 1739: Visualize the currents

11.16 Test Cases

In this section will be developed advanced cases of the use of the MOM module.

11.16.1 Test Case A

This example describesthe calculation ofthepoyting vector andthe maximum value of the field on an observationplaneofpointsnear the antenna.By calculating themultipolein this firstsimulation,thencalculatethe same dataonother observation, this timecylindrical,around yhe antenna and usingthe multipolefor a gainintime.

Step 1: Create a new MOM Project.

Open 'New Fasant' and select 'File --> New' option.

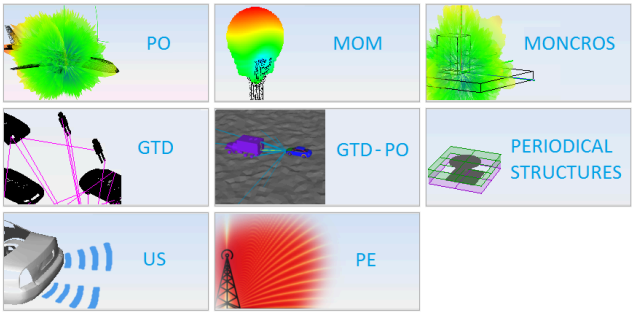


Figure 1740: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model.

Execute 'plane' command writing it on command line and sets the parameters as the next figure shows when command line ask for it.

```
command> plane
First corner of plane [x y z]: -0.5 -0.5 0
Plane size [width depth]: 1 1
command>
```

Figure 1741: 'Plane' Command parameters

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

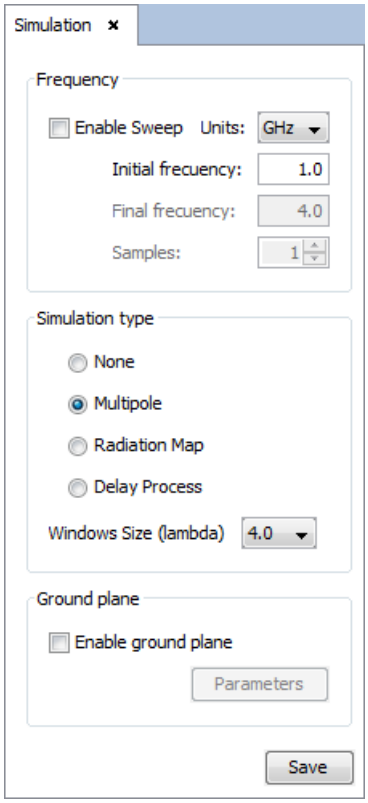


Figure 1742: Simulation Parameters panel

Step 4: Set the antenna parameters.

Select 'Antenna --> Primitives --> Coaxial Feed --> Dipole' option and set the parameters as show the next figure. Then save the parameters and the antenna appears.

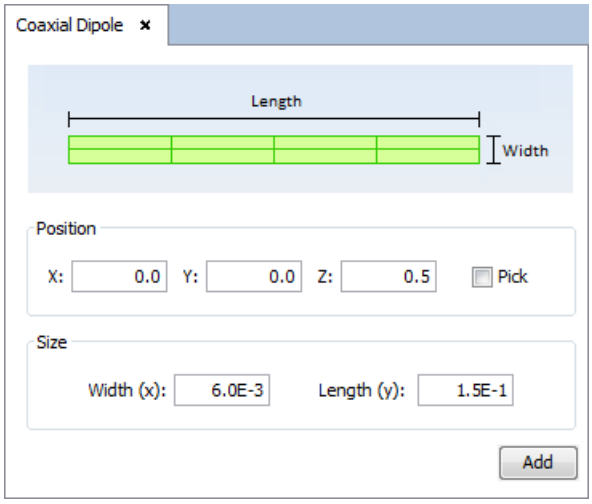


Figure 1743: Coaxial Dipole Antenna panel

Step 5: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

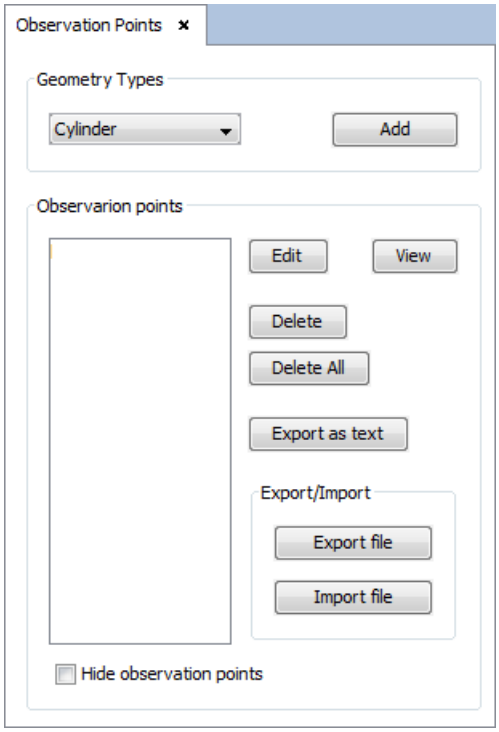


Figure 1744: Observation Points panel

To add a plane visualization, select 'plane' on the selector of 'Geometry Types' section and click on 'Add' button. The plane parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

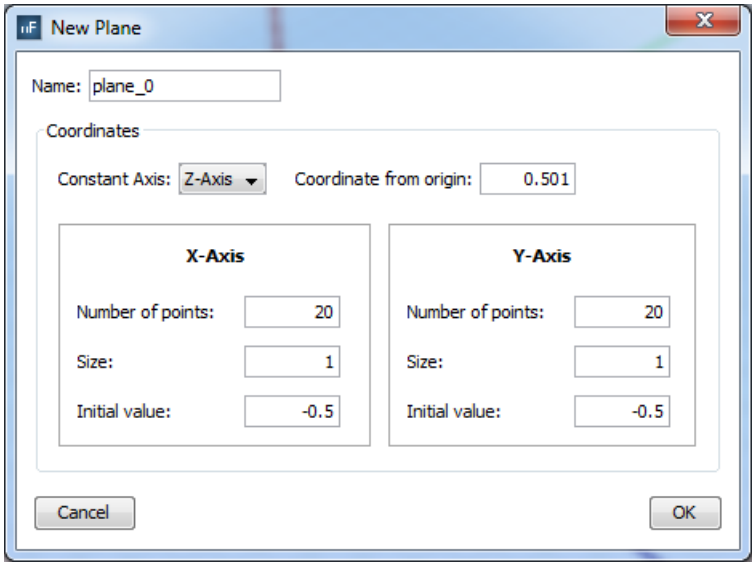


Figure 1745: Observation Plane panel

The observation will appear as a dashed line on the position configured.

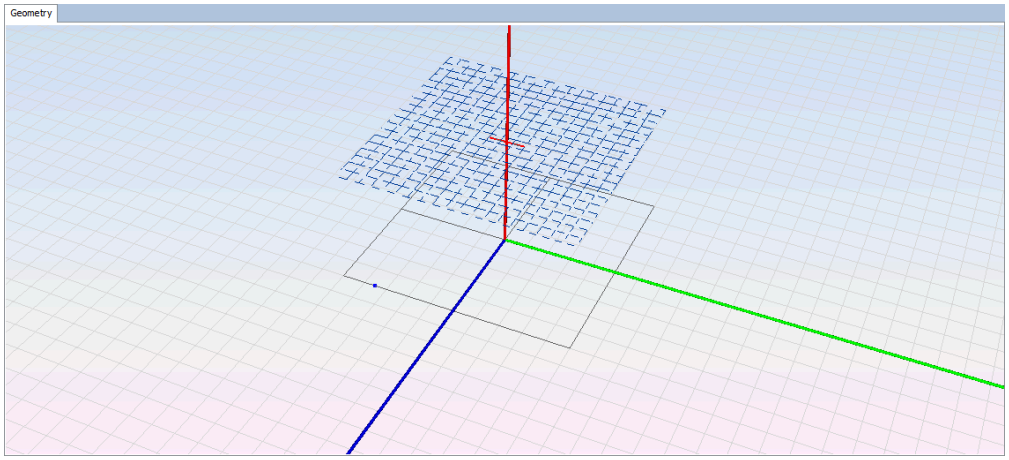


Figure 1746: Observation visualization

Step 6: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure.

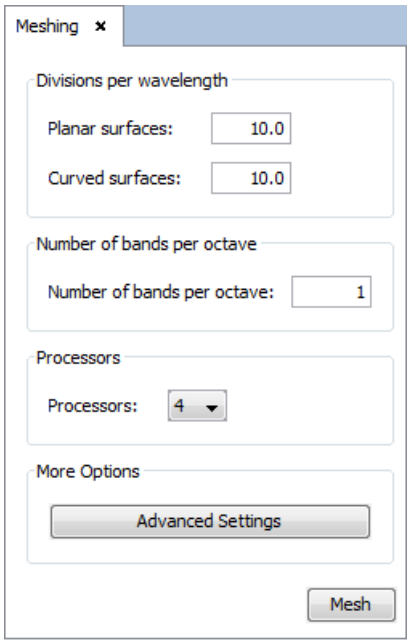


Figure 1747: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

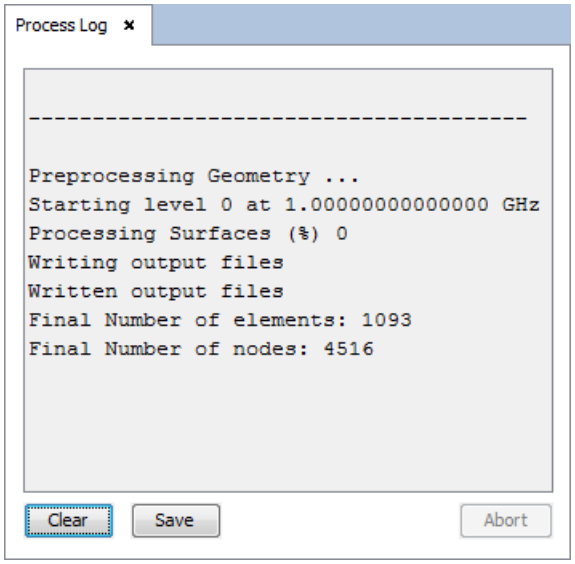


Figure 1748: Meshing process log

Step 7: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

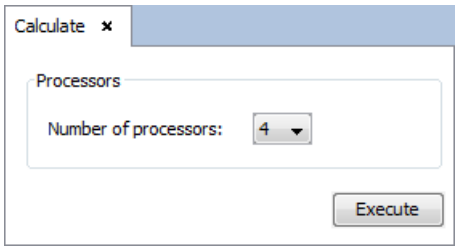


Figure 1749: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

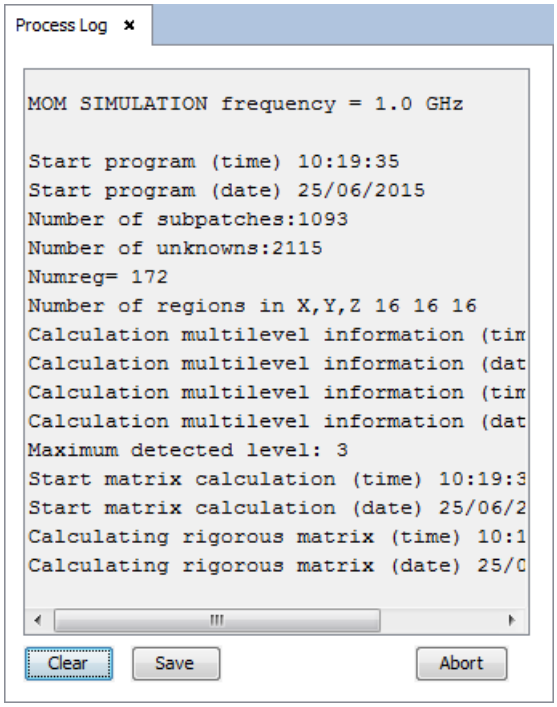


Figure 1750: Execute process log

Step 8: Show Results

Select 'Show Results --> Post-Process' option to calculate the puyting vector of the observation. To see the result as a diagram, select 'diagram' on the selector of 'New Trace' option and click on 'OK' button.

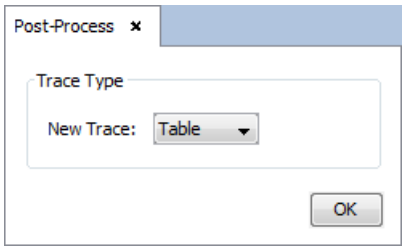


Figure 1751: Post-Process Trace panel

Then, a panel to configure the results appears. Set the parameters as next figure and add on 'OK' button.

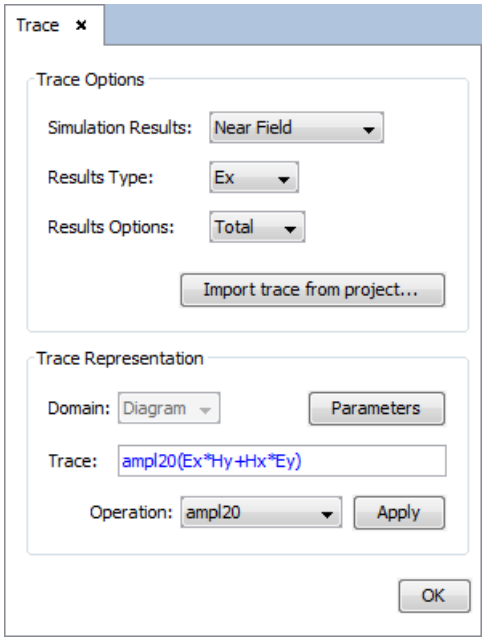


Figure 1752: Trace panel

The visualization of the results appears on the main panel.

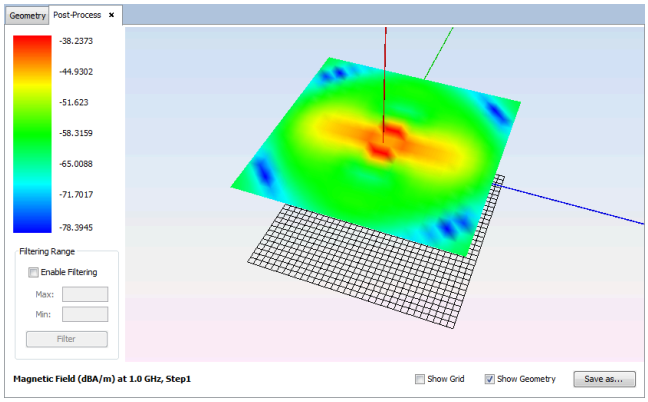


Figure 1753: Puyting Vector visualization

Select 'Show Results --> Post-Process' option to calculate the maximum field value of the plane observation. To see the result as a table, select 'table' on the selector of 'New Trace' option and click on 'OK' button.

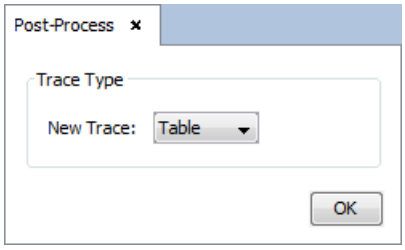


Figure 1754: Post-Process Trace panel

Then, a panel to configure the results appears. Set the parameters as next figure and add on 'OK' button.

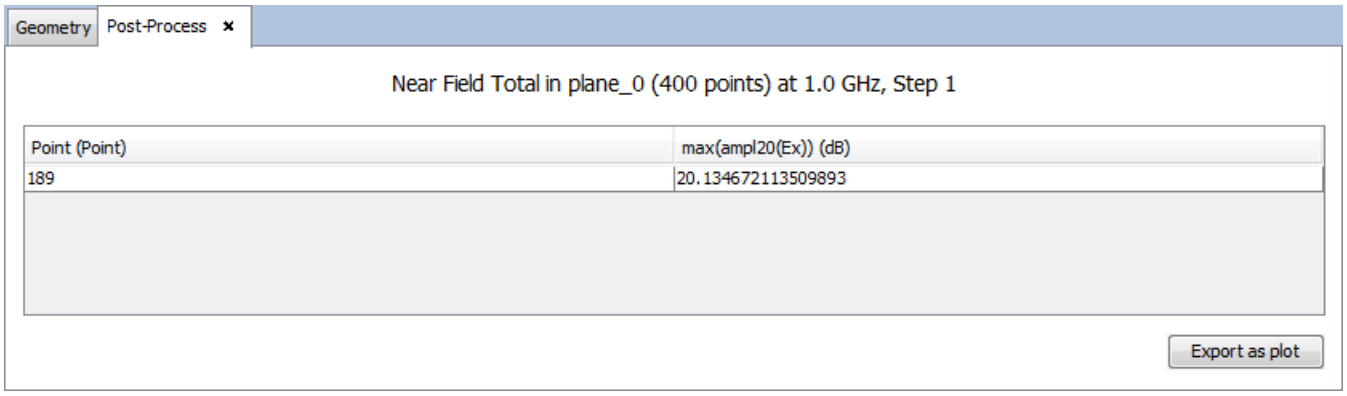


Figure 1755: Maximum Value visualization

Step 9: Save the multipole file to use it for calculating the second observation.

Select 'Show Results --> Export Multipole File' option to save the multipole file on a selected path for use on other simulations. Previous, select the step and frequency for the multipole file.

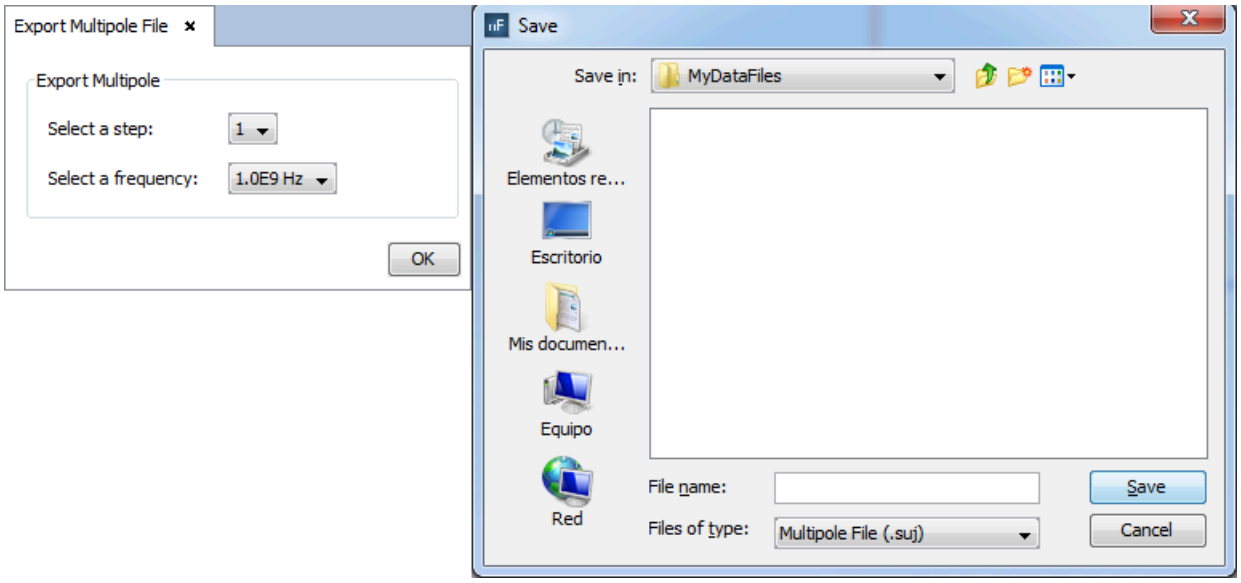


Figure 1756: Export Multipole

Step 10: Delete the geometry model, the antena and the observation plane. Then set the multipole file saved as antenna selecting 'Antenna --> Multipole Antenna' option and selecting the '.suj' file saved en the step 9. Then click on 'Save' button and the multipole appears as green cone.

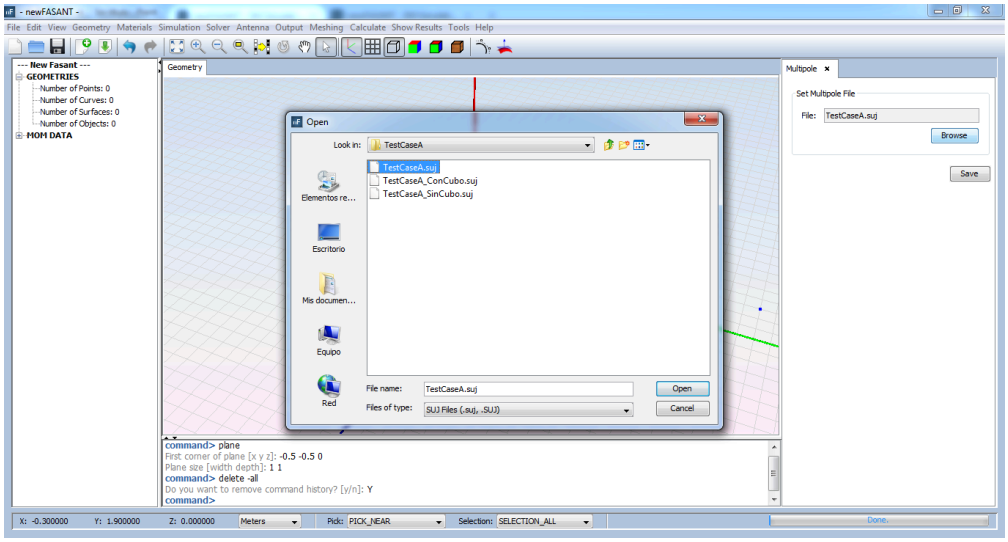


Figure 1757: Multipole Antenna panel

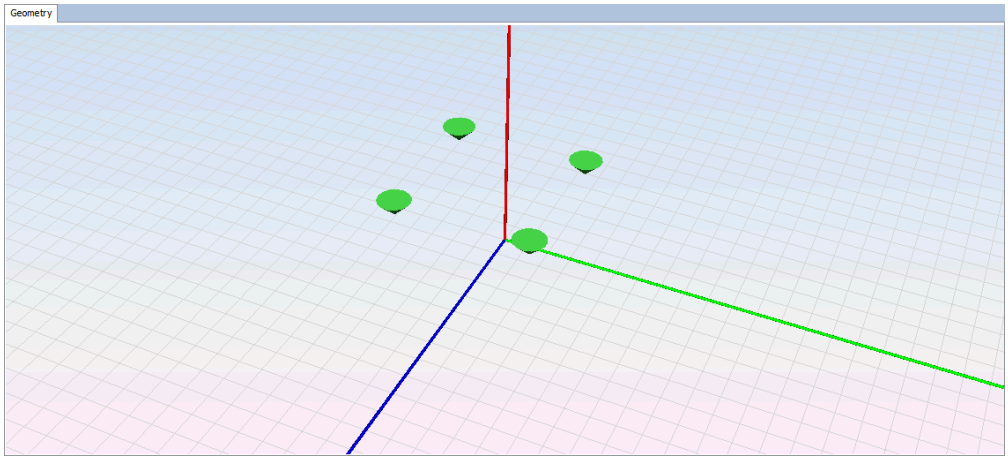


Figure 1758: Multipole Antenna visualization

Step 11: Set Near Field parameters.

Select 'Output --> Observation Points' option. To add a cylindrical visualization, select 'cylinder' on the selector of 'Geometry Types' section and click on 'Add' button. The cylinder parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

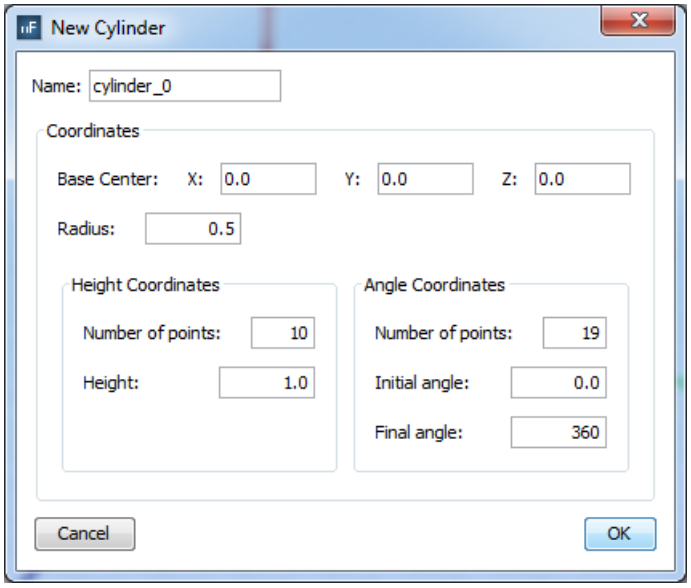


Figure 1759: Observation Cylinder panel

The observation will appear as a dashed line on the position configured.

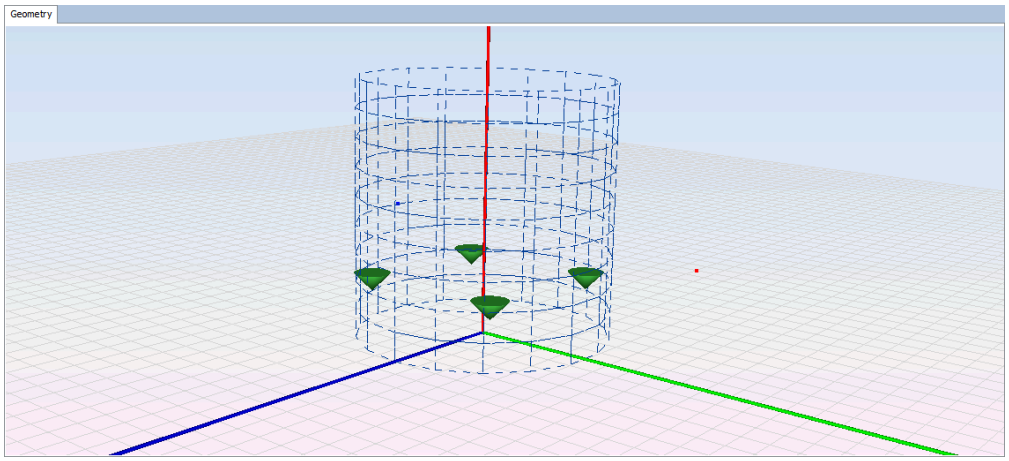


Figure 1760: Observation Cylinder visualization

Step 12: Calculate again (meshing is not necessary and not work without a geometry). This calculation uses the multipole antenna as the geometry and the coaxial dipole used in the first simulation.

Step 13: Show results again like step 8.

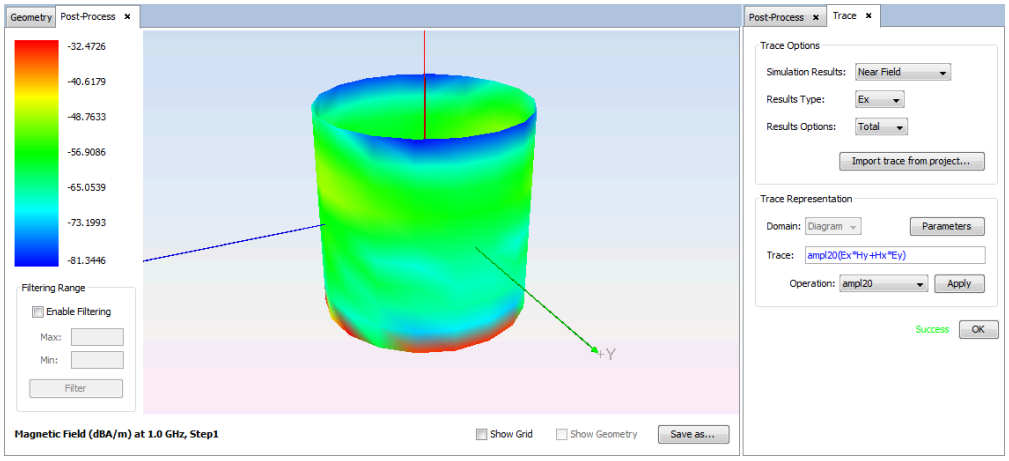


Figure 1761: Puyting Vector visualization

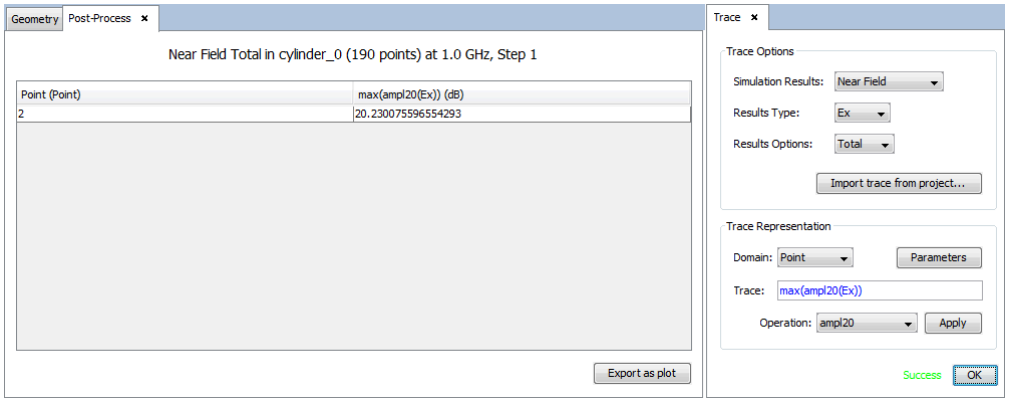


Figure 1762: Maximum Value visualization

11.16.2 Test Case B

This example describesthe calculation offield differencein aobservation planeofpointsbetween twoboxeswithoutandwith a slot, having adipoleinone of the boxes. The same processwill be performedon a lineofobservationinside the boxin which thedipoleis not found.

Step 1: Create a new MOM Project.

Open 'New Fasant' and select 'File --> New' option.

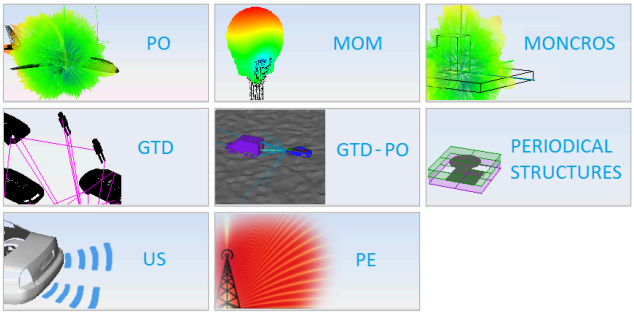


Figure 1763: New Project panel

Select 'MOM' option on the previous figure and start to configure the project.

Step 2: Create the geometry model.

Execute 'box' command writing it on command line and sets the parameters as the next figure shows when command line ask for it.

```
command> box
First corner of base [x y z]: -1.5 -0.5 0
Base size [width depth]: 1 1
Height [double]: 1
command>
```

Figure 1764: 'Box' Command parameters

To generate other box execute 'symmetric' command as the next figure shows.

```
command> symmetric -c
Select objects on screen (Press enter when done):
First point (for normal vector of symmetry plane) [x y z]: 0 0 0
Second point (for normal vector of symmetry plane) [x y z]: 1 0 0
command>
```

Figure 1765: 'Symmetric' command parameters

Step 3: Set Simulation Parameters

Select 'Simulation --> Parameters' option on the menu bar and the following panel appears. Set the parameters as the next figure shows and save it.

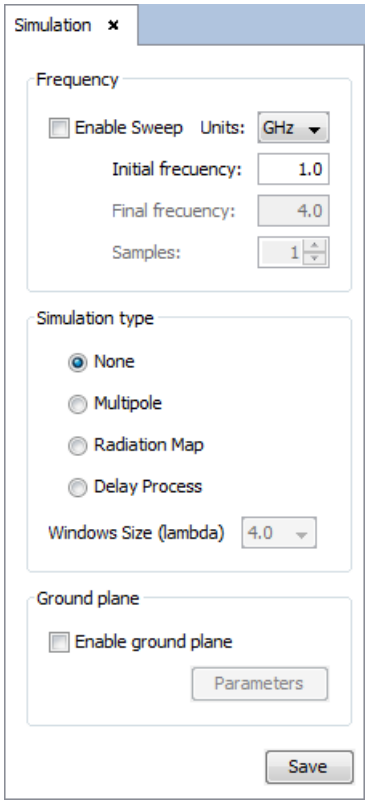


Figure 1766: Simulation Parameters panel

Step 4: Set the antenna parameters.

Select 'Antenna --> Dipole Antenna' option and set the parameters as show the next figure. Then save the parameters and the antenna appears.

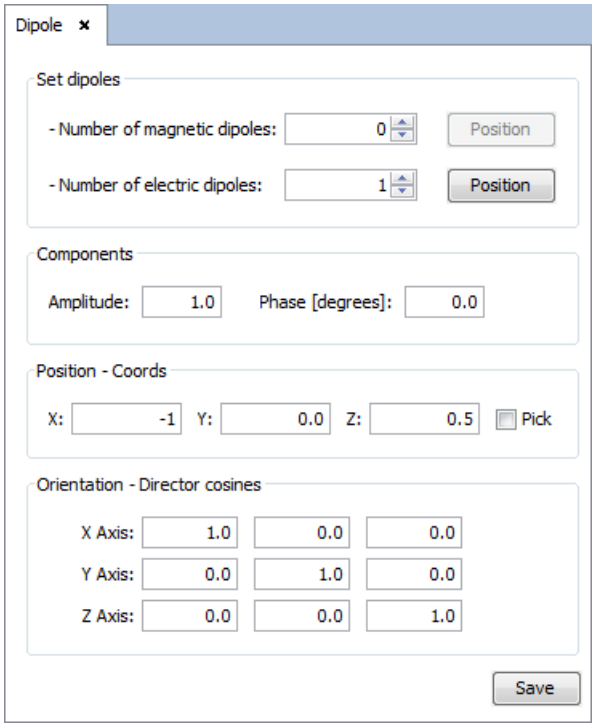


Figure 1767: Dipole Antenna panel

Step 5: Set Near Field parameters.

Select 'Output --> Observation Points' option. The following panel will appear.

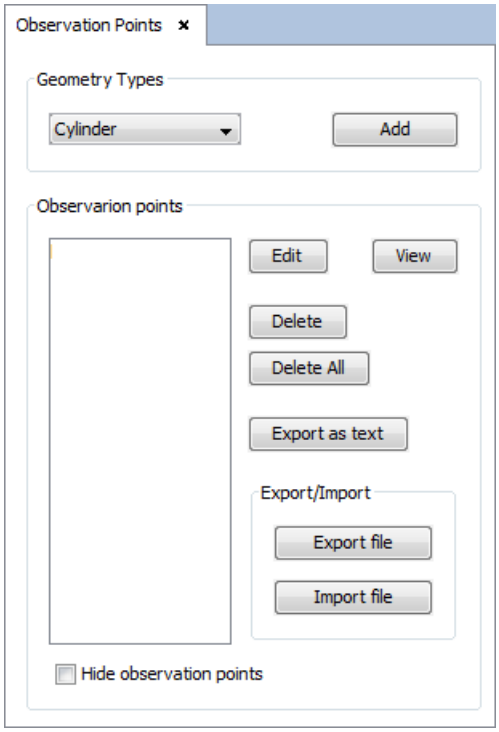


Figure 1768: Observation Points panel

To add a plane visualization, select 'plane' on the selector of 'Geometry Types' section and click on 'Add' button. The plane parameters panel will appear, then configure the values as the next figure show and accept it clicking on 'OK' button.

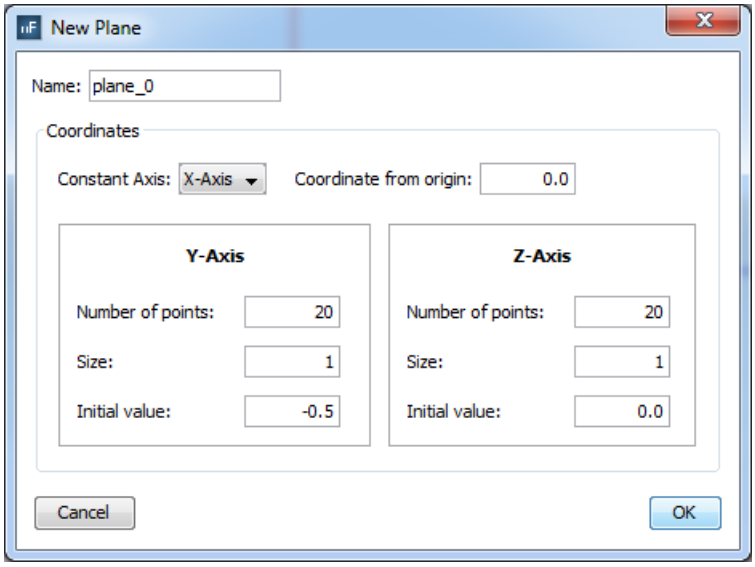


Figure 1769: Observation Plane panel

The observation will appear as a dashed line on the position configured.

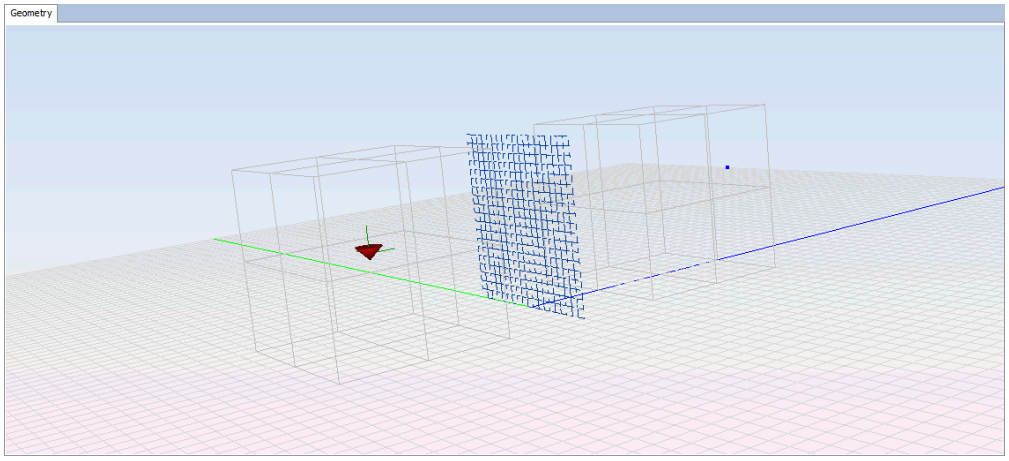


Figure 1770: Observation visualization

To add the line observation, repeat this step with the 'line' option on the selector of 'Geometry Types' and the following parameters.

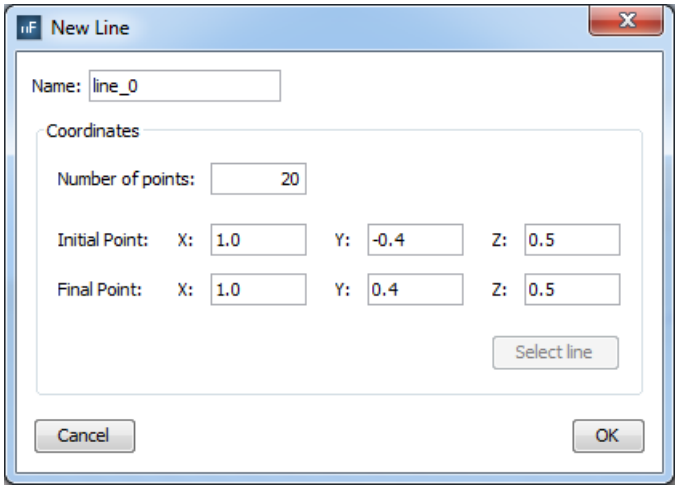


Figure 1771: Observation Line Panel

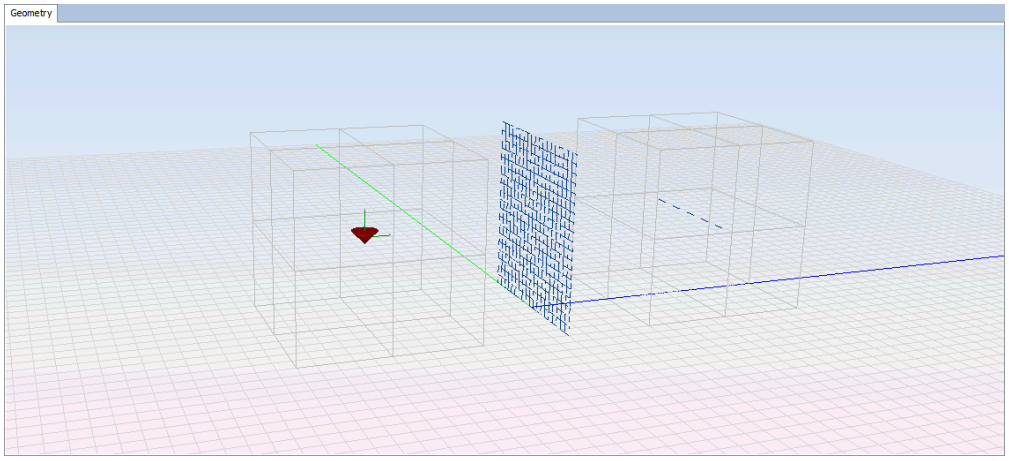


Figure 1772: Observation visualization

Step 6: Meshing the geometry model.

Select 'Meshing --> Parameters' to open the meshing configuration panel and then set the parameters as show the next figure.

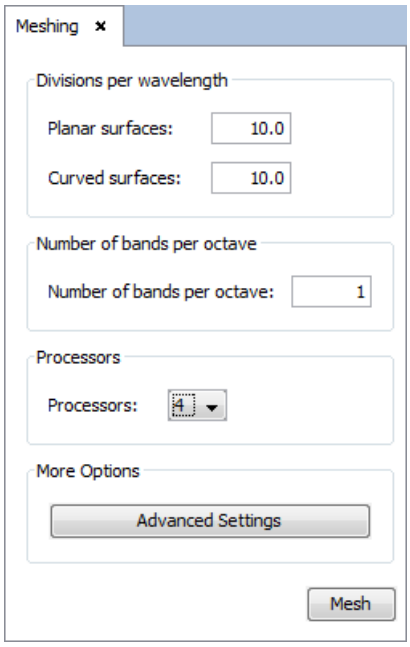


Figure 1773: Meshing panel

Then click on 'Mesh' button to starting the meshing. A panel appears to display meshing process information.

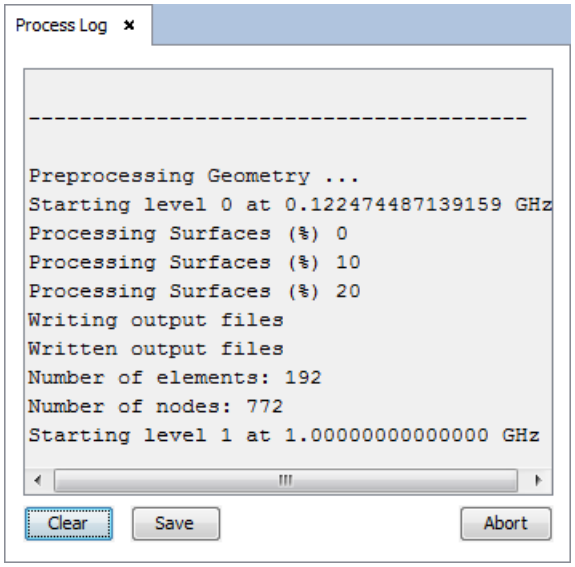


Figure 1774: Meshing process log

Step 7: Execute the simulation.

Select 'Calculate --> Execute' option to open simulation parameters. Then select the number of processors as the next figure show.

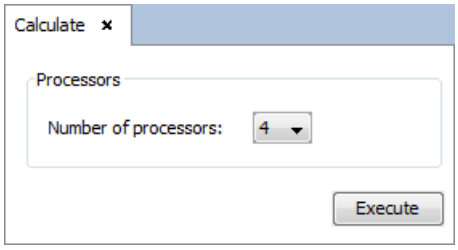


Figure 1775: Execute panel

Then click on 'Execute' button to starting the simulation. A panel appears to display execute process information.

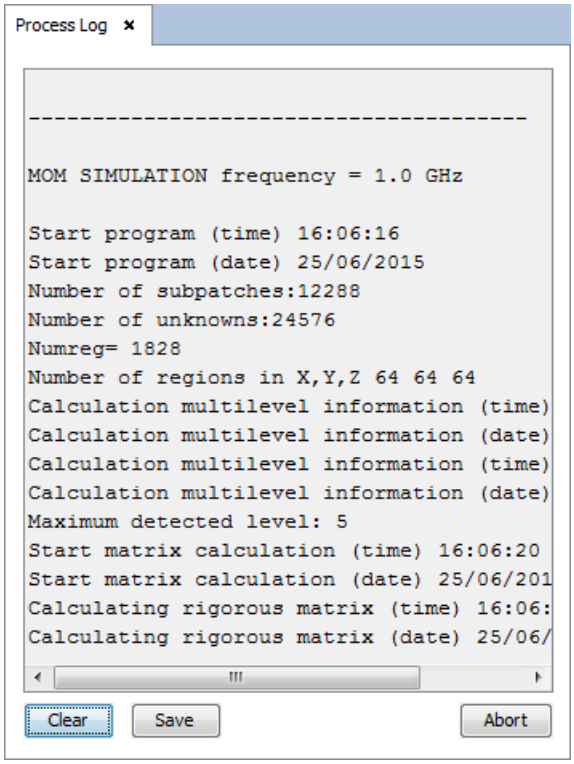


Figure 1776: Execute process log

Step 8: Save the project

Select 'File --> Save' option to save the project. Select a name and a path to save it on the file chooser that appears.

Step 9: Modify the geometry to make the slots in the boxes.

Execute 'explode' command with the boxes selected and follow the steps like shows the next figure. This command divide each object on independent surfaces.

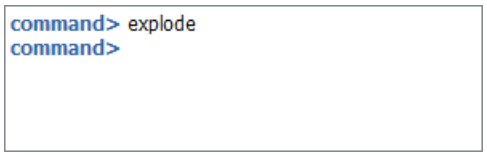


Figure 1777: 'Explode' command

Execute 'delete' command with the nearest surface to observation plane selected in the boxes.

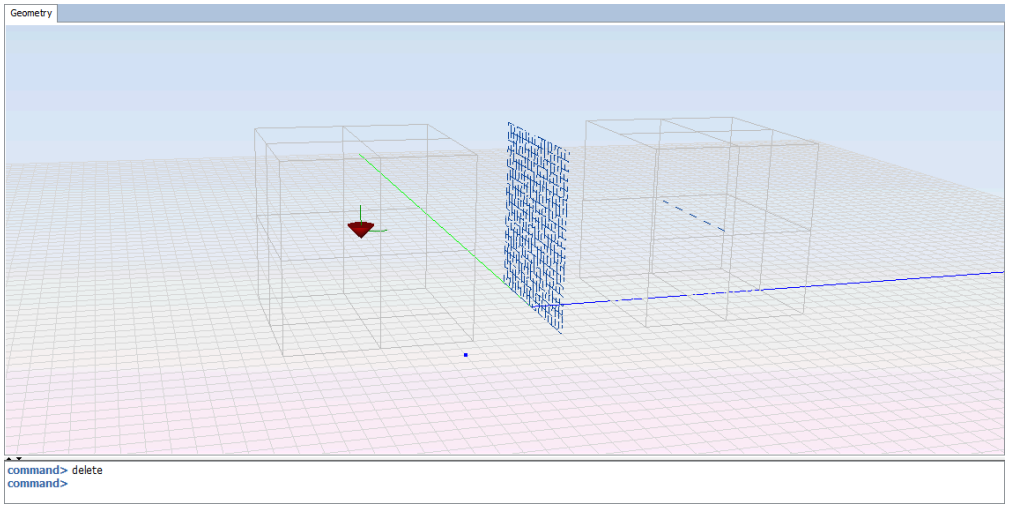


Figure 1778: 'Delete' command

Execute 'plane' command with the parameters shows on the next figure.

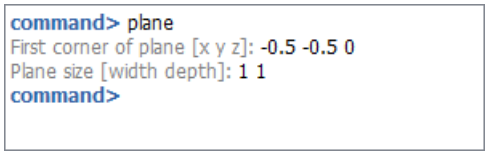


Figure 1779: 'Plane' command

Repeat the operation with the following parameters.

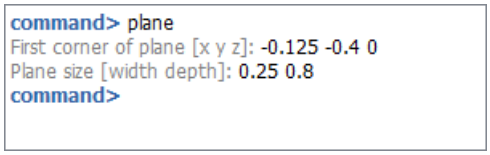


Figure 1780: 'Plane' command

Execute 'explode' command with the new planes selected.

Execute 'booleanDifference' command with the surfaces given for the previous 'explode'. This command ask for two surface to eliminate the second one from the first one.

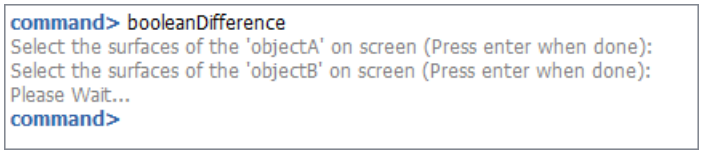


Figure 1781: 'booleanDifference' command

Execute 'rotate' command with the new surface previously selected, using the parameters shown on the next figure.

```
command> rotate
Start Point [x y z]: -0.5 -0.5 0.0
End Point [x y z]: -0.5 0.5 0.0
Angle [degrees]: -90
command>
```

Figure 1782: 'Rotate' command

Execute 'symmetric -c' command with the surface used on previous step. Use 'symmetric' parameters as the following figure shows.

```
command> symmetric -c
First point (for normal vector of symmetry plane) [x y z]: 0 0 0
Second point (for normal vector of symmetry plane) [x y z]: -1 0 0
command>
```

Figure 1783: 'Symmetric' command

Execute 'group' command with the surfaces composing one of the boxes previously selected. Then repeat the operation with the surfaces composing the other box.

The final result is a geometry with two boxes with a slot on the front surface to the observation plane.

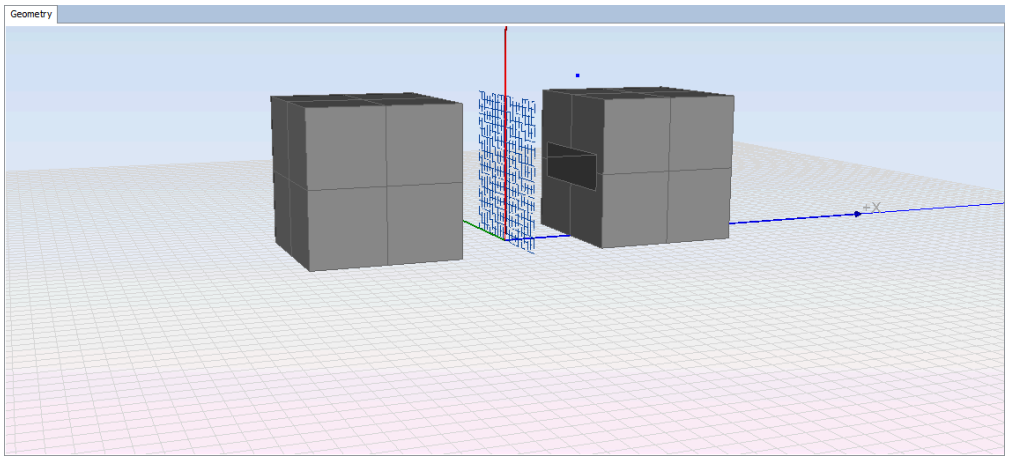


Figure 1784: Geometry visualization

Step 10: Meshing and calculate again like steps 6 and 7.

Step 11: Show results.

Select 'Show Results --> Post-Process' option to calculate the field difference on the plane observation between this example (slotted boxes) and the example executed on the first part of this guide (closed boxes). To see the result as a plot, select 'plot' on the selector of 'New Trace' option and click on 'OK' button.

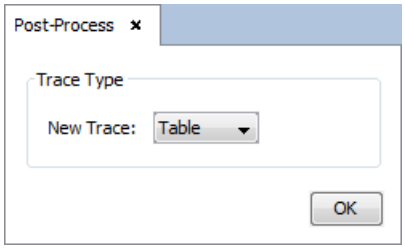


Figure 1785: Post-Process Trace panel

To import the result of the other project, first select the result to show on 'Simulation results', 'Results Type' and 'Results Options' combo boxes. In this case, the values of the next figure for 'Trace options' panel.

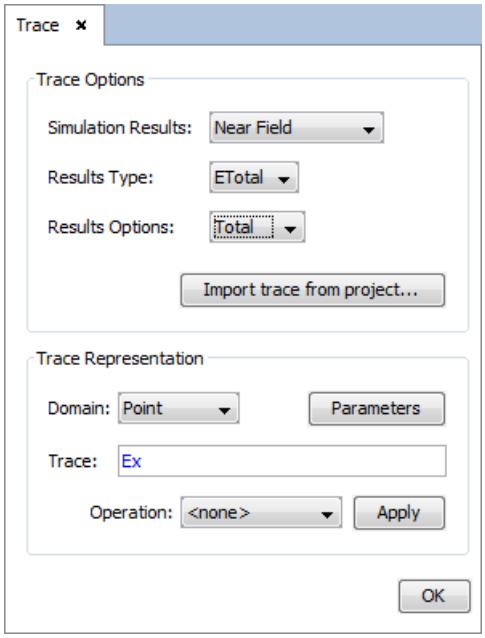


Figure 1786: Trace Options

Then click on 'Import trace form project' button to import the selected results to a project. In the panel that appears, select the project on the file chooser opened with 'Browse' button. Then click on 'Save' button to import the results. The imported results appears on 'Results Type' combo box as a new entry with the name of the selected to import with an index indicating the order of the imported projects.

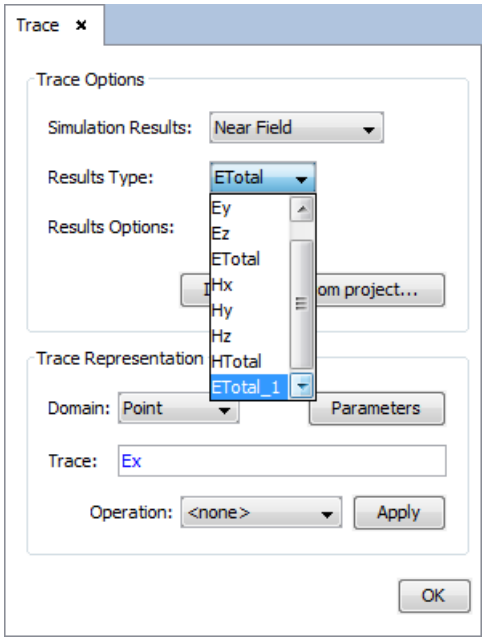


Figure 1787: Imported trace

Set the parameters as next figure and add on 'OK' button.

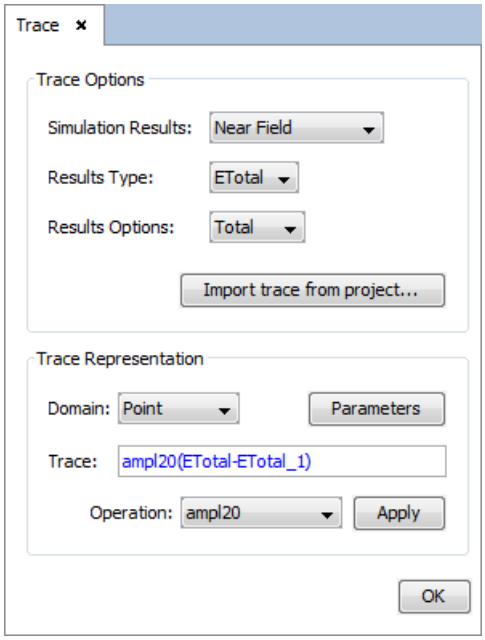


Figure 1788: Field Difference parameters

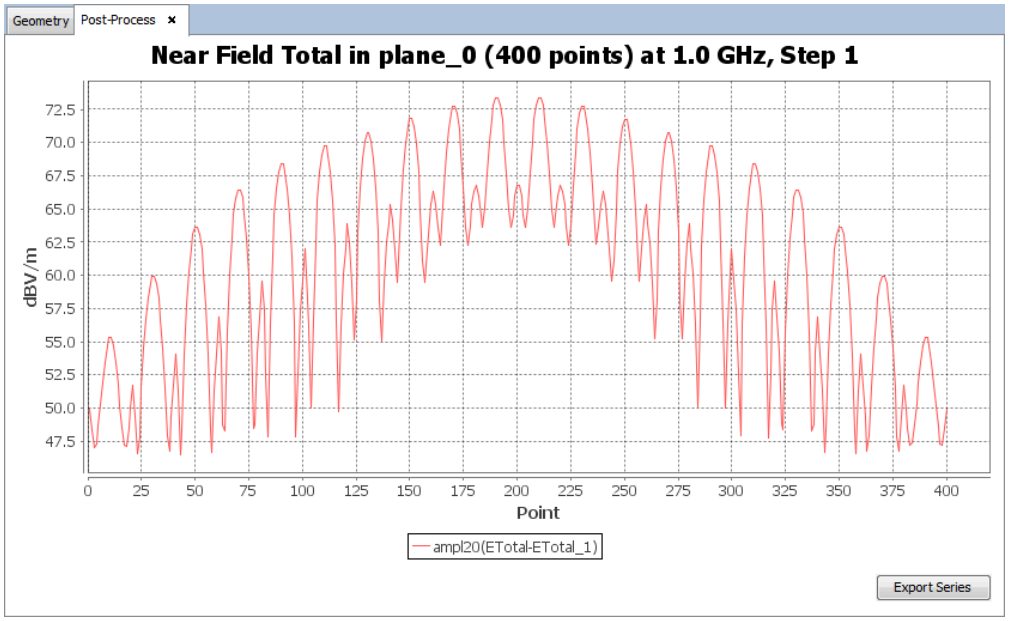


Figure 1789: Field Difference plot visualization on plane observation

To show the result as a diagram, repeat step 11 selecting 'diagram' on the 'Post-Process Trace' panel.

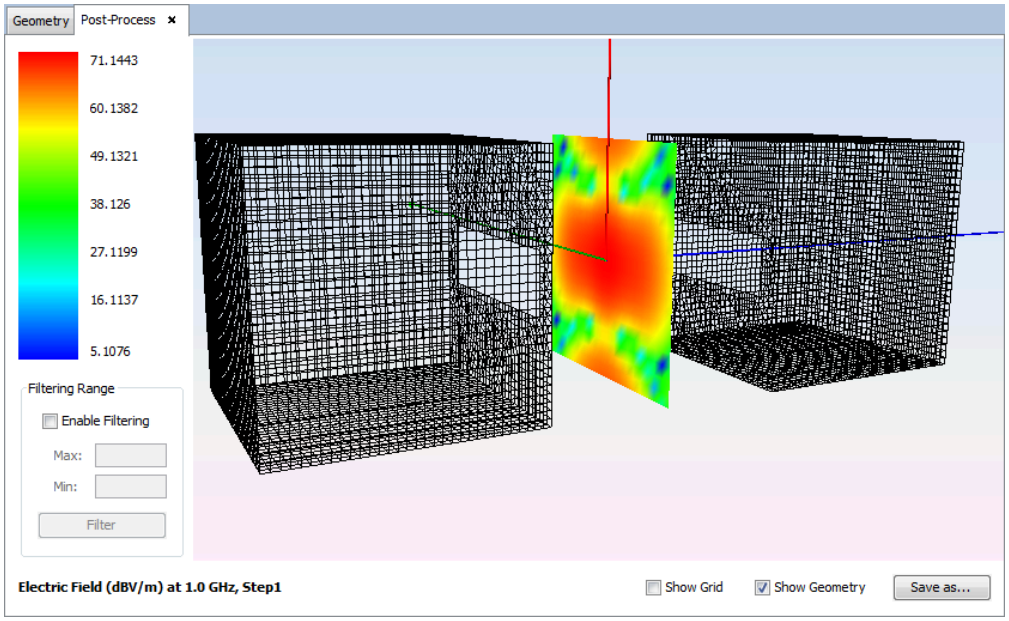


Figure 1790: Field Difference diagram visualization on plane observation

To show the same results over the line observation, follow the steps explained for the plane observation but previously to click on 'OK' button to show the results, click on 'Parameters' button on 'Trace Representation' panel. Then select the line observation on the panel that appears and confirm the changes.

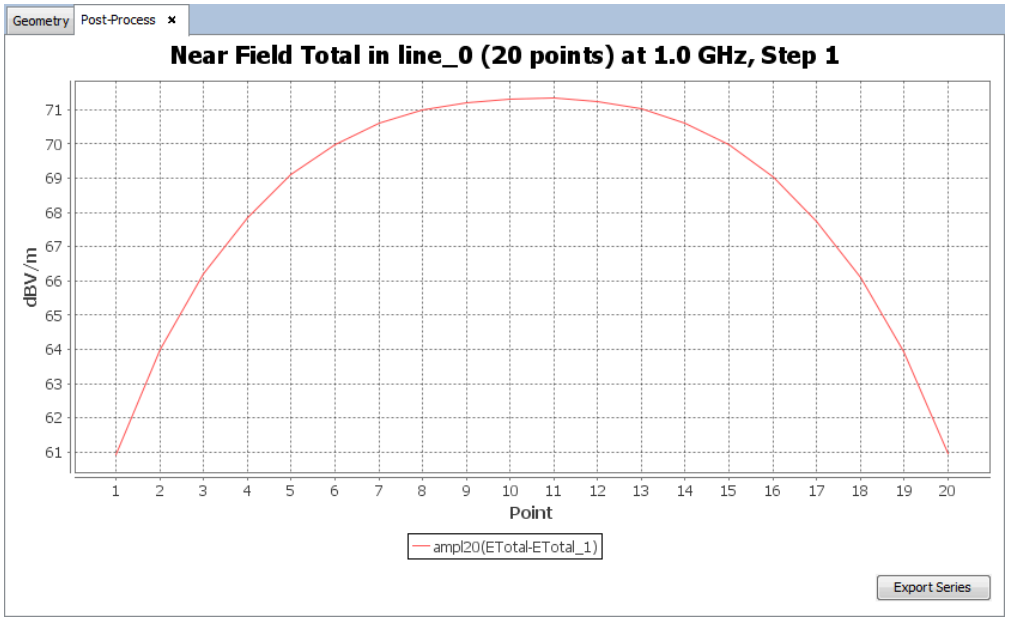


Figure 1791: Field Difference plot visualization on line observation

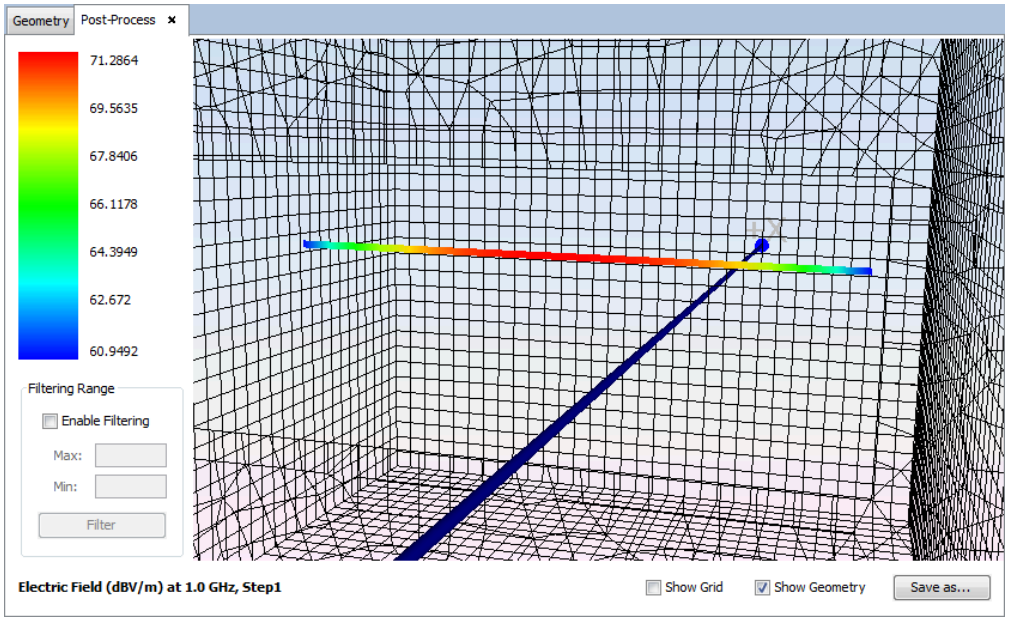


Figure 1792: Field Difference diagram visualization on line observation

Periodical Structures

12

Periodical structures is used for the analysis and design of reflection and transmission coefficients in periodical structures and metamaterials.

This chapter covers the following:

- 12.1 File Menu (p. 1465)
- 12.2 Edit Menu (p. 1466)
- 12.3 View Menu (p. 1467)
- 12.4 Geometry Menu (p. 1468)
- 12.5 Materials Menu (p. 1469)
- 12.6 Cell Menu (p. 1470)
- 12.7 Simulation Menu (p. 1472)
- 12.8 Solver Menu (p. 1473)
- 12.9 Meshing Menu (p. 1475)
- 12.10 Calculate Menu (Alt + C) (p. 1482)
- 12.11 Show Results Menu (p. 1484)
- 12.12 Tools Menu (p. 1488)
- 12.13 Help Menu (p. 1489)
- 12.14 Training Examples (p. 1490)
- 12.15 Annex 1: Creating a Reflectarray Database (p. 1501)
- 12.16 Annex 2: Analysis of a Reflectarray Database Creation (p. 1505)

12.1 File Menu

For information about the File menu, refer to File Menu.

12.2 Edit Menu

For information about the File menu, refer to File Menu.

12.3 View Menu

For information about the View menu, refer to View Menu.

12.4 Geometry Menu

In order to generate the geometry for the cell is recommended to pay attention the following considerations:

- For the cell construction the user geometries must be planar surfaces, located on XY planes (parallel to the floor) and centered over the 'Z' axis.
- If the user wants to locate a ground plane, a point where the ground plane will be located will be added. This point can be located in any order of the 'Z' axis but it will be considered as the bottom interface, i.e. the other interfaces must be located in upper orders of the 'Z' axis of this point.
- More points can be defined to generate empty interfaces but the reference plane only will be introduced on the bottom interface.

For information about the Geometry menu and the geometry primitives to add, refer to Geometry Menu.

12.5 Materials Menu

For information about the Materials menu, refer Materials Menu (Periodical Structures Module).

12.6 Cell Menu

The main options for the cell definition are included in the Cell menu.

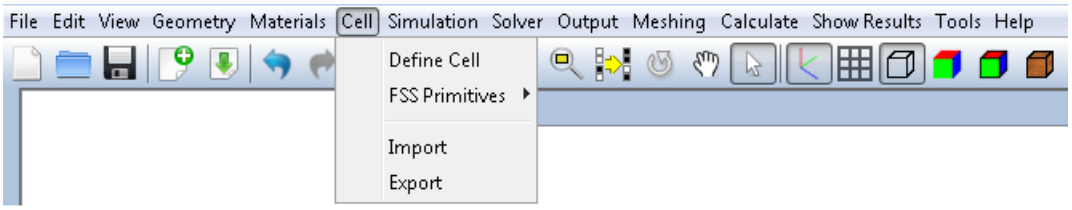


Figure 1793: Cell Menu

12.6.1 Define Cell

When this panel is opened, the cell is automatically created according to the geometry previously generated, see Geometry Menu.

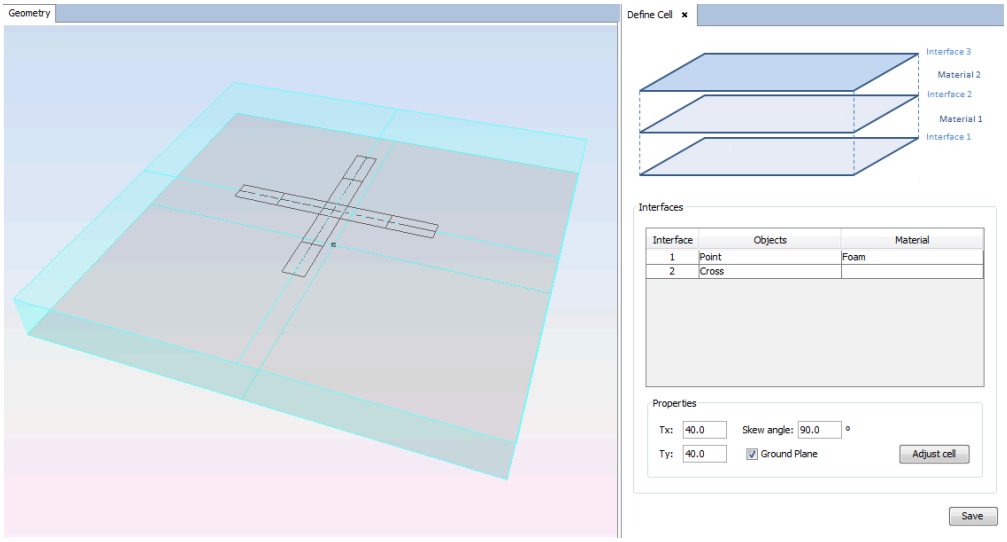


Figure 1794: Define Cell panel

This panel has the following considerations:

- An interface is generated for every different Z coordinate that is detected on the geometry. In the image, the cell has two interfaces because a point and the plane are defined on two different XY planes with different Z order.
- According to the scheme on the top of the panel, the interfaces and materials are numerated from the lowest Z coordinate to the highest one. In the image 'Interface 1' correspond with the interface where the point is defined and 'Material 1' that is 'Foam' corresponds with the blue layer. On the top, 'Interface 3' corresponds with the interface where the plane is defined.
- The material layers, that defines the material located over the interface of the same number, are only represented while this panel is opened.

Parameters:

- In the table the user can assign defined materials to the layers, except to the upper interface that is the top of the cell. To add a material to de materials database for use it for a layer see Materials Menu (Periodical Structures Module).
- **Cell dimension:** Tx and Ty determines the x-size and the y-size of the cell.
- **Skew angle:** Define the angle of displacement for the y-axis to the replication cells. Normally, a skew angle of 45 to 90 degrees is defined. The upper cells will move to the right on the y-axis.
- **Ground Plane:** The check box define a reflection (enabled) or transmission (disabled) cell. The ground plane is added automatically on the bottom interface, where a point must be presented or the present geometry will be removed. The ground plane will be painted on gray to visualize it when this panel is displayed.
- **Adjust Cell:** The button resize the cell to the bounding box geometry.

12.6.2 FSS Primitives

In this menu, the most common periodic primitives are defined. For information about the periodic primitives, refer to the Geometry menu in the Geometry Menu.

12.6.3 Import

Imports a previously saved cell, selecting the cell file on the file chooser that appears when this option is selected.

12.6.4 Export

Export to a .cell file the cell defined into the interface, selecting the path on a file chooser that appears when this option is selected.

12.7 Simulation Menu

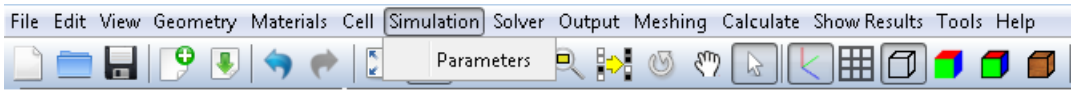


Figure 1795: Simulation Menu

12.7.1 Simulation Parameters

The parameters are defined by the user to the right of the window. It is possible to select a single frequency or a frequency sweep. The user can define the initial frequency, final frequency and the number of samples for the simulation.

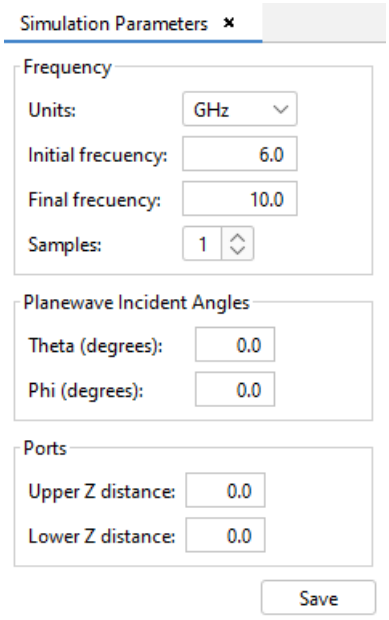


Figure 1796: Simulation Parameters panel

- **Frequency:** The user may select a single frequency or a frequency sweep to run.
- **Planewave Incident Angles**: Define in which angle the plane wave will incide.
- **Ports**: In this panel the user can specify some additional distance from the top and bottom of the structure. The specified distance will be added into the structure, into the Z-Axis.

After setting up the simulation parameters it is necessary to press the Save button to save the configuration changes.

12.8 Solver Menu

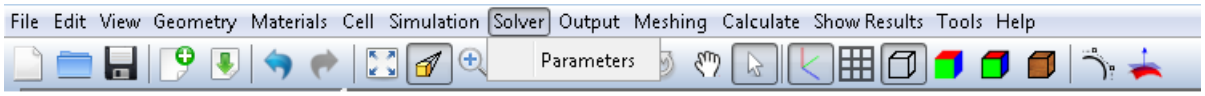


Figure 1797: Solver Menu

12.8.1 Solver Parameters

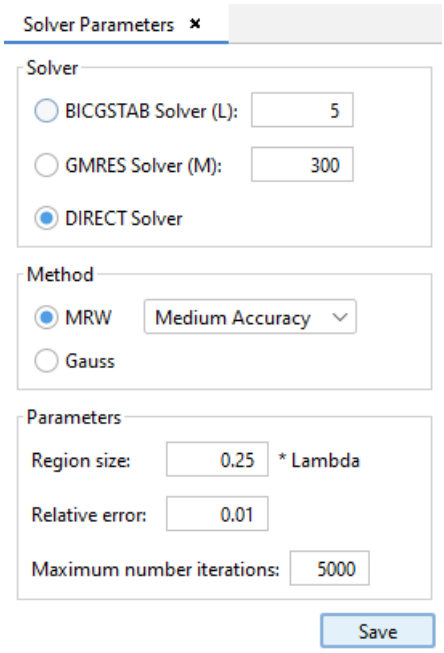


Figure 1798: Solver Parameters

- **Solver**: Algorithm to solve the iterative method. **BICGSTAB** (BiConjugate Gradient STAbilized method) and **GMRES** (Generalized Minimal Residual method) are available. If no convergence is achieved by using any of this methods, try to use the other one. direct solution method may require huge memory and time resources when a large number of unknowns is considered.
- **Method**:
  - **MRW**: Ma-Rokhlin-Wandzura (MRW) specialized quadrature rules are used in the computation of inductive and capacitive contributions of MoM's matrix. User can set this parameter to low, medium or high accuracy.
  - **Gauss**: Conventional Gauss quadrature rules are used in the computation of inductive and capacitive contributions of MoM's matrix.
- **Region Size**: This parameter defines the size edge (in terms of wavelengths) of the regions generated in the MLFMA-MoM (Multi-Level Fast Multipole Algorithm – Method of Moments) algorithm.

- **Relative Error**: It is the maximum value error allowed in the iterative process. When the relative error of any iteration is lower than the value specified, the current computation stay is considered as a valid solution and the iterative process is finished. The smaller is the Relative Error the more accurate is the provided solution, but the larger is the computation time.
- **Maximum number iterations**: Maximum number of iterative steps used to search an iteration that satisfies the specified Relative Error. If the maximum number of iterations is reached without getting a valid solution, the last iteration solution is saved.

Click on **Save** button to confirm the update configuration.

12.9 Meshing Menu

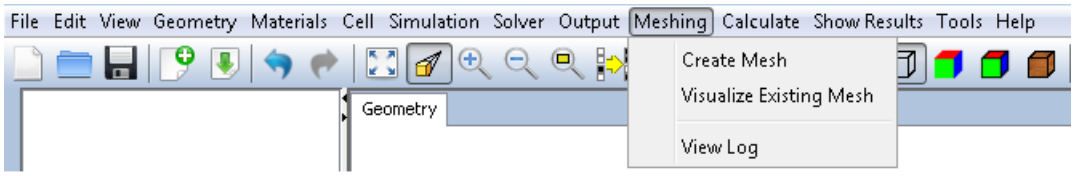


Figure 1799: Meshing Menu

12.9.1 Create Mesh

In the Periodical Structures module, the mesh generation is a required step that must be performed immediately before simulating. In the Meshing Parameters window, represented in the next figure, the following options are available:

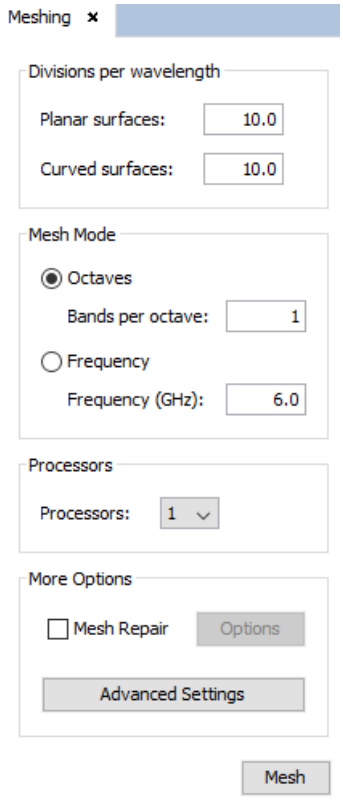


Figure 1800: Meshing Parameters panel

Divisions per wavelength The user may specify different mesh density in planar and curved surfaces. Planar surfaces are contained in a single plane, otherwise surfaces are curved. Let c be the speed of light in the vacuum (meters per second), given a number of divisions **D** and a frequency **f**, the size of the generated elements **L** is given by:

L = lambda / D ; lambda = c / f

This parameters are critical for the resolution of the analysis. By default, 10 divisions means that a flat side of 1 meter generate a mesh of 100 elements (10x10 divisions) if the mesh is 0.3 GHz. This module sizes frequently unit cell geometries are in relation to centimeters or millimeters so these values for meshing may not be sufficient to obtain sufficiently accurate results but meshing frequencies rise to the order of GHz or tens of them. In these cases it is recommended to increase this number of divisions for more accurate results, for example to 20 divisions per wavelength.

Number of bands per octave:
When a frequency swept is enabled, the meshing frequencies are different to the simulation ones. An automatic frequency ranges per octaves is performed, that depends on the initial frequency, the final frequency, and the number of bands per octave specified. The higher is this parameter, the more frequencies are considered for the meshing process.

- **Frequency:**Select this option to use the same frequency for meshing all the frequencies of the swept. The frequency must be specified in GHz.

Processors
Use this to set the number of processors used for the meshing process.

- **Mesh Repair** Select this option to evaluate and repair the generated mesh. It is recommended for analyzing complex geometries, specially whenever a good convergence is not achieved. Several problems will be studied and solved isolated and spurious elements are removed, and overlapped regions are repaired. The detection of this defects on the mesh depends on two parameters that are explained below, and can be edited by clicking on **Options** button. Elements wit smaller area than the minimum allowed, isolated elements, or parallel elements closer than a minimum distance are deleted from the output mesh. The minimum area and distance depends on the **wavelength**, the number of **divisions per wavelength** selected, and the **Area** and **Border**, and they are determined as:

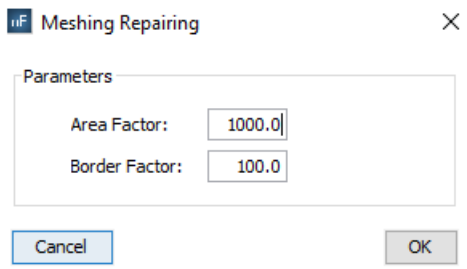


Figure 1801: Meshing Repairing options

$$MinimumArea = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)^2}{AreaFactor_{MeshingRepairParameter}} \quad (13)$$

$$MinimumDistance = \frac{\left(\frac{lambda}{maximum_{divisionsperwavelength}(PlanarSurfaces,CurvedSurfaces)}\right)}{BorderFactor_{MeshingRepairParameter}} \quad (14)$$

- The **Advanced Settings** are explained below.

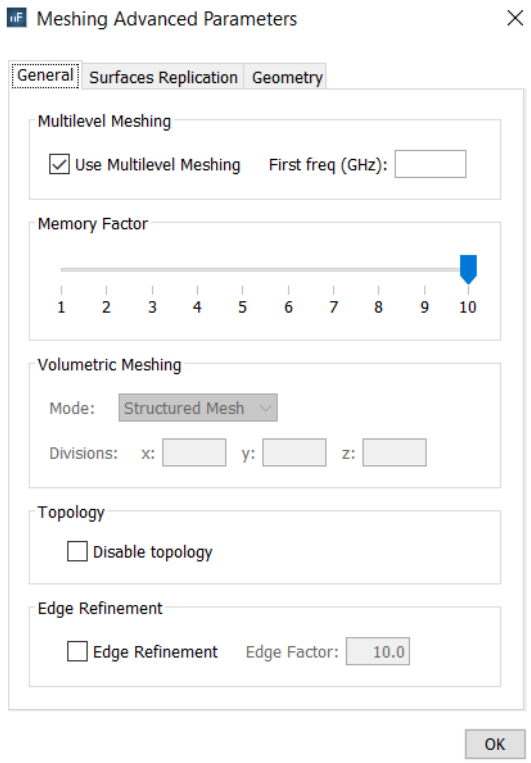


Figure 1802: Meshing Advanced Parameters panel (I), General tab

- General there are some options to specify in this tab.
  - **Multilevel Meshing**. Enable this option to generate the mesh automatically in several steps. This option is usually more efficient (in terms of runtime) than common mesh generation, so it is selected by default. However, minor differences may appear between the meshes obtained with and without multilevel mode. The frequency of the first step may be specified by the user in the **First freq** field, but when this field is empty, the first frequency is automatically computed.
  - **Memory Factor**. This option allows a reduction of the memory resources required in the mesh generation process. The memory allocation is an automatic feature in this process, so the highest one is selected by default to ensure that the mesh will be successfully obtained. If the memory resources exceed the available memory, the message depicted in the next Figure will appear to suggest decreasing this factor.

Figure 1803: Warning panel

- **Volumetric Mode**. This option allows to change the algorithm used in the mesh generation process of dielectric objects (defined as Volumes). Two different modes are available:

- ◦ Structured Mesh All elements of the volumetric mesh are perfect parallelepipeds. This method is an approximation of the real shape of the dielectric objects by simple cubes. Some parts of the mesh can be out of the volume and other regions can have incomplete regions, but the average volume of the mesh is very similar to the volume of the original dielectric objects. This algorithm is the fastest one.
  - ◦ Conformed Mesh Most of elements of the volumetric mesh are perfect parallelepipeds, but only in the completely inner regions of them. The inner structured mesh is joined with the boundaries of the volume by using hexahedrons of variable shapes. This method is more accurate than the Structured Mesh, but is also slower and irregular elements may appear in specific cases.

Different divisions per wavelength for the X, Y and Z dimensions may be considered for meshing the volumetric objects.

- **Topology**. If the electrical continuity between the surfaces have to be broken, the option **Disable topology** must be selected. Note that the accuracy may be reduced when the electrical continuities are not correctly analyzed because virtual fissures are introduced.
- **Edge Refinement**. This option allows enabling the modelling of border effect in the mesh generation. The Edge Factor field represents the portion of size of the final elements to model the border effect. It is only enabled when Edge Refinement is selected.
- Surfaces Replication there is only one option called Parameters in this tab that is composed by some sub-options.

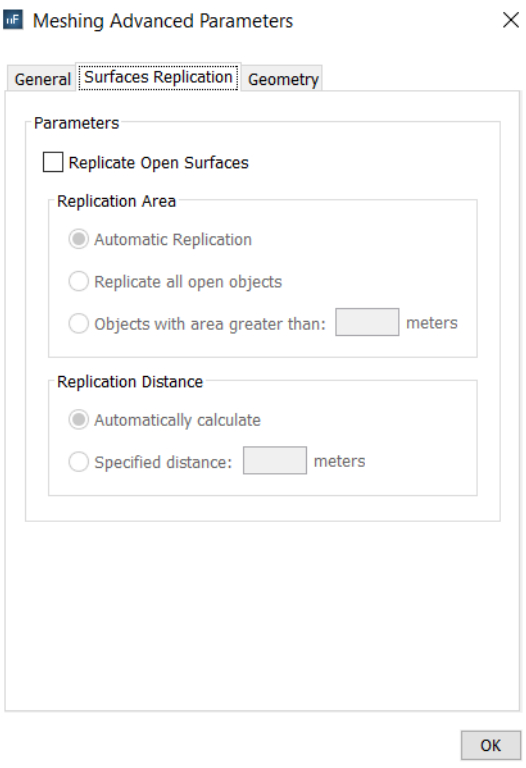


Figure 1804: Meshing Advanced Parameters panel (II), Surfaces Replication tab

- By marking the **Replicate Open Surfaces** option, a parallel layer of open objects is generated in the mesh generation process. If replication is marked, the elements of all the open objects are replicated to an automatic frequency-dependent distance from the original ones. By clicking on the advanced button, the window shown in the next Figure will appear where additional features can be configured.
- **Replication Area**. The criterion to replicate open objects can be edited.

$$a \geq 10 \cdot \left(\frac{lambda}{maximum(div_{plane}, div_{curve})} \right)^2 \quad (15)$$

- Replicate all open objects. Any object is replicated, by default.
- Replicate objects with area greater than. To specify manually the threshold of the minimum total area of the objects to be replicated, in square meters.
- **Replication Distance**. The distance between the original objects and their parallel replicas can also be edited.
  - Automatically calculate. By default, the distance is given by the following expression:

$$d = 0.03 \cdot \left(\frac{lambda}{maximum(div_{plane}, div_{curve})} \right) \quad (16)$$

- Specified distance. To specify manually the desired distance, in meters.
- Geometry the followings parameters can be specified.

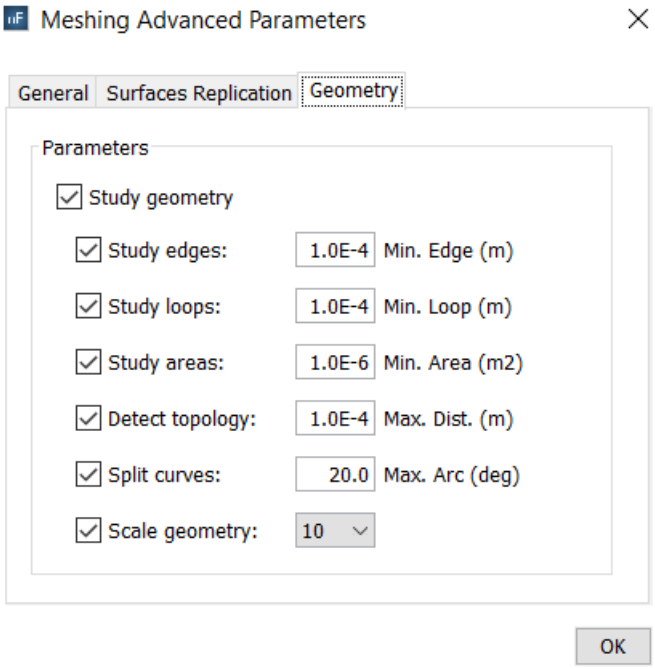


Figure 1805: Meshing Advanced Parameters panel (III), Geometry tab

- **Study Geometry**. To pre-process the input geometry, evaluating some features such as its sizes or electrical continuities.

- **Study Edges**. To delete the edges of the input surfaces that are shorter than the Minimum Edge parameter (in meters), extending its adjacent edges. This option is only enabled when the Study Geometry button is enabled.
- **Study Loops**. To delete the loops of the input surfaces that are shorter than the Minimum Loop parameter (in meters). This option is only enabled when the Study Geometry button is enabled.
- **Study Areas**. To delete the input surfaces that are smaller than the Minimum Area parameter (in square meters). This option is only enabled when the Study Geometry button is enabled.
- **Detect Topology**. To detect automatically electrical continuity between neighboring surfaces that are very close but have not been modeled with precise continuity. The maximum separation allowed to set the electrical continuity is the Maximum Distance parameter (in meters). This option is only enabled when the Study Geometry button is enabled.
- **Split Curves**. To divide the curved borders of the input surfaces that have a curvature greater than the Maximum Arc parameter (in degrees). This option is only enabled when the Study Geometry button is enabled.
- **Scale Geometry**. To scale internally the geometry during the meshing process. When the mesh generation finishes, the output mesh has the same sizes as the original geometry. This option is useful for meshing very small structures. This option is only enabled when the Study Geometry button is activated.

12.9.2 Visualize Existing Mesh

This option can be chosen by the user to load the meshes files (msh file extension). When this option is selected the following window appears:

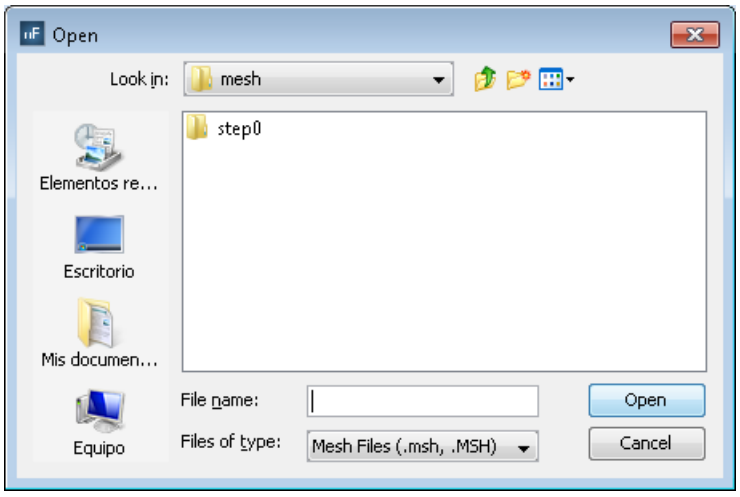


Figure 1806: Visualize Existing Mesh

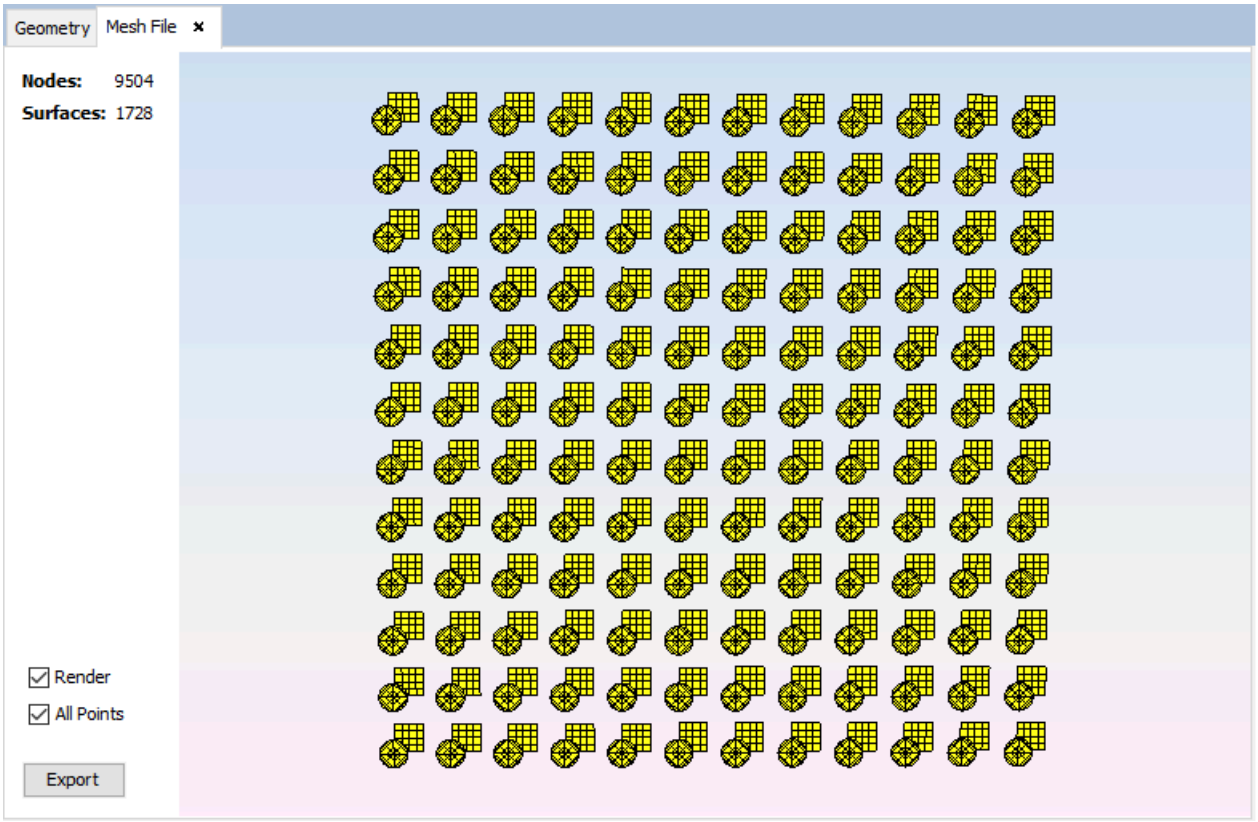


Figure 1807: Mesh View

12.9.3 Visualize Mesh Log

This option loads a file named mesh_log.txt that contains information about meshes. This information is printed in a new tab on the screen.

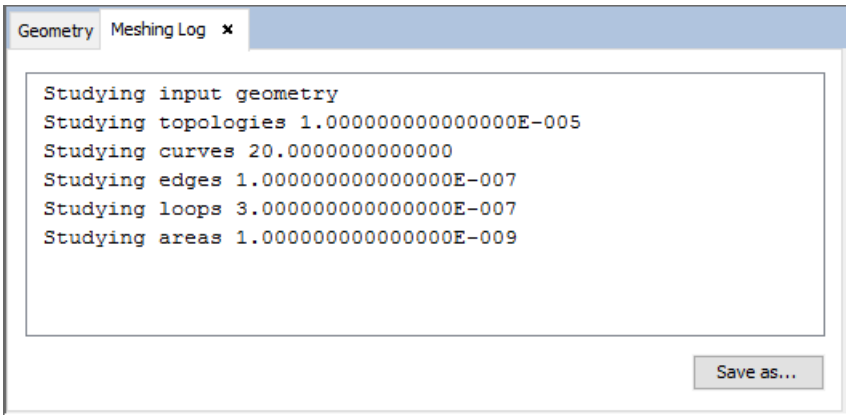


Figure 1808: Mesh Log

12.10 Calculate Menu (Alt + C)

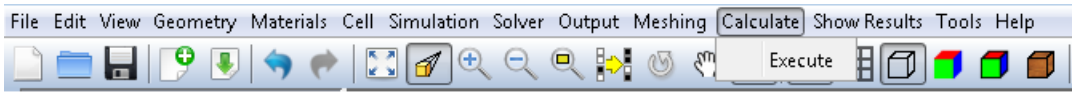


Figure 1809: Calculate Menu

Before executing the simulation the user needs to make sure that every simulation parameter has been correctly configured. If something is wrong with a user defined parametera warning message is displayed to let the user know that it is necessary to correct one or more parameters.

12.10.1 Execute

This option runs the simulation. The user can define the number of processors that will run the simulation.

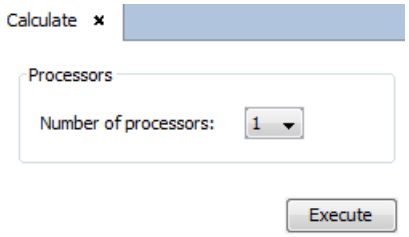


Figure 1810: Execute panel

When the simulation starts, a new screen is displayed:

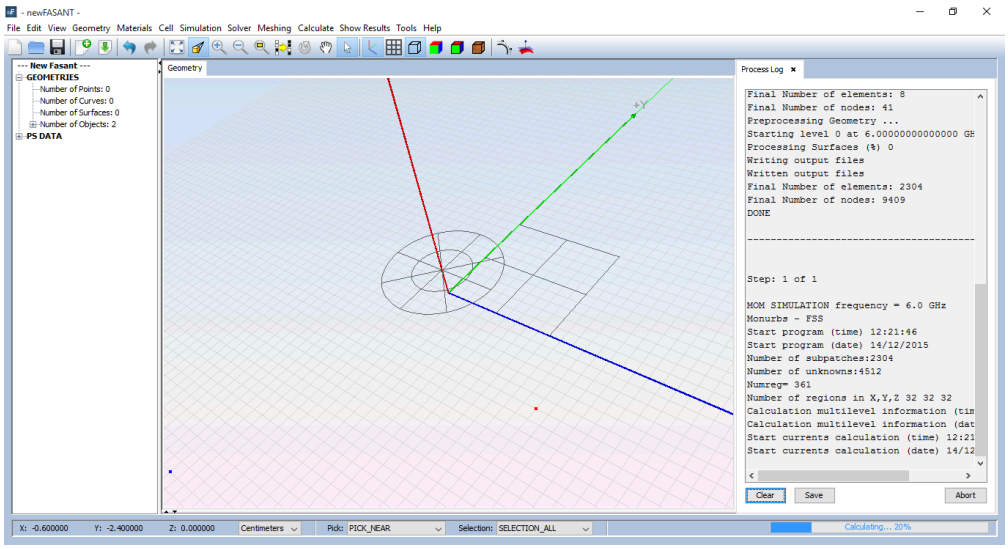


Figure 1811: Processing command

During a solution process, the screen reports the status of each phase of the calculation process:date and CPU time for every phase of the solution process, etc.

This data can be saved by the user by clicking the Save button, as shown in the next Figure.

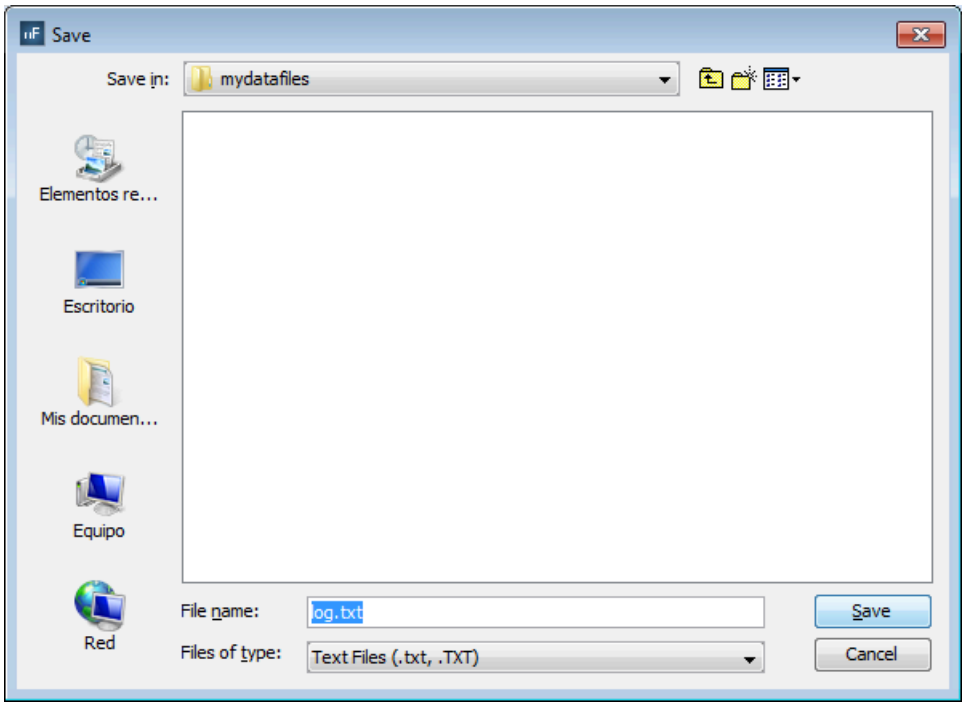


Figure 1812: Save processing

12.11 Show Results Menu

This option allows the user to see the results obtained on the project simulation.

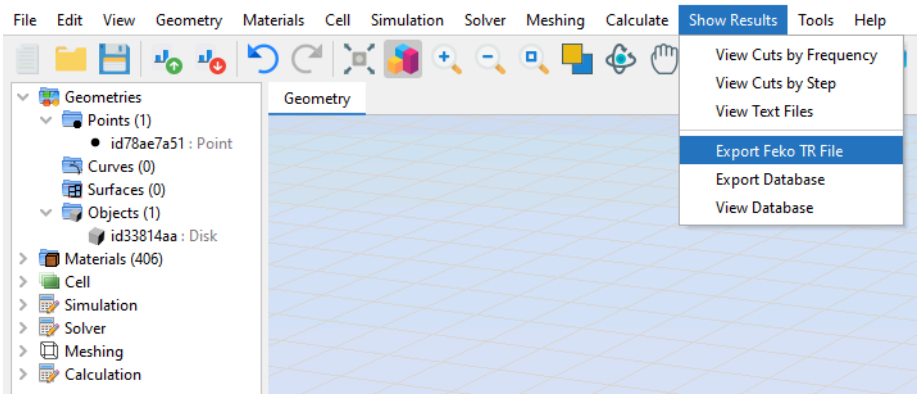


Figure 1813: Show Results Menu

Most of plots show 2-D curves of a result field versus an input parameter. This type of graphics share additional features of zoom, raxis and curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

12.11.1 View Cuts by Frequency

This command plots the amplitude (dB or nat) or the phase of the fields computed for each frequency of the simulation periodical structure. The user can select different components to view.

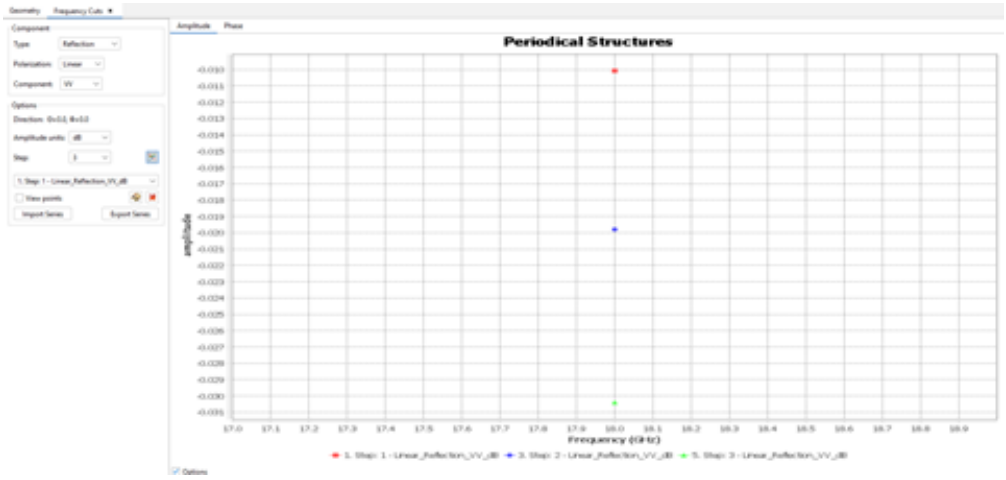


Figure 1814: View cuts by frequency

It is also possible to delete the selected trace of the list with the Remove Series button. The display option allows to change the color of the series and display points. The Import Series and Export Series buttons are used for importing and exporting the selected series in List of Series to a data text file.

12.11.2 View Cuts by Step

This command plots the amplitude (dB or nat) or the phase of the fields computed for each step of the simulation periodical structure. The user can select different components to view.

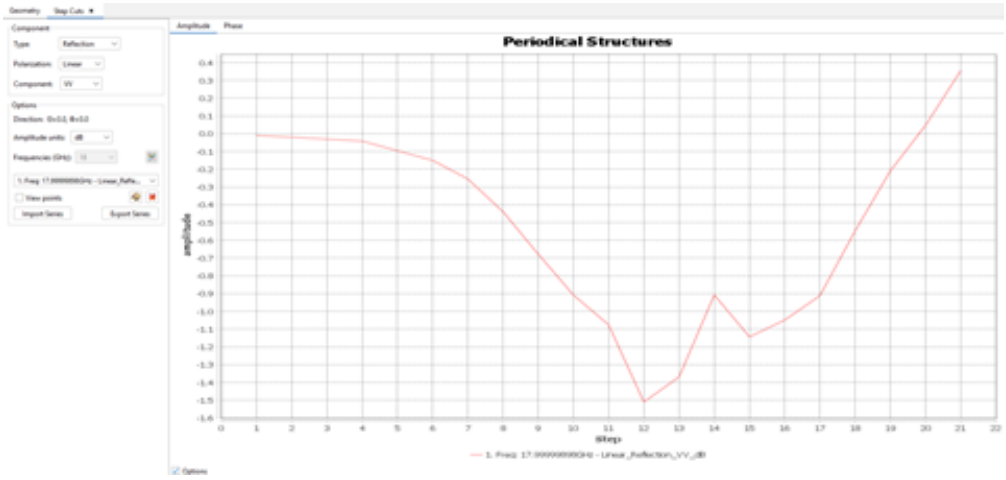


Figure 1815: View cuts by step

It is also possible to delete the selected trace of the list with the Remove Series button. The display option allows to change the color of the series and display points. The Import Series and Export Series buttons are used for importing and exporting the selected series in List of Series to a data text file.

12.11.3 View Text Files

This command shows the calculated results as a text file.

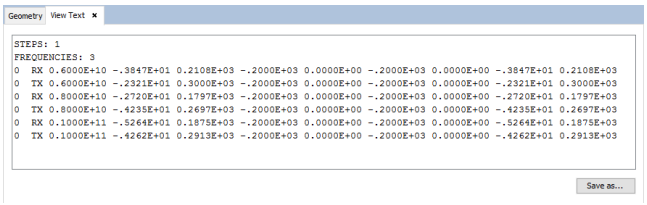
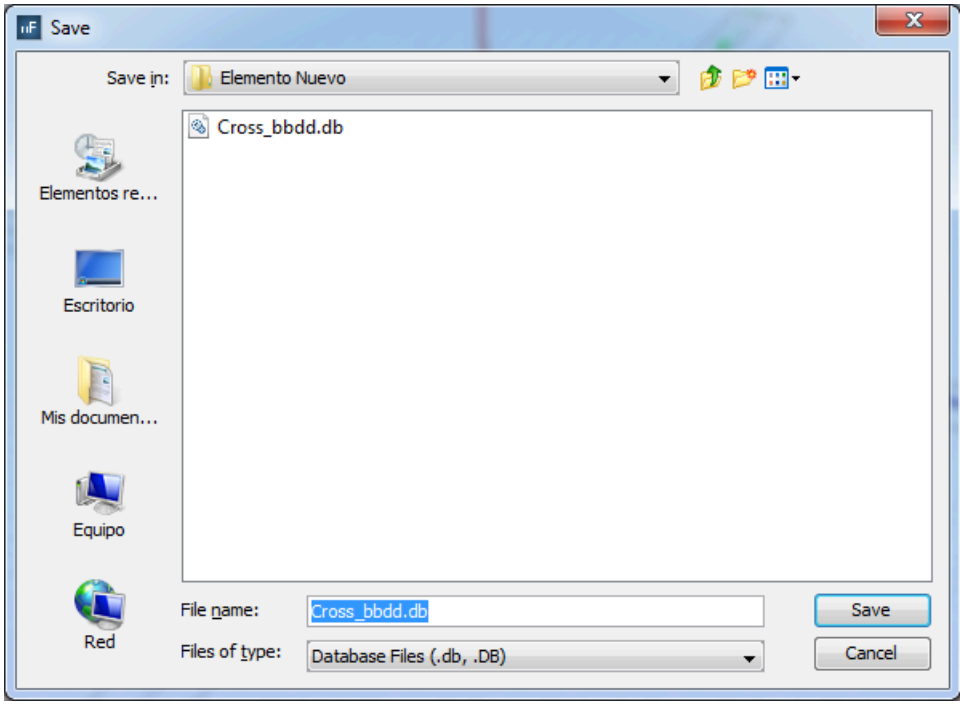


Figure 1816: View Text Files

12.11.4 Export Database

This option allows the user to export the database in a .db file. This database can be used in MoM module to generate the reflectarray/transmitarray layout.

See Annex 1: Creating a Reflectarray Database and New Layout for more information.



12.11.5 Export Feko TR File

This option allows the user to export a `.tr` file. This file could be used in both Feko and newFASANT.

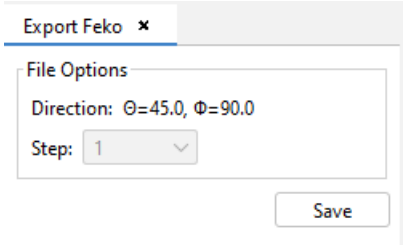


Figure 1817: Export a `.tr` file.

Where **step** is the parametric step to select. These should have been defined prior to the calculation when designing the simulation.

The data structure for the `.tr` files is described in the Feko User Guide (see Appendix *A-11.1 Native Files*).

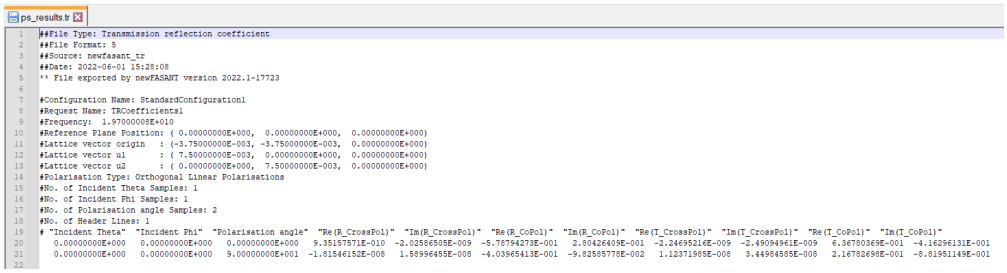


Figure 1818: An example of a `.tr` file.

12.11.6 View Database

This command allow the user to represent all the information about the generated database, if it would be generated. The panel show information about the number of steps or different configurations of the cell, including the option to represent each one, the material layers, cell dimension and replication and if the cell has a skew angle. This panel show information about if the generated object is reflectarray or transmitarray and information about the phase values for a selected frequency of the computed frequencies and the desire component. The phase can be represented on a table of values or plot.

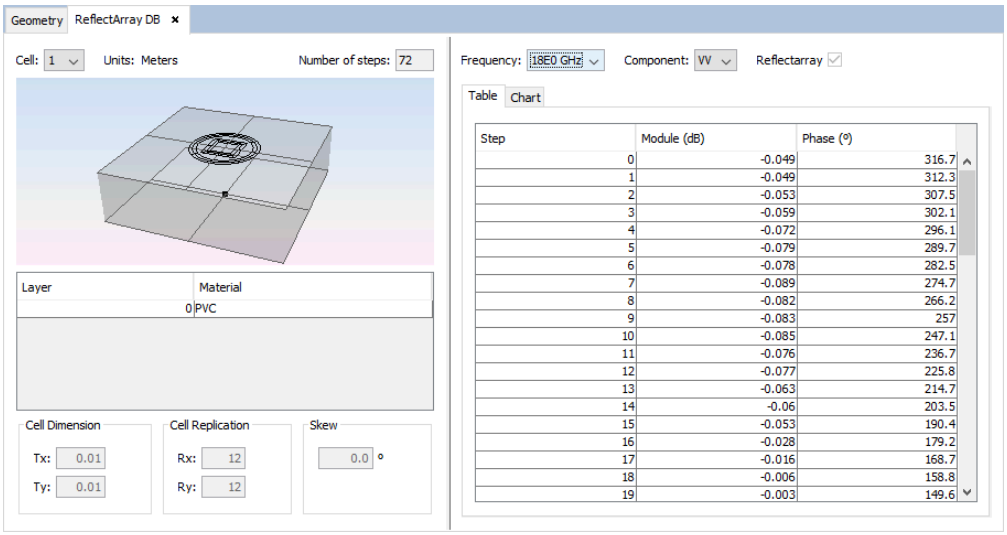


Figure 1819: View DB information

12.12 Tools Menu

For information about the File menu, refer to File Menu.

12.13 Help Menu

For information about the File menu, refer to File Menu.

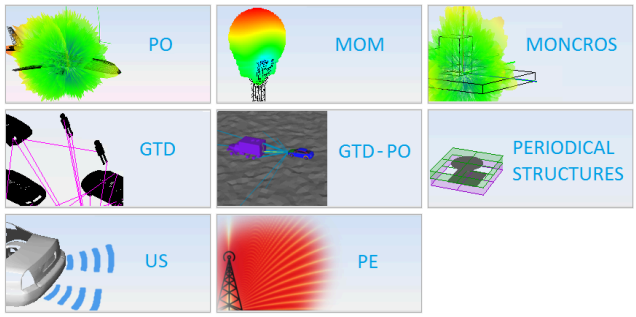
12.14 Training Examples

12.14.1 Example 1: Analysis of a Cell with Disk Geometry

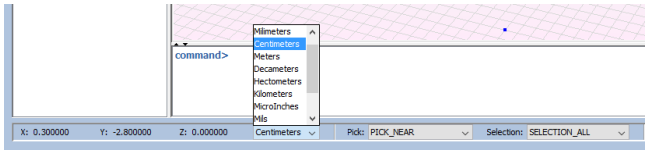
Step 1: Start newFASANT.

Step 2: Select File and click on New.

Step 3: Select Periodical Structures.



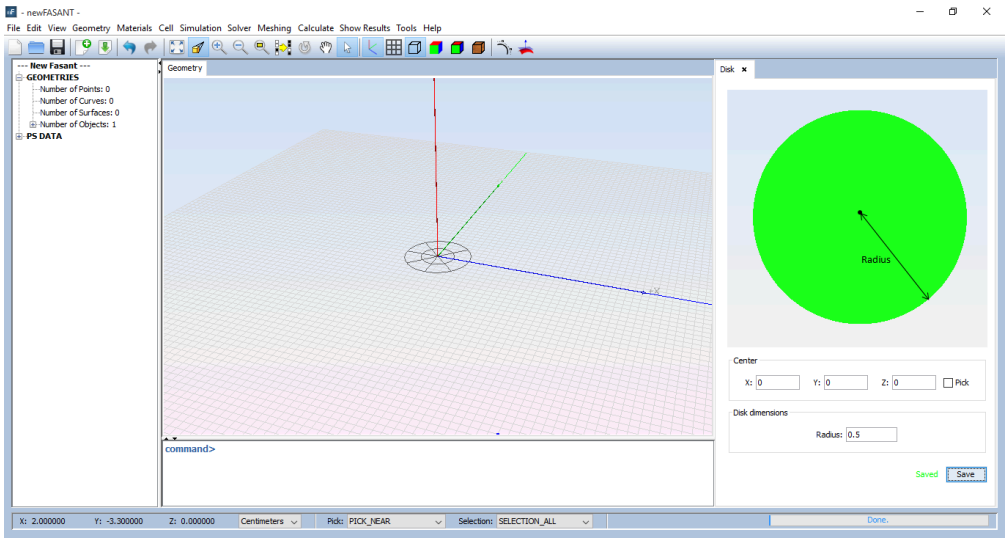
Step 4: Select Centimeters units.



Step 5: Click on "Cell → FSS Primitive → Disk", which requires the center and the radius, as shown in the next figure.

In this example the user enters the following values:

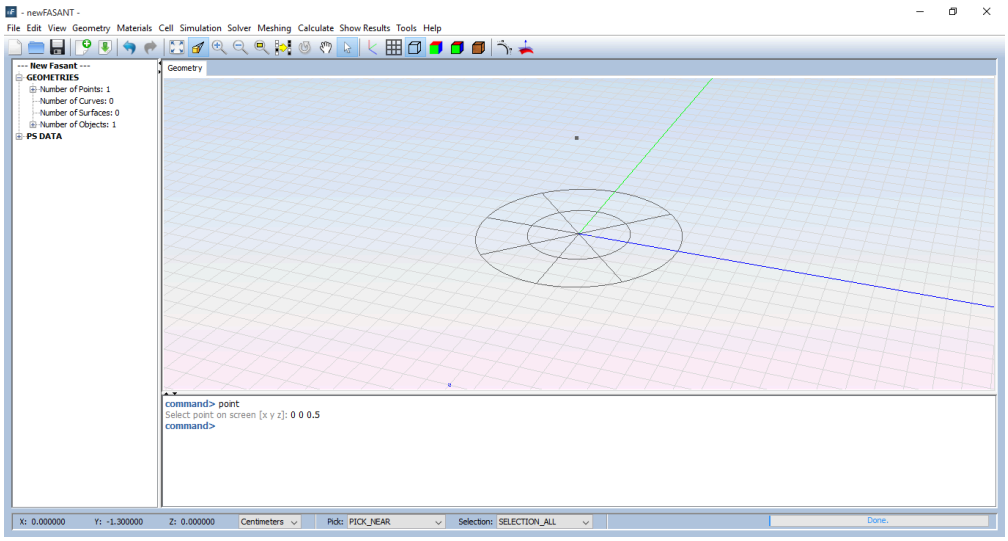
- Center 0 0 0
- Radius 0.5



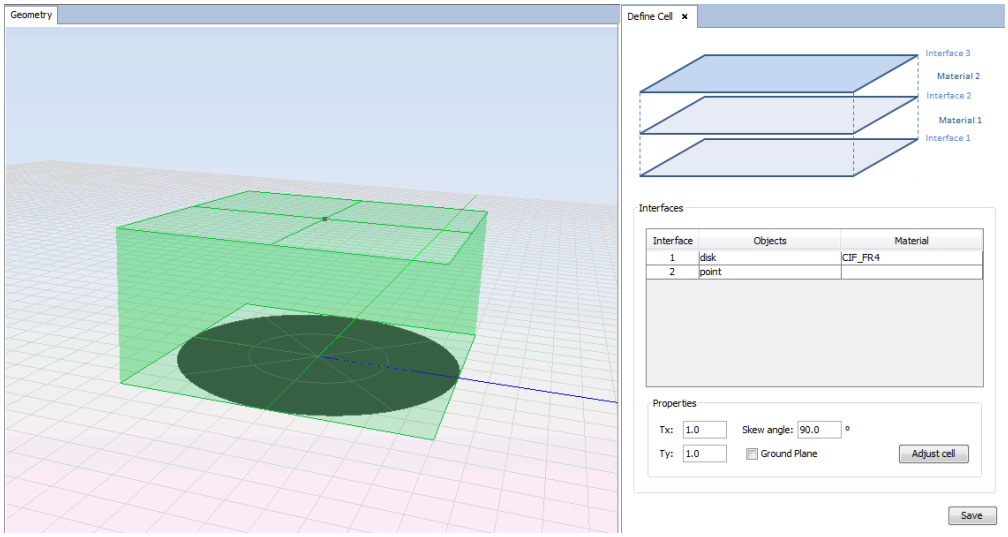
Step 6:. Click on "Geometry → Point → Single point", which requires the x,y and z coordinates as shown in the next figure.

In this example the user enters the following values into the command line:

- Select point on screen [x y z] 0 0 0.5



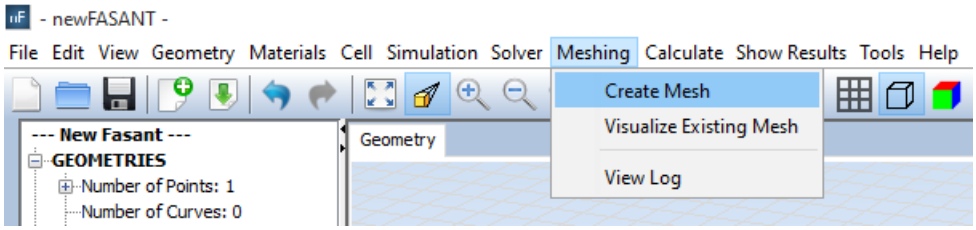
Step 7: Click on "Cell → Define cell". Assign the material of the layer.



Step 8: Click on "Simulation → Parameters". Configure the simulation with this parameters:

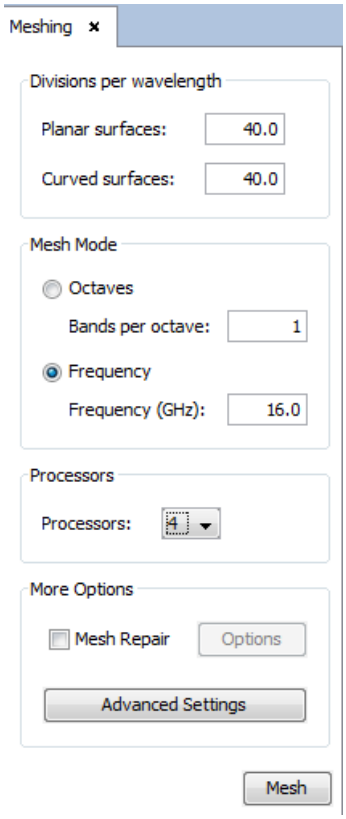
- Units: GHz
- Initial frequency: 10
- Final frequency: 16
- Samples: 4
- Planewave Definition: Symmetric

Step 9: Before running the case, select "Meshing → Create Mesh".

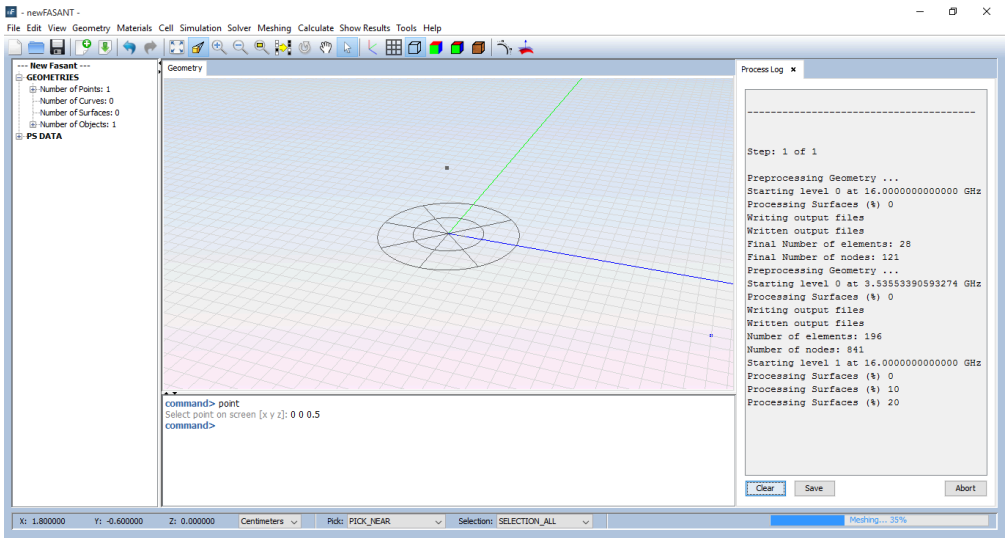


Configure the meshing with this parameters:

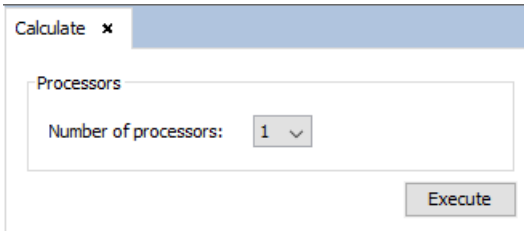
- Planar surfaces: 40
- Curved surfaces: 40
- Mesh mode: Frequency, 16.0 GHz
- Processors: (Processors available)

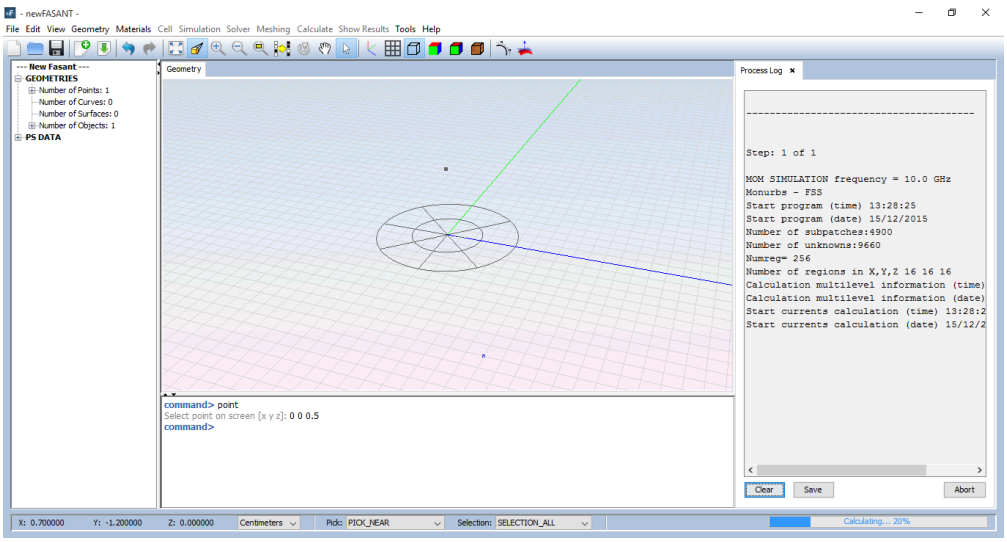


Click on "Mesh" button launch the meshing engine as shown in the next figure:

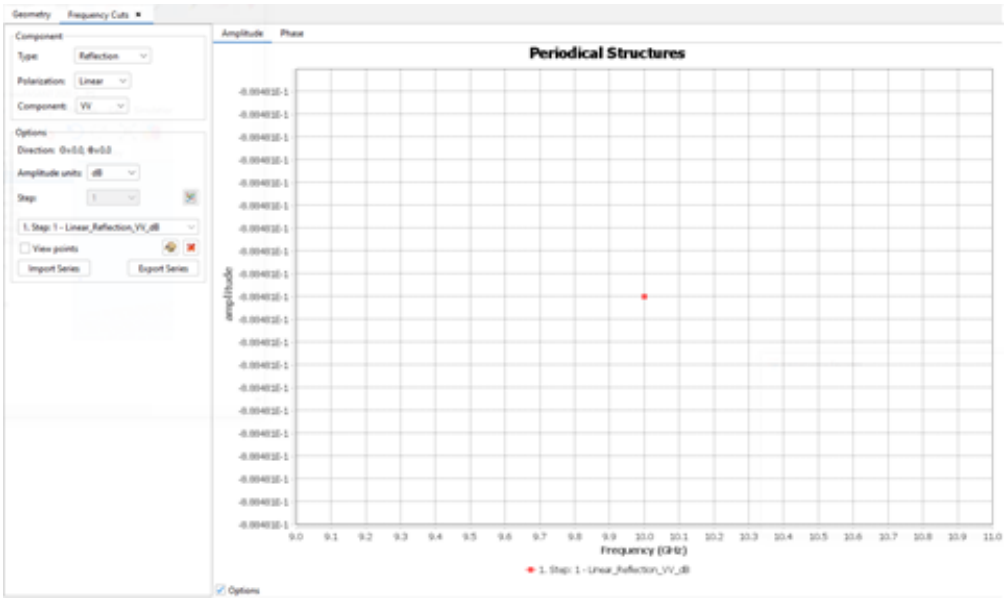


Step 10: Select "Calculate → Execute" and select the number of processors available.

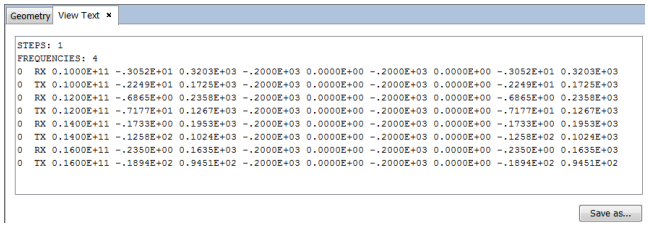




Step 11: When the simulation finishes we can visualize the simulation results. Click on "Show Results → View Cuts By Frequency".



Step 12: To view text file results, click on "Show Results → View Text Files".

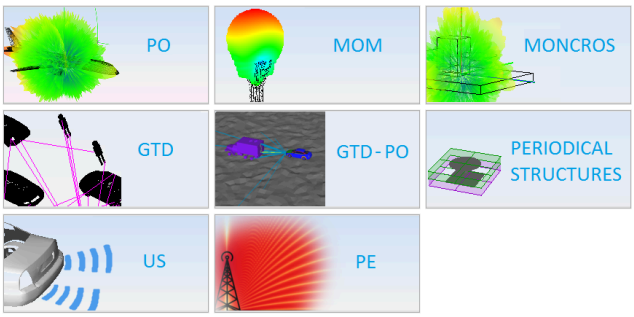


12.14.2 Example 2: Analysis of a Cell with Spiral Geometry and Skew Angle

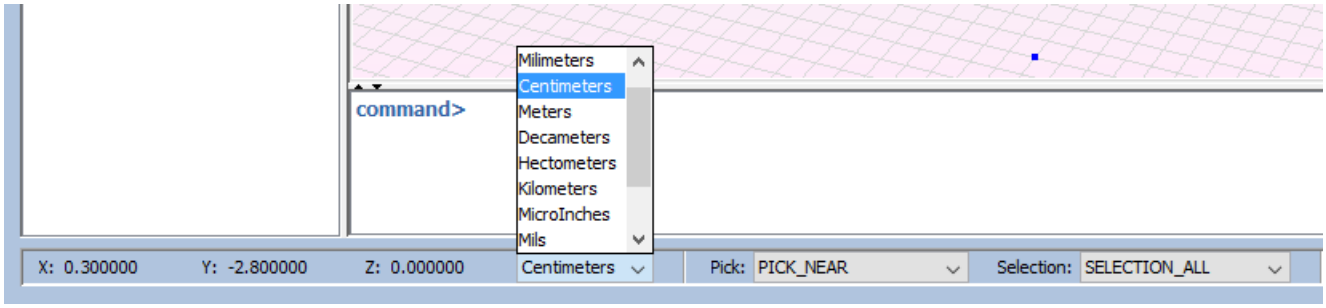
Step 1: Start newFASANT.

Step 2: Select File and click on New.

Step 3: Select Periodical Structures.



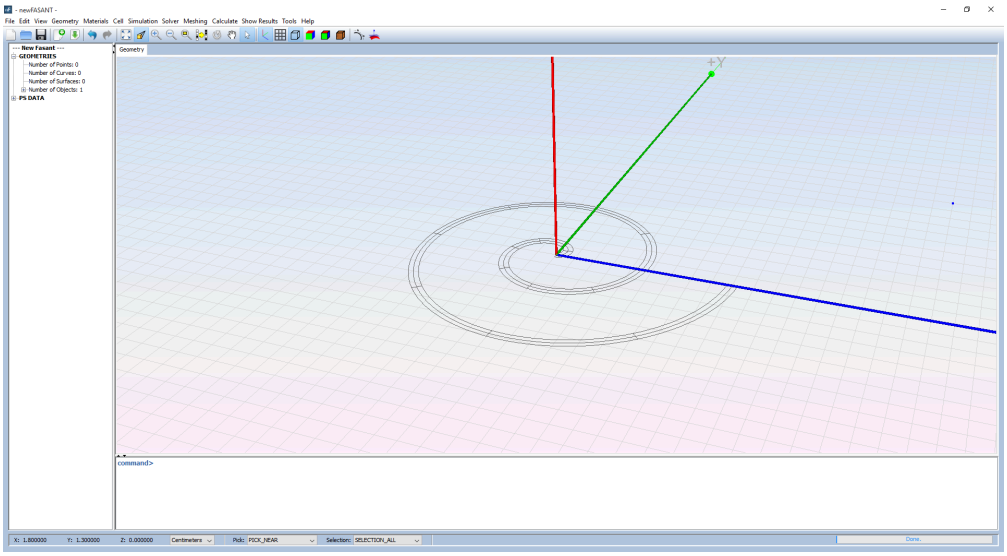
Step 4: Select Centimeters units.



Step 5: Click on "Cell → FSS Primitive → Strips → Spiral", which requires the center, width, number of turns and the radius.

In this example the user enters the following values:

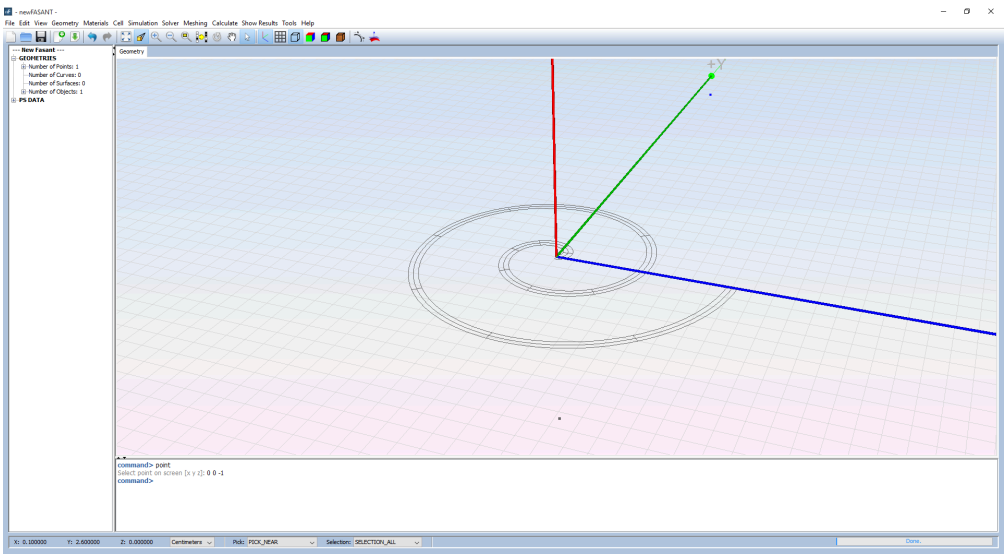
- Center position: 0 0 0
- Width: 0.05
- Number of turns: 2
- Radius: 0.9



Step 6: Click on "Geometry → Point → Single point", which requires the x,y and z coordinates as shown in the next figure.

In this example the user enters the following values into the command line:

- Select point on screen [x y z]: 0 0 -1



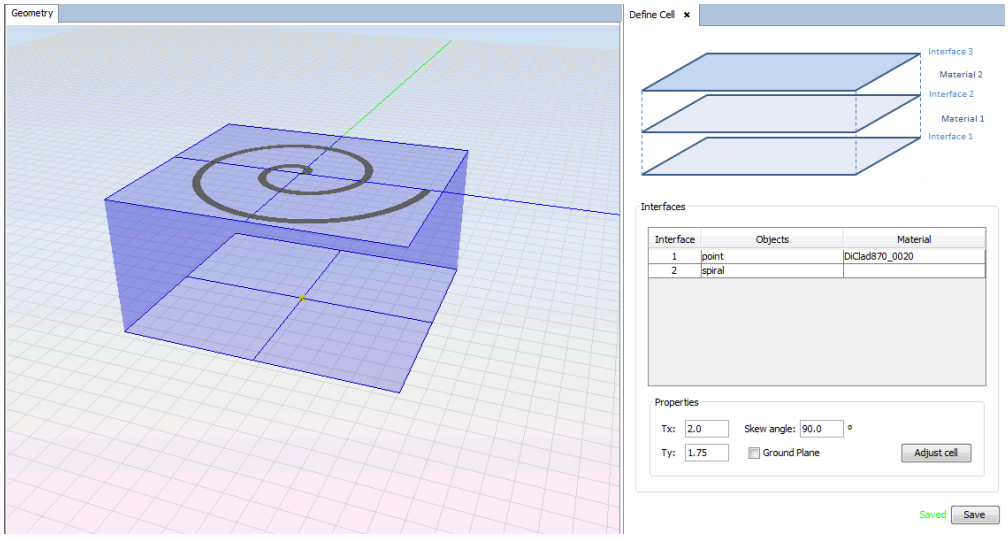
Step 7: Click on Cell → Define cell.

- Assign the material of the layer. The selected material is DiClad870_0020

Change the cell dimensions to ensure that the cell contains the whole metallic structure.

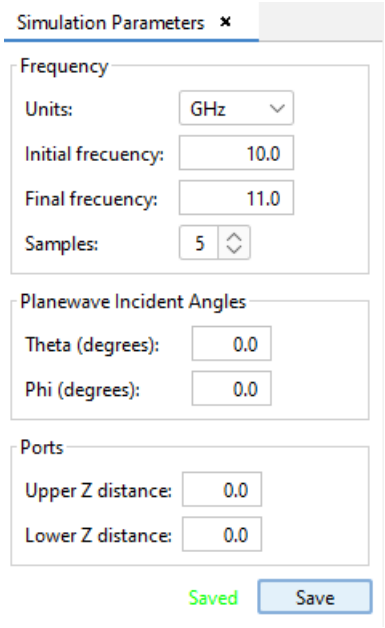
- Tx: 2
- Ty: 1.75

The final design of the cell is shown in the next figure.

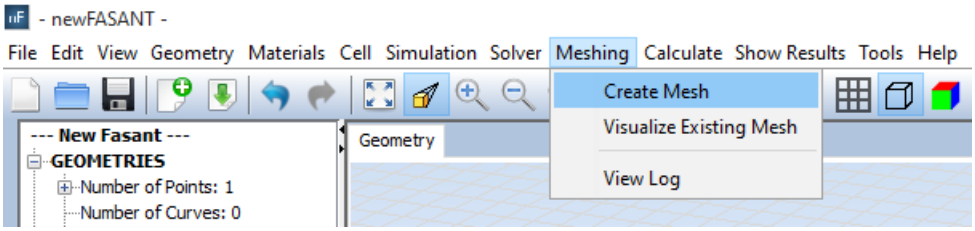


Step 8: Click on "Simulation → Parameters". Configure the simulation with this parameters:

- Units: GHz
- Initial frequency: 10
- Final frequency: 11
- Samples: 5
- Theta incidence: 0.0
- Phi incidence: 0.0

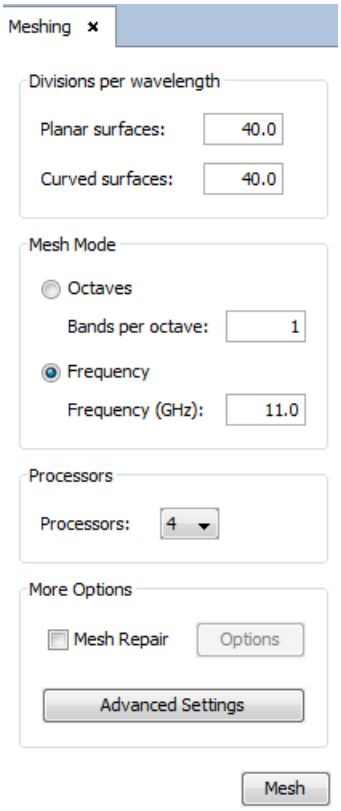


Step 9: Before running the case, select "Meshing → Create Mesh".

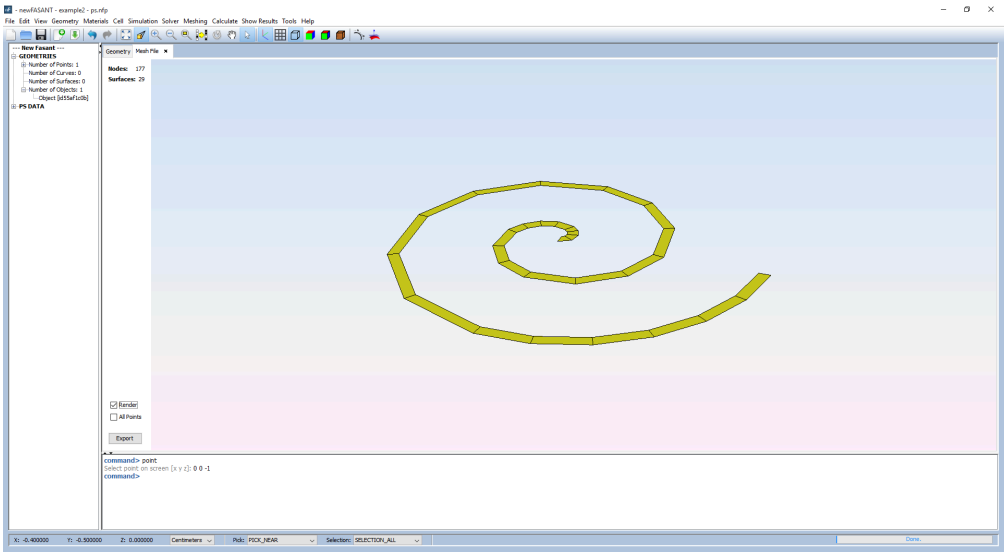


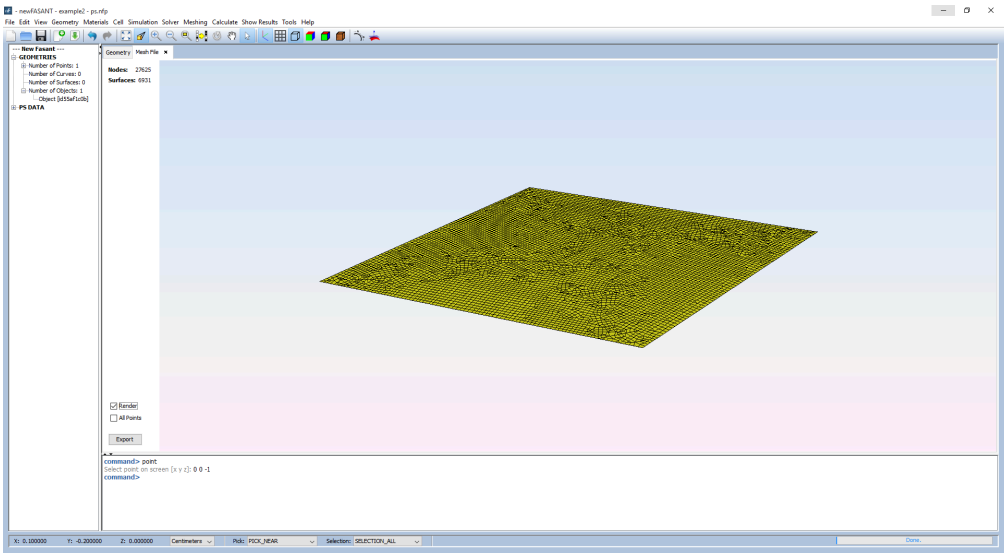
Configure the meshing with this parameters:

- Planar surfaces: 40
- Curved surfaces: 40
- Mesh mode: Frequency, 11.0 GHz
- Processors: 2



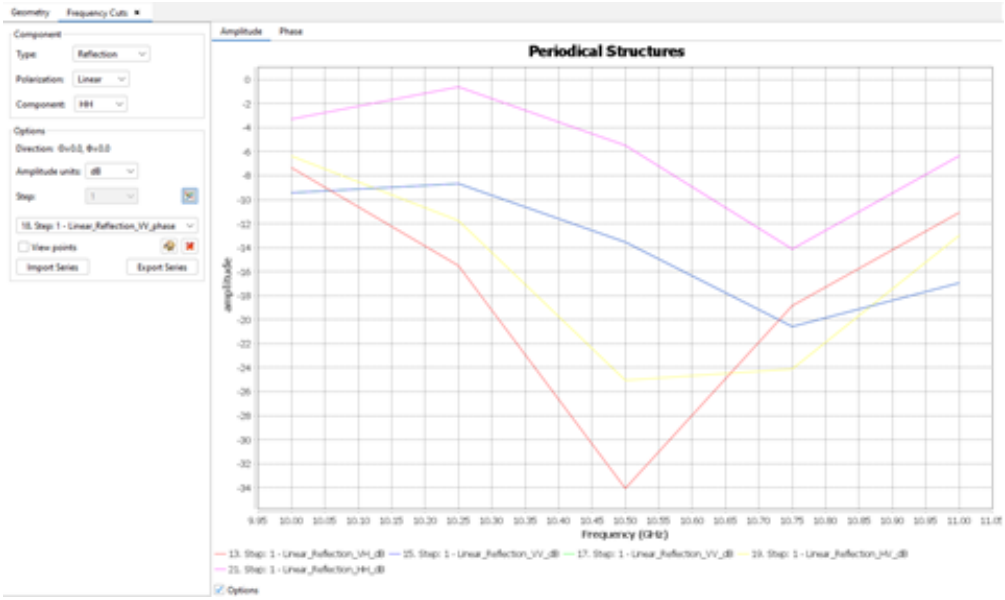
Click on "Meshing → Visualize Existing Mesh" to visualize all the generated meshes.



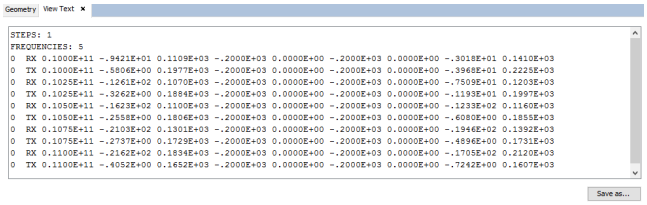


Step 10: Select "Calculate → Execute" and select the number of processors available.

Step 11: When the simulation finishes we can visualize the simulation results. Click on "Show Results → View Cuts By Frequency".



Step 12: To view text file results, click on "Show Results → View Text Files".



12.15 Annex 1: Creating a Reflectarray Database

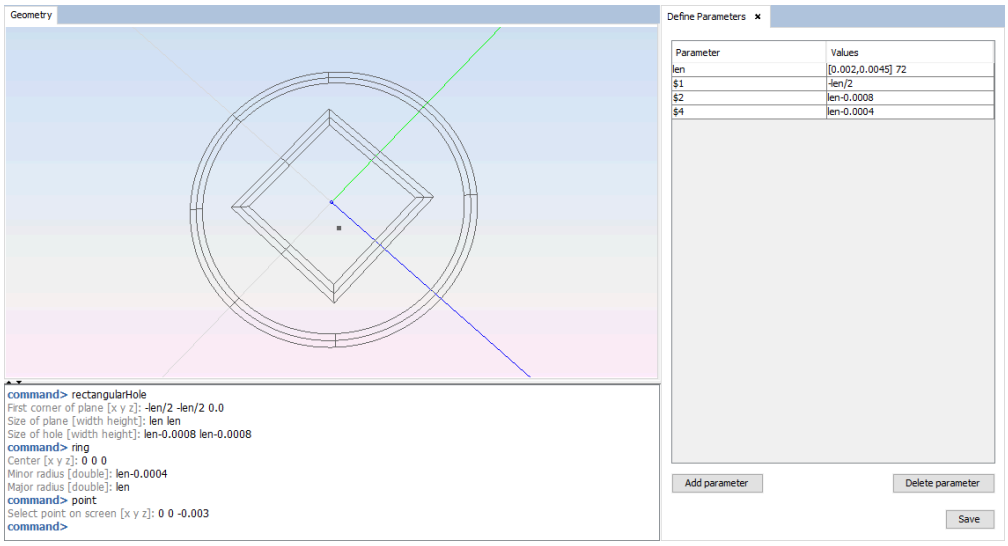
This chapter summarizes the main steps for generating a database in order to build and analyze reflectarray/transmit array structure.

Step 1: Set the variable parameters for the structure. Select "Geometry --> Parameters --> Define Parameters" option on the menu bar. On this tab, all the parameters used in the construction of geometry will be set, see Define Parameters.

Step 2: Build the geometry for the elementary cell of the reflectarray / transmitarray structure. For this purpose the user can use all options available to construct geometries (see Geometry Menu) only with the following restrictions:

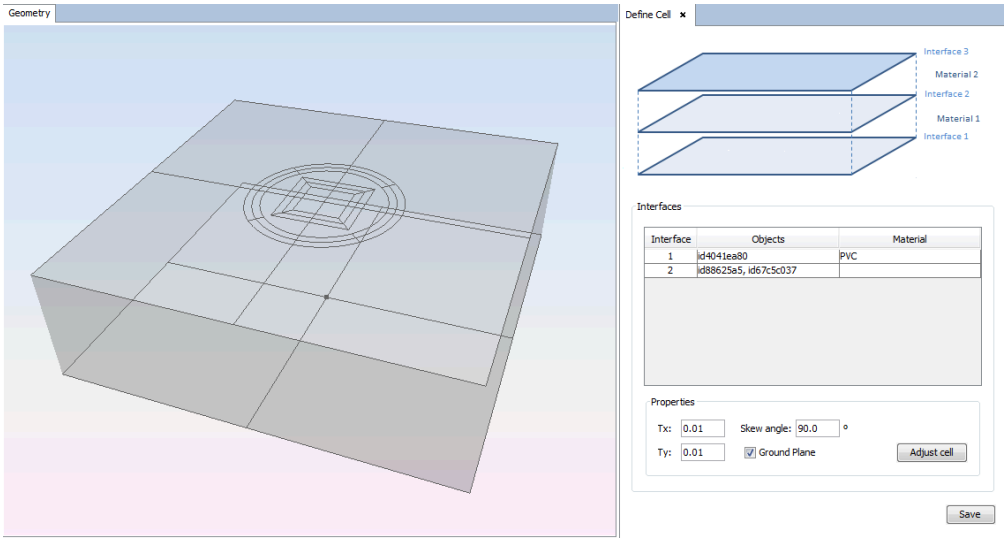
1. The used geometries must be planar geometries.
2. The geometries must be built in parallel planes to the XY plane (floor).
3. If the user wants to leave some empty interfaces on the cell, a point for each empty interface must be added to detect an interface on the cell editor tab .
4. The used geometries must be created with some parameters in order to have different cells.

This is an example of a cell geometries, where two elements has been added at the same interface, and a point has been inserted below to set the thickness of the material:



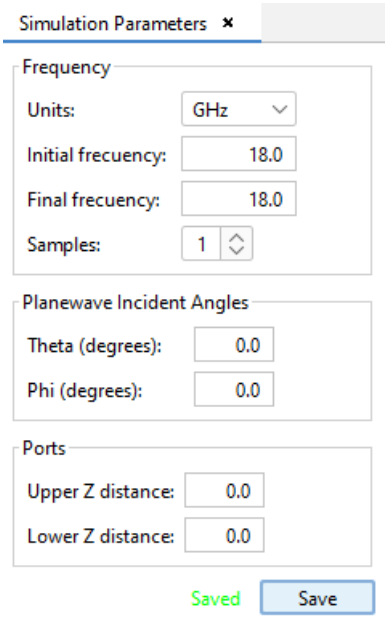
Note: Only first parameter (called 'len') has been defined, and the next ones are auxiliary parameters that depends directly on the first one. The auxiliary parameters are automatically defined when an operation is performed on an existing parameter, and their name starts always with the '$' character to be identified.

Step 3: Build the elementary cell of the structure. Select "Cell --> Define Cell" option on the menu bar. On this tab the parameters as layer material, cell size, cell replication, cell type (reflectarray or transmitarray) or skew angle will be configured.

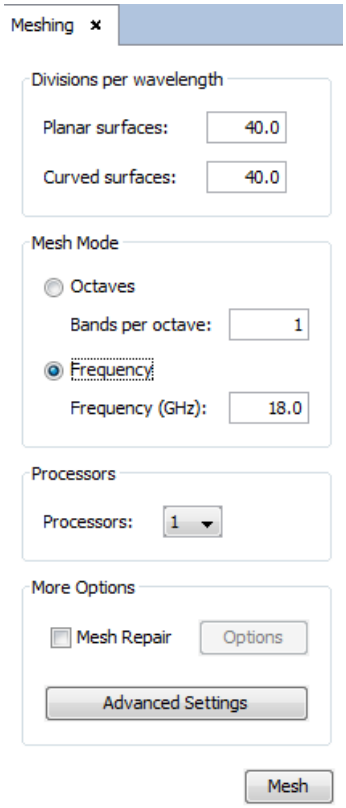


The example has two interfaces, defined by the two geometries added on the previous step. The materials for the layers are defined as the image shows, this is one material for each interface except on the top interface.

Step 4: Set simulation parameters. On this tab the values for the frequency, planewave and observation direction will be configured. For database creation is recommended to select only one frequency and if the cell geometry is not symmetric for the vertical and horizontal polarizations select asymmetric polarization type to analyze both polarizations.

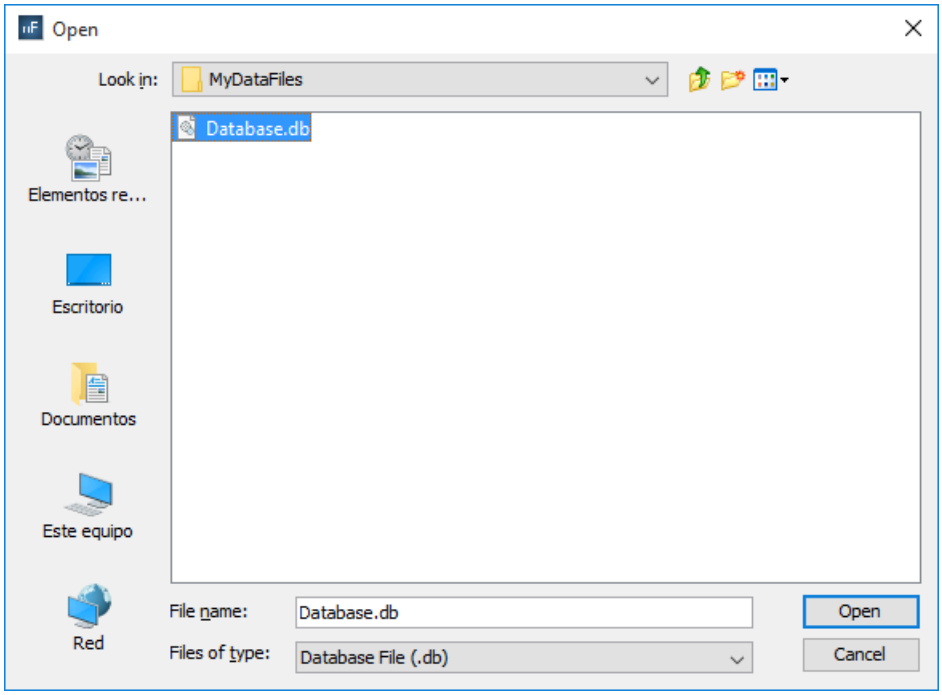


Step 5: Meshing the periodical cell structure. Select "Meshing --> Create Mesh" option on menu bar. Parameters as divisions per wavelength, frequency and number of processors will be configured.

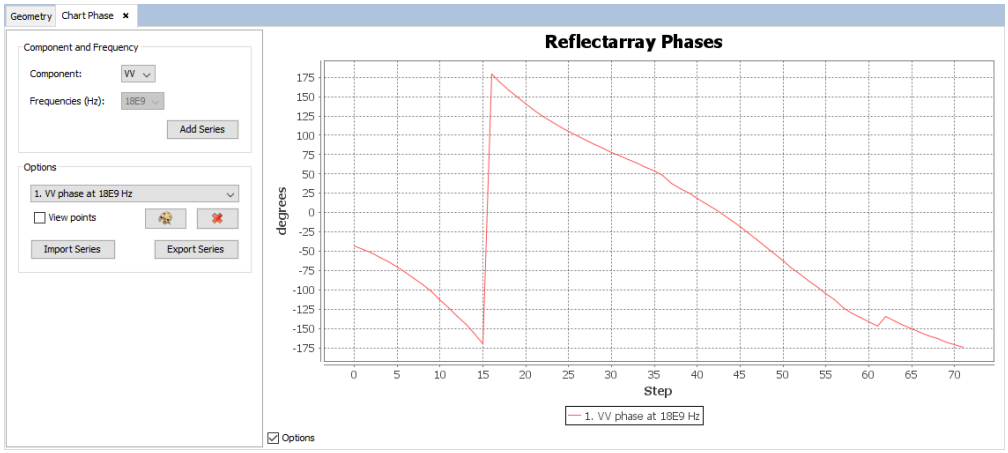


Step 6: Analyze the periodical cell structure. Select "Calculate --> Execute" option on menu bar.

Step 7: Select "Show Results --> Export Database".

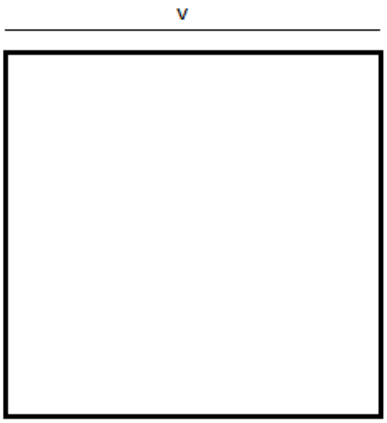


Step 8: Show results. Select "Show Results --> View Cuts by Step".


Geometry
Chart Phase
Component and Frequency
Component:
VV
Frequencies (Hz):
18E9
Add Series
Options
1. VV phase at 18E9 Hz
View points
Import Series
Export Series
Reflectarray Phases
degrees
Step
1. VV phase at 18E9 Hz
Options

12.16 Annex 2: Analysis of a Reflectarray Database Creation

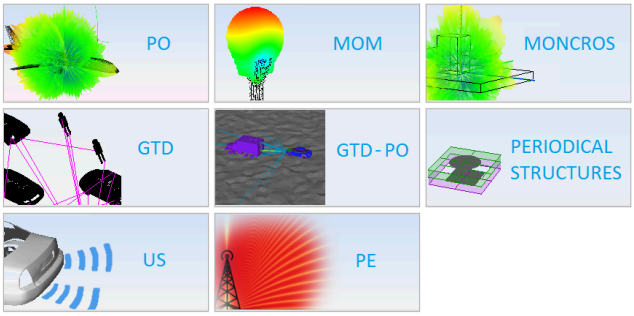
This chapter analizes the squared geometry model proposed by F. Zubir et al.[3]



This case analizes cases varying values on two of the critical parameters for this type of cases:

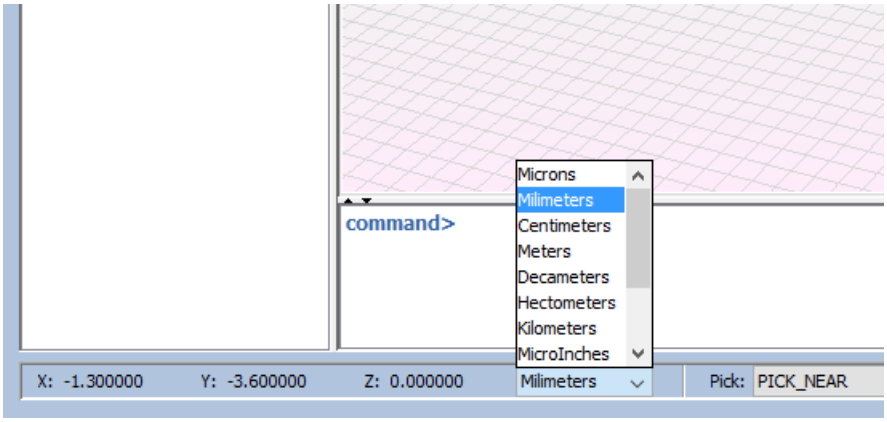
1. Two possibilities for the geometry steps 7 and 37 different sizes of the geometry.
2. Two possibilities for the number of divisions per wavelength on the meshing process 10 and 20 division.

Step 0: Open newFASANT software and generate a new PERIODICAL STRUCTURES project selecting the option on the tab that appears when the 'File --> New' option of the menu bar is selected.

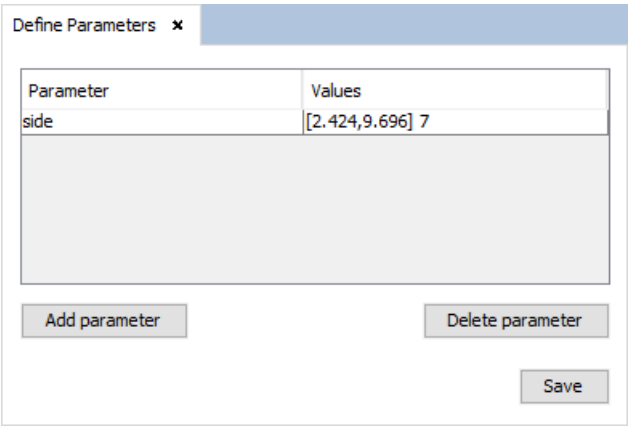


Then, set the interface units on 'millimeters' using the selection list on the bottom left of the main window.

3. reference: F. Zubir, M. K. Abd Rahim, O. B. Ayop, and H. A. Majid, "Design and analysis of microstrip reflectarray antenna with minkowski shape radiating element," Progress In Electromagnetics Research B, Vol. 24, 317-331, 2010. doi:10.2528/PIERB1007120, http://www.jpier.org/pierb/pier.php?paper=10071208



Step 1: Set the variable parameters for the structure. Select "Geometry --> Parameters --> Define Parameters" option on the menu bar. On this tab, all the parameters to be used in the construction of geometry will be set as described in Define Parameters.

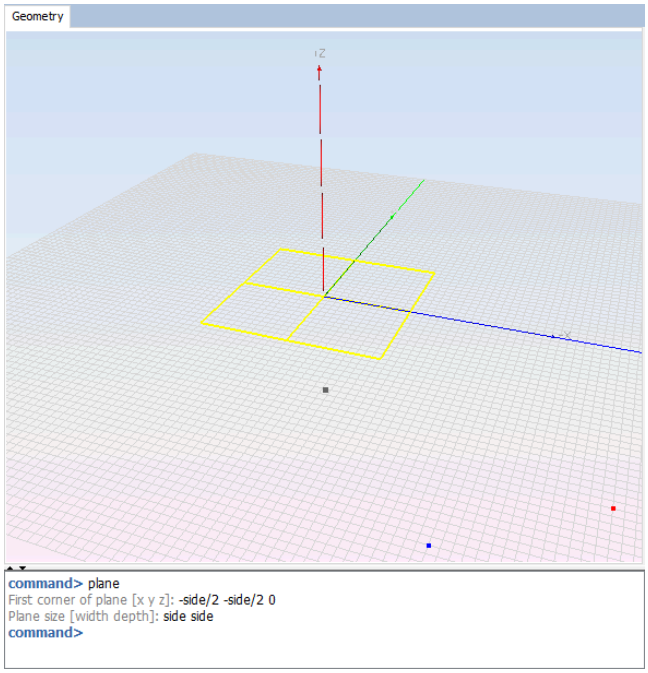


This parameter is referenced to the size of the plane for the resonant frequency at 11 GHz. For this frequency parameter 'v' = 6.06mm. This way the plane has 7 different sizes having in the middle step the size of the resonant frequency at 11 GHz and in the upper step the maximum size possible to the cell that will be defined on next steps. Note to use 37 steps for 37 different sizes the parameters must be defined changing the samples value of 7 to 37 on the parameters definition.

Step 2: Build the geometry for the elementary cell of the reflectarray/transmitarray structure. For this purpose the user can use all options available to construct geometries (see Geometry Menu) only with the following restrictions:

1. The used geometries must be planar geometries.
2. The geometries must be built in parallel planes to the XY plane (floor).
3. If the user wants to leave some empty interfaces on the cell, a point for each empty interface must be added to detect an interface on the cell editor tab.
4. The used geometries must be created with some parameters in order to have different cells.

Generate a plane typing 'plane' on the command line panel, and introducing the parameters as the next figure shows. To see information about the use of the command line panel, see Command Line.

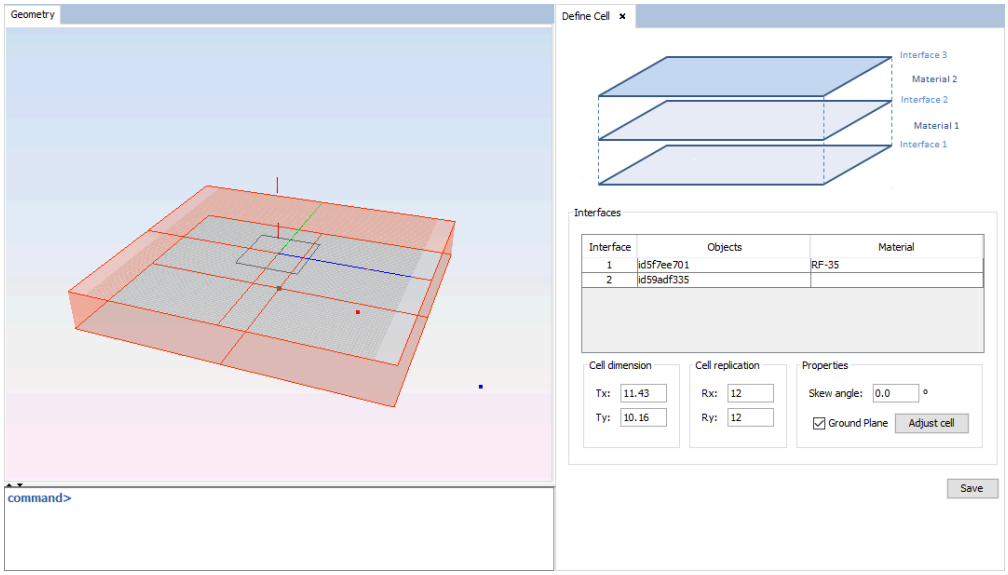


Add a point to add the reference of phase where a ground plane will be placed on the cell definition. Add the point at '0, 0, -1.524'. This value is the thickness for the cell layer.

Step 3: Create the material for the layer (see Materials Menu (Periodical Structures Module)). The name used for the new material is RF-35 with the values Epsilon = (3.54, -0.0018) and Mu = (1.0, 0.0).

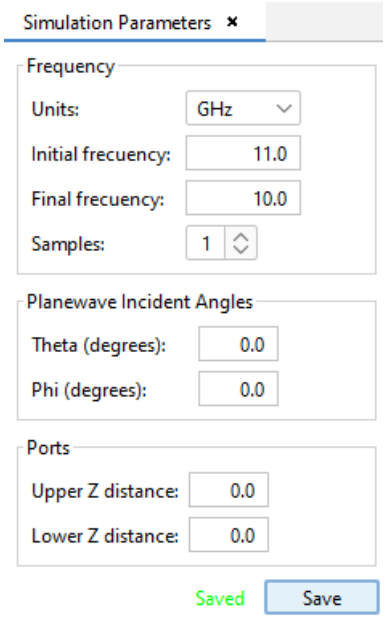
Step 4: Edit the unit cell. Use "Cell --> Define Cell" option for the menu bar. Default, the window takes a size of the cell with the maximum values in X and Y axis, and the first material on the list. Edit the cell as follow:

1. In the table, each row means an interface with it object. The material column refers to the material of the layer above the interface, i.e. the layer between the interface indicate by the row and the next interface. This interfaces are ordered from the lowest Z order to the upper (that can ´t has a material). For the material on interface 1 select 'RF-35' on the list (the material added on the previous step).
2. For the cell dimension select Tx = 11.43 and Ty = 10.16.
3. For the cell replication, and the skew angle, default values are correct.
4. Set the ground plane, selecting the check box. Then, the floor of the cell will be painted on grey.



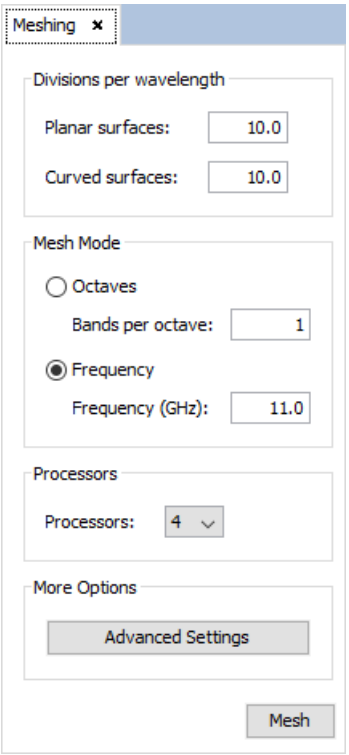
Step 5: Set the simulation parameters using "Simulation --> Parameters" option on the menu bar. This configuration will be identical for both examples.

Set the initial frequency to 11 GHz, the final frequency with default values and the frequency samples set to 1. This options allows to configure a frequency sweep to analyze the cases for different frequencies. For this case only 11 GHz will be analyzed.



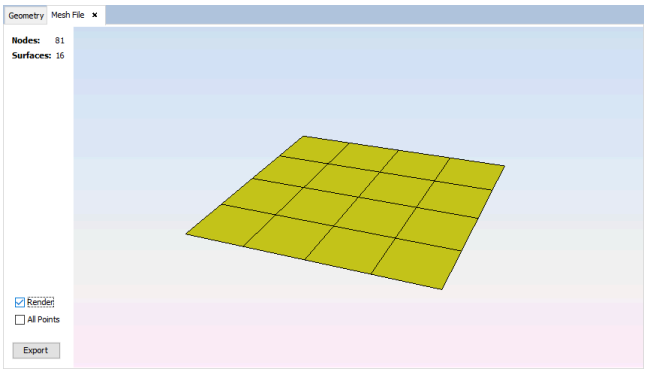
Step 6: Set meshing parameters using "Meshing --> Create Mesh" option on the menu bar.

In this step a meshing with 10 and 20 divisions per wavelength will be compared for the cases with 7 and 37 steps. First, the models will be meshed with 10 divisions per wavelength to the simulation frequency. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed.

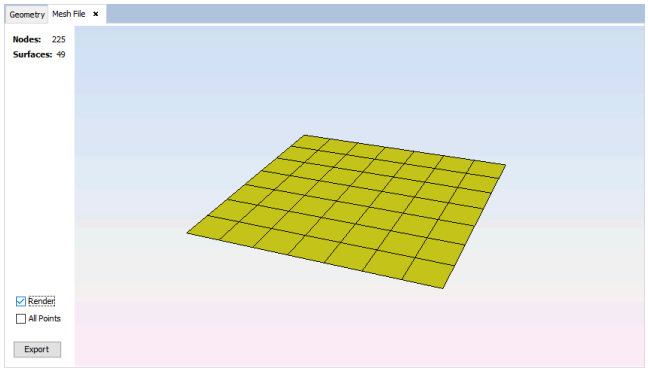


Then, the simulations for each size of the cell component will be executed. When the execution has finished the result meshes can be opened with "Meshing --> Visualize an existing Mesh" option on the menu bar, and selecting the 'fss_1.msh' (mesh of the unitary element) file from the directory of the selected step and frequency. The next images corresponds with the files of the last size ('step6' or 'step36') and the unique frequency ('f0'). On this steps the difference between 10 and 20 divisions is greater than on 'step0' because the size is the biggest on the cases and on the 'step0' the size is low to generate differences.

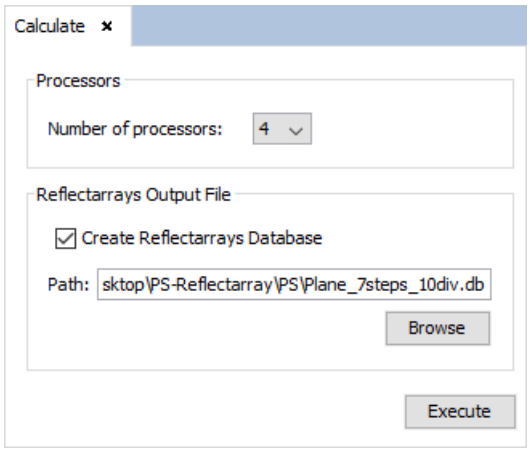
The next image corresponds with the mesh with 10 divisions per wavelength. This result is identical for the cases of 7 and 37 steps.



Then, meshing the structures with 20 divisions per wavelength, the meshes will be larger and the results will be more efficient. The next image corresponds with the mesh with 20 divisions per wavelength. This result is identical for the cases of 7 and 37 steps.



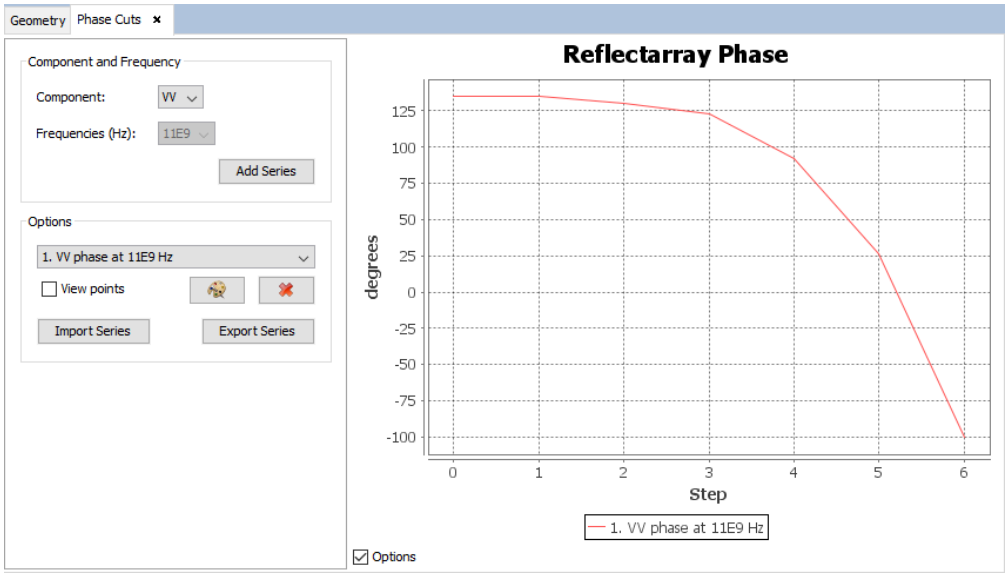
Step 7: Set calculate parameters using "Calculate --> Execute" option on the menu bar. The number of processors to be selected, to improve the efficiency on time, will be the number of the processors of the machine where the example will be executed. Select the path for the database file, previously checked the box to generate it.



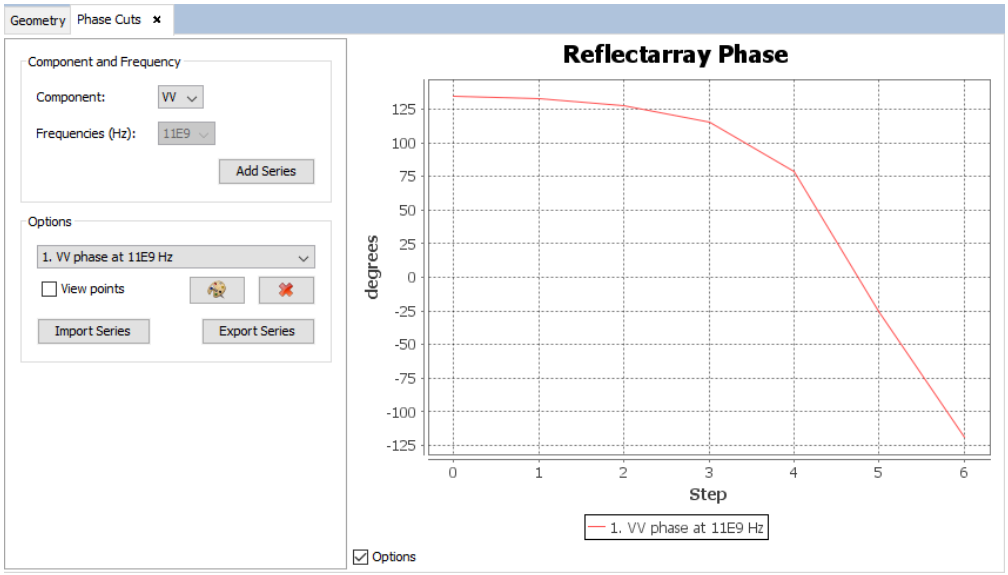
Then, the simulations for each size of the cell component will be executed. When the simulations finish the results will be enabled to display it.

Step 8: Display the information about the phase with "Show Results --> Show Rx/Tx Phase".

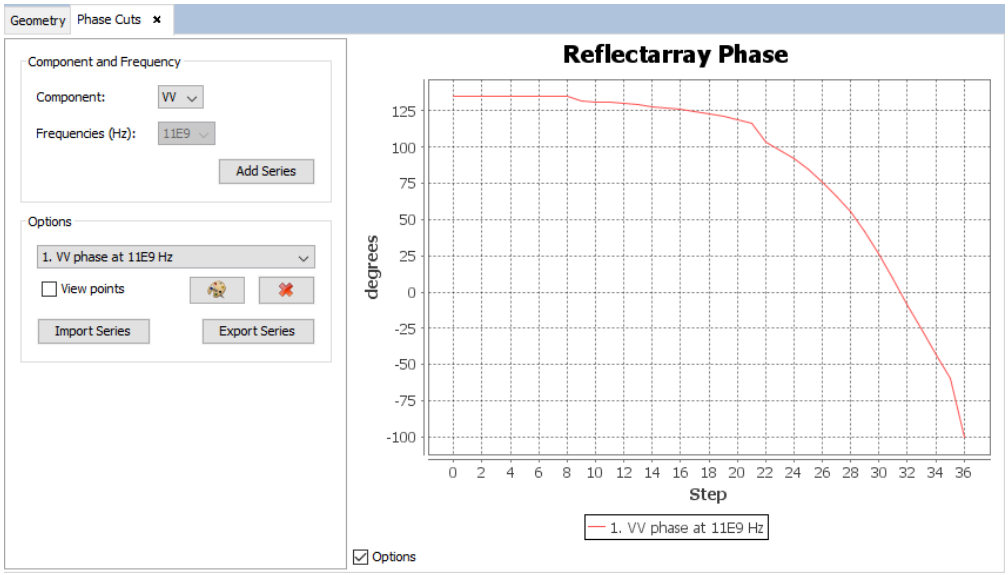
First see the phase for the case with 7 steps and 10 divisions per wavelength.



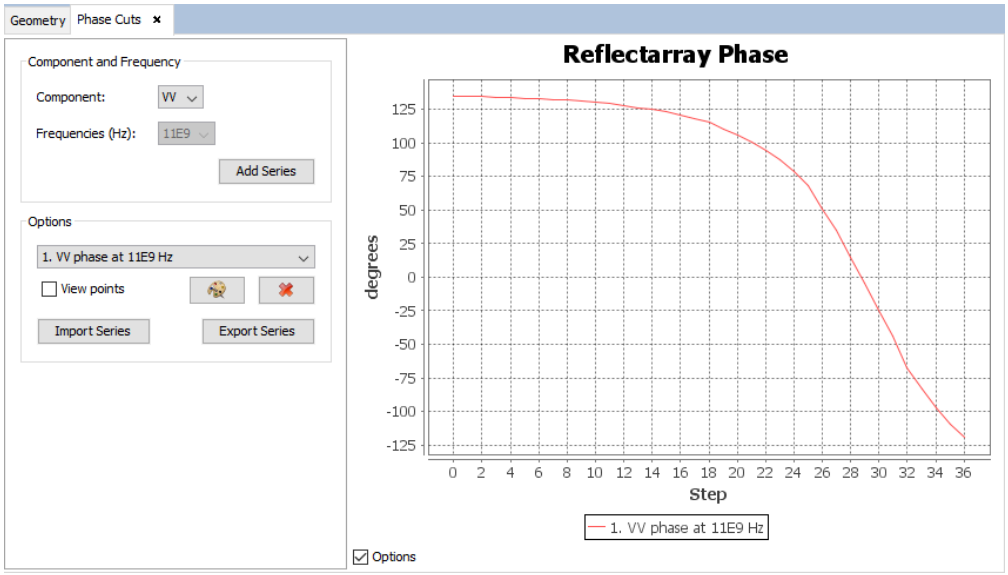
Then see the information for case with 7 steps and 20 divisions.



Now see the case with 37 steps and 10 divisions.



Then see the information for case with 37 steps and 20 divisions.



Analyzing the results we can see that on reflectarray cases with elementary cell of size around millimeters is recommended 20 or more divisions per wavelength. On reflectarray cases, to improve the quality of the results is recommended to have a sufficient number of sizes for the unitary cell.

Parabolic Equation

13

PE is used for the analysis of radio-wave propagation over land and sea for medium and large distances.

This chapter covers the following:

- 13.1 File Menu (p. 1514)
- 13.2 Edit Menu (p. 1515)
- 13.3 View Menu (p. 1516)
- 13.4 Terrain Menu (p. 1517)
- 13.5 Profiles Menu (p. 1520)
- 13.6 Simulation Cases Menu (p. 1524)
- 13.7 Calculate Menu (p. 1527)
- 13.8 Results Menu (p. 1528)
- 13.9 Annex 1: PE Module Commands (p. 1531)
- 13.10 Training Examples (p. 1534)

13.1 File Menu

For information about the File menu, refer to File Menu.

13.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

13.3 View Menu

For information about the View menu, see View Menu.

13.4 Terrain Menu

The Terrain menu contains options that work with terrains. The options are shown in the following figure:

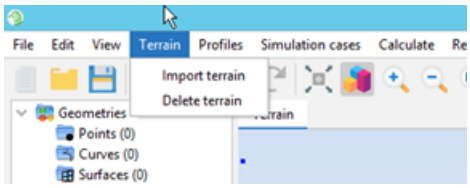


Figure 1820: Terrain menu

The available options are the following:

- **Import terrain**: Allows the user to import a terrain model into the project. Terrain models can be used for extracting the topographic profiles that will be used in simulation cases. The user can import terrain models with the GeoTiff or ARC/INFO ASCII GRID formats.
- **Delete terrain**: Deletes the currently imported terrain model from the project.

13.4.1 Import Terrain

This option allows the user to import a terrain model (with the GeoTIFF or ARC/INFO ASCII GRID) into the project. Note that only one terrain model can be imported at the same time.

When the user selects this option, the following panel will appear:

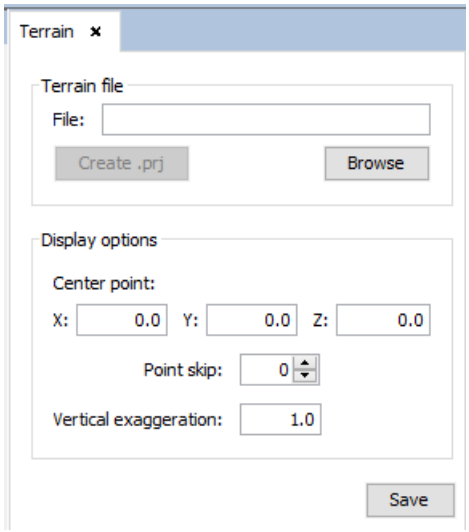
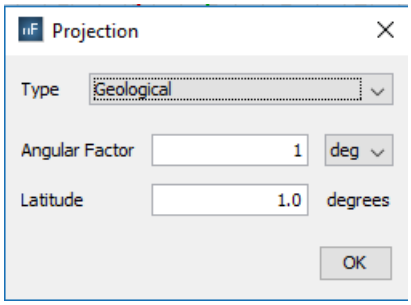


Figure 1821: Import Terrain panel

The user needs to click the "Browse" button to open a file explorer on which the user needs to select the file to be imported.

In case the selected file is ARC/INFO ASCII GRID, additional information is necessary for the proper representation of the terrain. If there is already a PRJ file named as the terrain file, the user can edit it or, if it does not exist, create a new one by clicking the button placed on the left of the browse button. Then the following panel will appear:



This parameters will affect both the visualization of the terrain and the topographic profiles extracted from it:

- **Type**: determines the way the terrain is represented. If it's projected on a plane, **Projection** will be selected, otherwise, if it's represented on the earth's surface, **Geological** must be chosen.
- **AngularFactor**: conversion factor of the angular units. It can be specified for degrees or radians.
- **Latitude**: it's only necessary if the Geological is selected. Specifies the approximate medium latitude of the terrain in degrees.
- **LinearFactor**: it's only necessary if the Projection is selected. Conversion factor of the linear units. It must be specified in meters.

The Display options panel allows the user to control certain options of visualization for the imported terrain. These options only affect the visualization of the imported terrain in the geometry panel and will not impact the topographic profiles created from it:

- **Center point**: Specifies the location at which the terrain will be centered in the geometry panel.
- **Point skip**: In some cases, terrain models will have too many points. This may cause the graphical user interface to lag due to the excessive computational resources needed to render the terrain model. This field indicates how many columns/rows of the terrain model are skipped between two successively rendered columns/rows. For instance, a value of 0 indicates that all points will be rendered, while a value of 2 indicates that the GUI will render one column/row of points, ignore the following two columns/row, render the next one and so on.
- **Vertical exaggeration**: Sometimes, the area of the terrain model is much greater than the variations of height of the same terrain model, making those variations of height unappreciable. This field allows the user to specify how many times the vertical features of the terrain model are amplified, so visually it's easier to identify the relief of the terrain model.

When the above parameters have been set, the user can press the "Save" button to import the terrain. The terrain model will be shown in the geometry panel.

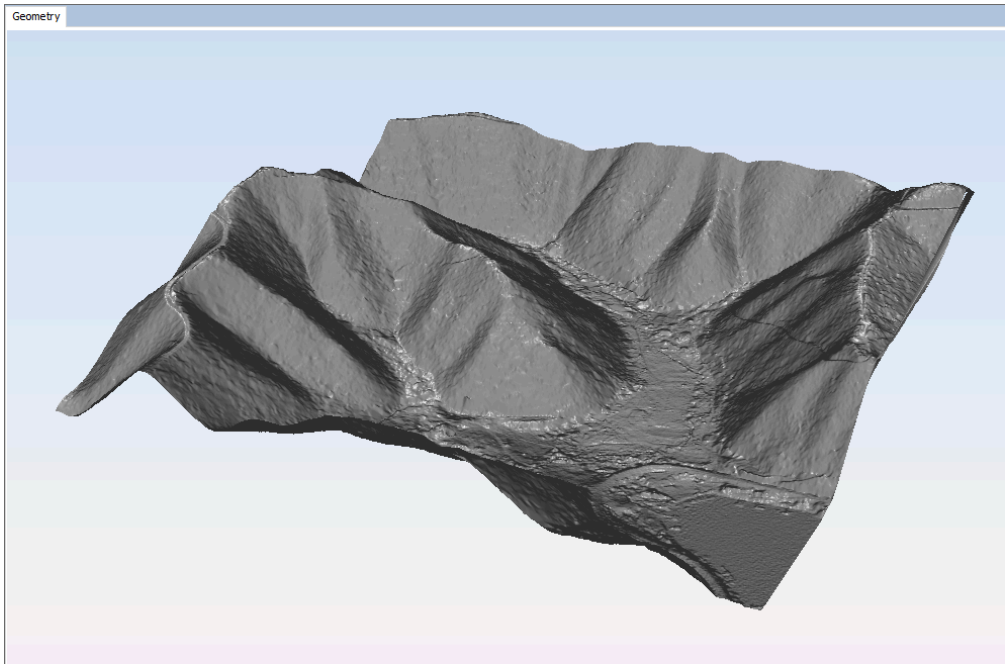


Figure 1822: Terrain visualized in the Geometry Panel

13.4.2 Delete Terrain

This option removes the currently imported terrain model from the project, if there is any. After deleting a terrain model, the user may import another one by using the "Import terrain" option under the "Terrain" menu.

13.5 Profiles Menu

This option allows the user to manage the topographic profiles that are contained in the project. A topographic profile can be created by using any of the following methods:

- By extracting them from the loaded terrain model.
- By importing a text file that contains the points (distance and height) of the topographic profile.
- By creating a topographic profile manually using the editor.

The first method of creating profiles has been explained in the previous section. It is possible to create a topographic profile from a text file or using the editor by using the options present in the "Profiles" menu.

The following figure shows the available options under this menu:

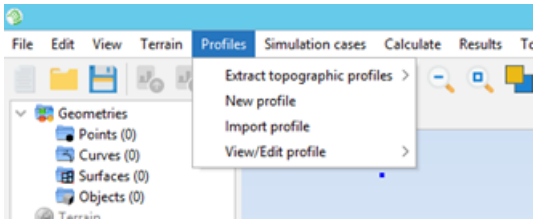


Figure 1823: Profiles menu

The available options are the following:

- **Extract topographic profiles**: Allows the user to create new topographic profiles from the currently imported terrain model.
- **New profile**: Creates a new (empty) topographic profile.
- **Import profile**: Creates a topographic profile from the contents of a text file with the points of the topographic profile. The format of the text file needs to be:
  - First, any number of lines starting with "#" that will be ignored by the parser. These lines may be used as comments.
  - Then, any number of lines, each containing a point of the topographic profile. Each of these lines must specify the distance from the origin (in meters) and the height of the point (in meters). Both values must be separated by any number of spaces.
- **View/Edit profile**: Under this submenu, all the topographic profiles of the project will be listed. Clicking on any of them will allow the edition of the profile.

Selecting any of these options will open the **Topographic Profiles Editor**, which will allow the user to edit the new or existing topographic profile.

13.5.1 Extract Topographic Profiles

This submenu shows two options that allow the user to create topographic profiles from an imported terrain model. The options under this submenu are the shown in the following figure:

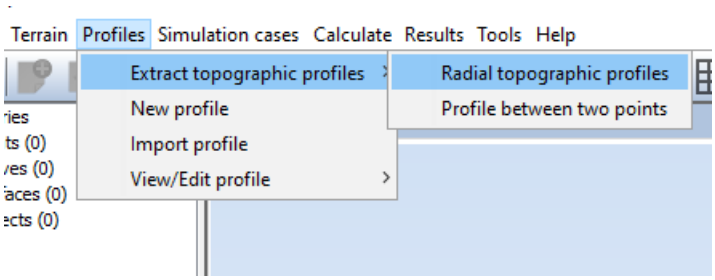


Figure 1824: Extract Topographic Profiles submenu

The available options for extracting topographic profiles from a terrain model are:

- **Radial topographic profiles**: Allows the user to extract multiple topographic profiles by choosing a starting point (also referred to as "the center"), and a radius (which will be the length of each topographic profile). The obtained profiles will match the radii of the circle with the specified radius and center point.
- **Profile between two points**: Allows the user to extract a single topographic profile between two points of the terrain model.

Selecting any of these options will call the matching command, see Annex 1: PE Module Commands for more details.

13.5.2 Topographic Profile Editor

The Topographic Profile Editor allows the user to create new topographic profiles or edit the existing ones. The Topographic Profile Editor will be opened when clicking any of the options under the "Profiles" menu.

The Topographic Profile Editor is shown in the following figure:

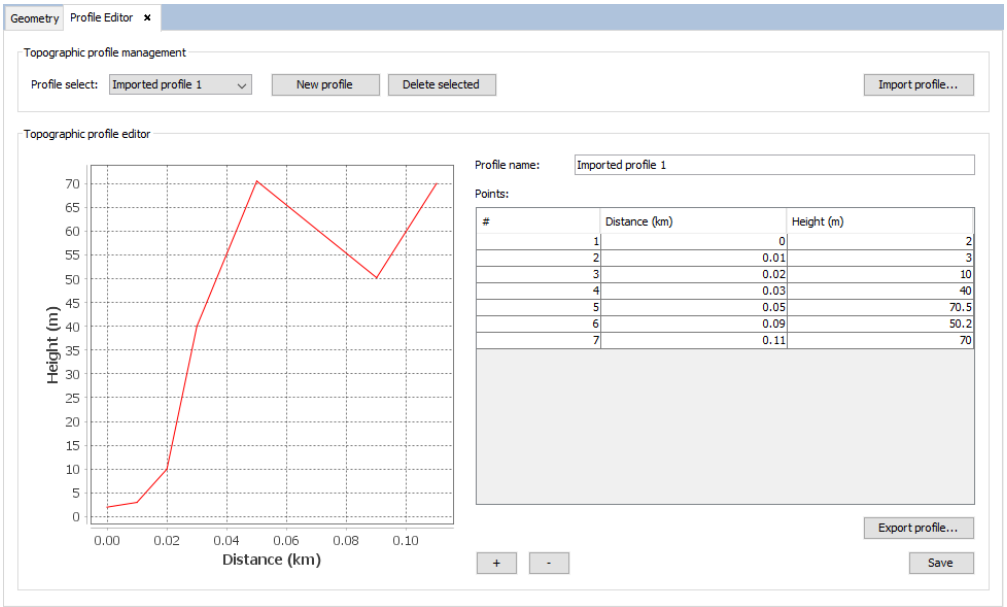


Figure 1825: Topographic Profile Editor

In this panel, it is possible for the user to manage the topographic profiles of the project. The "Topographic profile management" panel allows the user to manage the list of topographic profiles of the project, allowing the following actions:

- Select a profile so it is loaded into the editor (allowing further actions on it such as adding/ removing points, modifying points or exporting the profile).
- Create a new (empty) profile.
- Delete the selected profile.
- Import a profile from a text file. The required format is the same as when importing the topographic by using the "Import profile" option under the "Profiles" menu:
  - First, any number of lines starting with "#" that will be ignored by the parser. These lines may be used as comments.
  - Then, any number of lines, each containing a point of the topographic profile. Each of these lines must specify the distance from the origin (in meters) and the height of the point (in meters). Both values must be separated by any number of spaces.

The "Topographic profile editor" panel allows the user to visualize the selected topographic profiles, as well as performing the following actions on the selected profile:

- Changing the name of the topographic profile.
- Editing the height of the points of the topographic profile in the table of points.
- Exporting the points of the topographic profile to a text file. The format of the resulting text file is the same format required to import a topographic profile from a text file (explained above).
- Adding a new point to the topographic profile by using the "+" button. When doing this, the program will ask the user the distance (in km) where the point is located. The new point will be inserted into the table (note that points will be sorted by distance), allowing the user to edit its height.

- Removing a point from the topographic profile. To do this, the user needs to select the row corresponding to the point they want to remove and press the "-" button.

When the user has finished editing a topographic profile, they need to press the "Save" button to confirm the changes.

13.6 Simulation Cases Menu

The options under this menu allows the user to manage the list of simulation cases of the project. The following figure shows the contents of the "Simulation cases" menu:

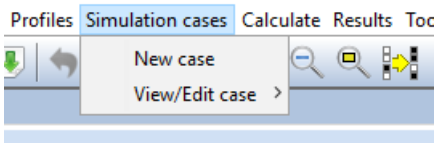


Figure 1826: Simulation Cases menu

The available options under this menu are the following:

- **New case** Allows the user to create a new (empty) simulation case.
- **View/Edit case** This submenu will list the simulation cases of the project. Clicking on any of them allows to edit the parameters of the case.

Any of these options will open the Simulation Case Editor to edit the parameters of the new or existing simulation case.

13.6.1 Simulation Case Editor

The Simulation Case Editor allows the user to manage the list of simulation cases of the project, as well as changing the parameters of already existing simulation cases.

The Simulation Case Editor consists on two different tabs:

- **Refraction Profiles** This tab allows the user to create and edit the refraction profiles that will be used in the simulation case. A refraction profile specifies how the refraction index of the air varies along with the height.
- **Case Definition** This tab allows the user to configure the numerical parameters of the simulation case, such as the parameters of the antenna and the spatial bounds of the simulation. In this tab, the user also needs to set the topographic profile used for the simulation case, as well as the assignation of the refraction profiles created in the previous tab to the different range values.

In addition, the top of the Simulation Cases Editor (shown in the next figures of this section) allows the user to create new simulation cases or delete already existing ones. The combobox located at the top of the panel allows the selection of the simulation cases that is going to be edited, and whose contents are shown in the Simulation Cases Editor. When the user finishes the edition of the parameters of the simulation case, they need to press the "Save Simulation Case" button located on the bottom of the panel.

The options in each of the tabs of the Simulation Cases Editor will be explained next.

Refraction Profiles Tab

The contents of this tab are shown in the following figure:

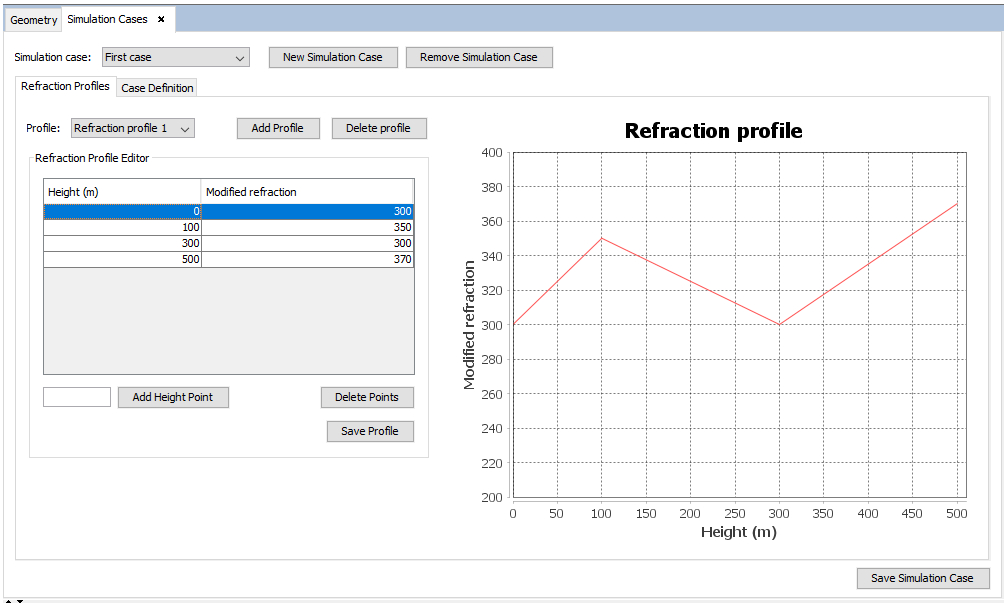


Figure 1827: Refraction Profiles Tab of the Simulation Case Editor

In this tab, it is possible to manage the list of refraction profiles contained in the simulation case. Therefore, it is possible to create new profiles and delete the selected profile. When selecting a refraction profile in the combobox, the table will show the points of the refraction profile, each composed of the height (in meters) and the modified refraction (M).

It is possible to add new points to the refraction profile by entering the height value in the field next to the "Add Height Point" button and then pressing this same button. The points of the refraction profile will be ordered in the table by height value. The chart on the right will be updated automatically to represent the values of the refraction points of the table.

When the user has finished editing the points of the refraction profile, they must press the "Save Profile" button to confirm the changes.

Case Definition Tab

The contents of this tab are shown in the following figure:

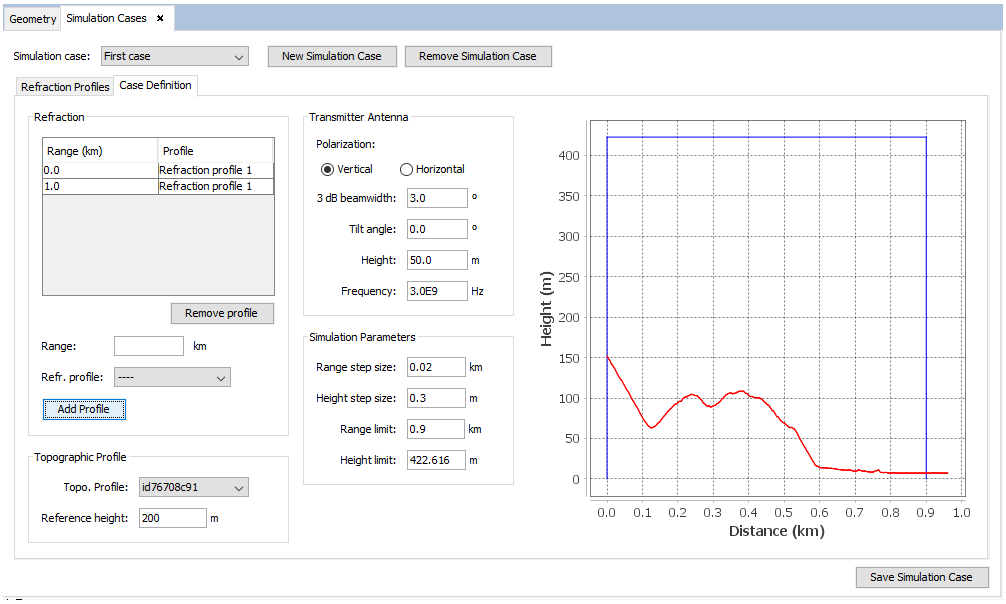


Figure 1828: Case Definition Tab of the Simulation Case Editor

This tab allows the user to define the parameters of the simulation case:

- The **Refraction** panel allows the user to specify the refraction profiles that will be used in the simulation case. Each refraction profile is assigned a range, which is the distance from the origin the profile will take effect at. It is recommended to assign refraction profiles to ranges so that the refraction profiles span the whole range interval of the simulation (from 0.0 to the value specified in the "Range limit" field). To add a refraction profile, the user needs to enter the range in the "Range" field (in km), select the refraction profile in the drop-down list just below and press the "Add Profile" button.
- The **Topographic Profile** panel allows the user to specify the topographic profile used for the simulation (depicted in red on the chart on the right side). The "Reference Height" panel allows the user to shift the Z-coordinate (or height coordinate), so that height 0 will be located in another height value. It is recommended to set this value to be close to (but not greater than) the minimum height of the topographic profile used.
- The **Transmitter Antenna** panel allows the user to specify the parameters of the antenna of the simulation (the antenna will always be located at range = 0). The parameters that can be setup are the polarization of the antenna, the half power beamwidth angle, the height (above the terrain at range = 0) and the frequency.
- The **Simulation Parameters** panel contains the parameters of the SSPE (Split-Step Parabolic Equation) algorithm. On the one hand, it allows to specify the range and height increments that the algorithm will use to compute the fields. On the other hand, it allows to specify the computational domain of the simulation with the "Range limit" and "Height limit" fields. These bounds are depicted in blue on the chart.

13.7 Calculate Menu

This menu contains only one option "Execute", which allows the execution of the simulation cases added to the project. When the user selects the "Execute" option, the following panel is shown:

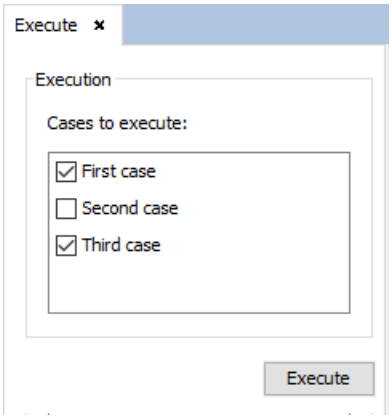


Figure 1829: Execute panel

This panel allows the user to select which of the cases of the project will be simulated. Only the simulation cases with a checked mark next to them will be simulated. The user can click on the name of a simulation case to toggle the state of the case.

To perform the simulation, the user needs to click the "Execute" button. This will open the Process Log, which will show the state of the simulation.

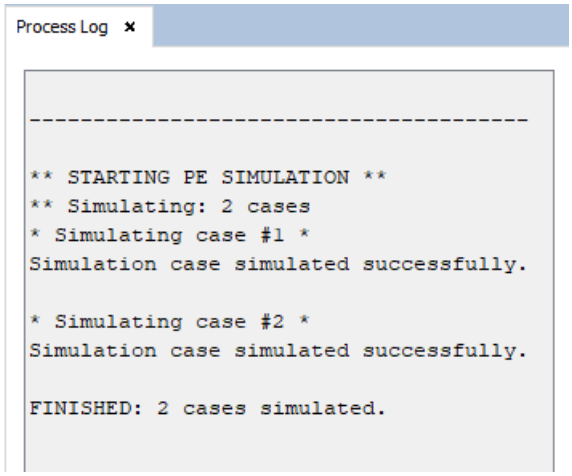


Figure 1830: Process Log

13.8 Results Menu

This menu allows the user to visualize the computed results for the executed simulation cases. The contents of this menu are shown in the following figure:

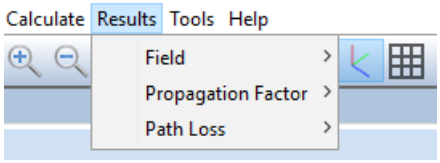


Figure 1831: Results menu

- The **Field** submenu contains options to visualize the field at a given height value or range value.
- The **Propagation Factor** submenu contains options to visualize the propagation factor, that is, the strength of the field relative to the free-space value, at a given height or range value.
- The **Path Loss** submenu contains options to visualize the path loss, that is, the ratio between radiated power and power available at a point, at a given height or range value.

13.8.1 Visualizing Results

For each possible result type (field, propagation factor and path loss), there are three possible visualization options show the results at a given height for every possible value of range, show the results at a given range for every value of height, or show a color map of the value of the results of all the profile. The options under each submenu of the "Results" menu allow to choose between the three possibilities:

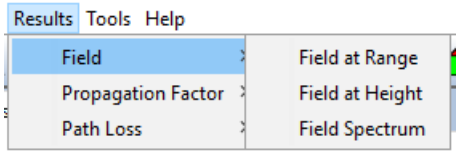


Figure 1832: Visualization options

By Height or Range

When any 'at Range' or 'at Height' option is selected, a new panel will be opened and will look like the one in the following figure:

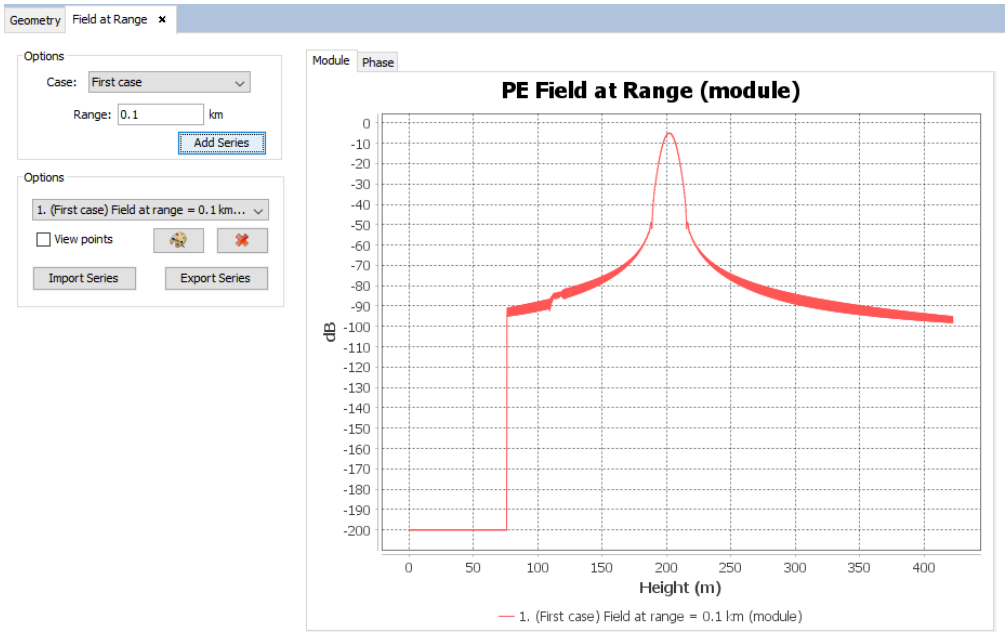


Figure 1833: PE Field at Range panel

In this panel, it is possible to select the simulation case the user wants to show the results for, as well as the range or height value, depending on the visualization mode. To add a new series, the user needs to press the "Add Series" button after selecting the simulation case and range/height value.

In addition, the user can manage the added series by using the "Options" panel. In this panel, the user can remove series, change the color of a series and import/export series from/to text files.

Color Map

If any of the 'Spectrum' options is selected a panel like this will appear:

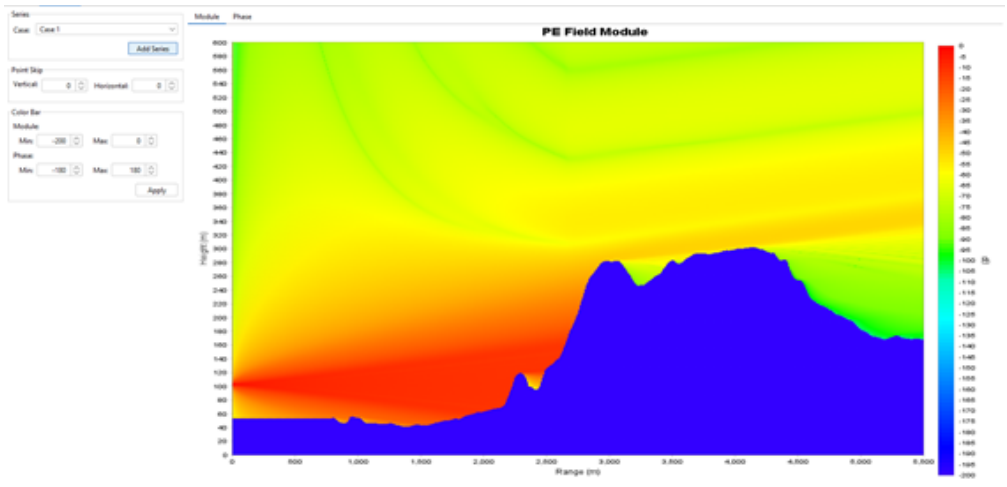


Figure 1834: PE Prop. factor Spectrum

In this panel, only one result set can be shown at once. To add or change a set, the user must select the appropriate case, select the point skip, explained in the import terrain section and press the "AddSeries" button.

In the Color Bar section, the user can define the range of values represented by the map. Any values lower or higher than the limits will be represented with the same color as the correspondent limit. These values can be changed before of after adding the values. The module or the phase can be changed. To change it afterwards, click "Apply".

If the user wants to see a more detailed view of the values at range or height, pressing right click anywhere in the graph will show a pop-up menu with those options.

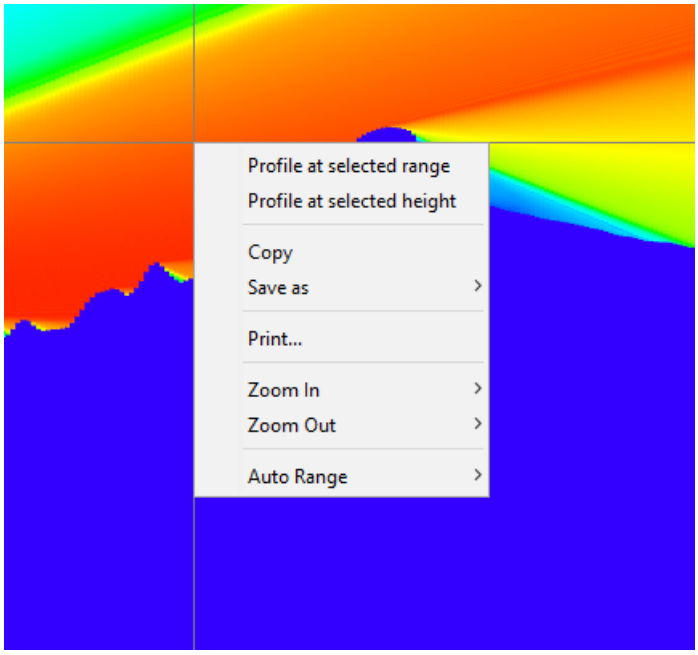


Figure 1835: Spectrum Pop-Up Menu

Selecting any of those options will add the series to a panel like the one described above at the range or height where the menu was triggered.

13.9 Annex 1: PE Module Commands

The Parabolic Equation module includes several commands for use in the command-line that can be used for managing terrains and creating topographic profiles. In this section, these commands will be detailed.

13.9.1 importTerrain command

Description This command is used to import a terrain model from a GeoTIFF or ARC/INFO ASCII GRID file.

Usage When called without arguments, it will open the same tab as the menu option to continue the process.

The command can also be called with command line arguments. In this case, it will need to be called the following way:

importTerrain -p "path to file" centerX centerY centerZ verticalExaggeration pointSkip

These parameters are the same shown in the Import Terrain panel when accessing this function from the menu:

- Center Specifies the location at which the terrain will be centered in the geometry panel.
- **Point skip** In some cases, terrain models will have too many points. This may cause the graphical user interface to lag due to the excessive computational resources needed to render the terrain model. This field indicates how many columns/rows of the terrain model are skipped between two successively rendered columns/rows. For instance, a value of 0 indicates that all points will be rendered, while a value of 2 indicates that the GUI will render one column/row of points, ignore the following two columns/row, render the next one and so on.
- **Vertical exaggeration** Sometimes, the area of the terrain model is much greater than the variations of height of the same terrain model, making those variations of height unappreciable. This field allows the user to specify how many times the vertical features of the terrain model are amplified, so visually it's easier to identify the relief of the terrain model.

In case the file is Ascii Grid (.asc), 3 extra parameters are necessary. The command must be called as shown:

importTerrain -p "path to file" projectionType angularFactor (latitude | linearfactor) centerX centerY centerZ verticalExaggeration pointSkip

- **ProjectionType** Type of representation of the terrain. It can be projected on a plane (p) or represented on the earth's surface (g).
- **AngularFactor** conversion factor of the angular units. It must be specified in degrees.
- **Latitude** In case it is a geological representation Medium latitude of the location of the terrain.
- **LinearFactor** In case it is a projection conversion factor of the linear units. It must be specified in meters.

13.9.2 deleteTerrain command

Description This command deletes the currently loaded terrain.

Usage The command does not need any arguments.

13.9.3 profileBetweenPoints command

Description Creates a topographic profile between two different points of the loaded topographic profile.

Usage When called without arguments, the command will ask for the following parameters in order:

- Geometrical coordinates (x, y) of the first point of the terrain, where the starting point of the topographic profile will be located.
- Geometrical coordinates (x, y) of the second point of the terrain, where the final point of the topographic profile will be located.

In order to easily determine these two points, the user can click on a point of the loaded terrain model in the geometry panel, and the point will automatically be filled in the console.

When called with arguments, the command must be called the following way:

```
profileBetweenPoints -n profile_name -p x0 y0 x1 y1
```

Where "profile_name" is the name given to the created profile, (x0, y0) are the coordinates of the first point and (x1, y1) are the coordinates of the second point.

13.9.4 radialProfiles command

Description Creates multiple topographic profiles, all starting at the same point (the center point) and with the same distance (the radius length). The created profiles with match the radii of a circle centered at the specified point and with the given radius length.

Usage When called without arguments, the command will ask for the following parameters in order:

- Geometrical coordinates (x, y) of the center point. The user can click on a point of the terrain model to fill this value automatically.
- Radius length (i.e. profile length). The user can click on a point of the terrain model to calculate the radius between the clicked point and the center point.
- The initial angle. Angles will be measured from the positive X axis and go up counter-clockwise. At this angle will be the radius of the first topographic profile.
- The final angle. At this angle will be the radius of the last topographic profile.
- Number of topographic profiles that will be taken. This number must be of two or greater.

The result of this command will be as many topographic profiles as specified, each one matching a radius of the circle defined by the center point and radius length, and each profile is sampled at constant angle intervals between the initial angle and the final angle.

When called with arguments, the command must be called the following way:

```
radialProfiles -n profile_name_prefix -p xCenter yCenter radius thetaStart thetaEnd
 numProfiles
```

The names of the topographic profiles will have the prefix given by "profile_name_prefix" and will be suffixed with a "_" followed by the number of the profile sample (1, 2, etc.). The rest of parameters are the same as in the interactive version of the command.

13.10 Training Examples

13.10.1 Example 1: Propagation Factor on an Existing Terrain

In this example, the propagation factor of a simple antenna is calculated and visualized.

Step 1

Start newFASANT.

Step 2

Select File and click on New.

Step 3

Select PE.

Step 4

Click on Terrain→Import Terrain, and a new tab will appear.

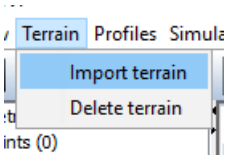
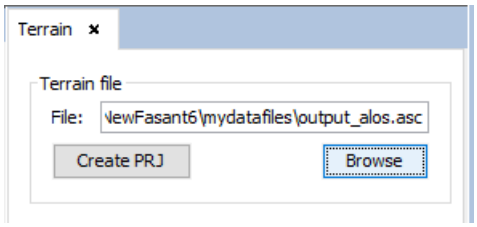


Figure 1836: Import Terrain Menu

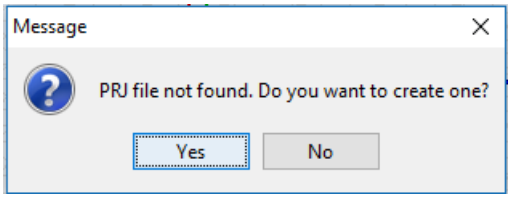
Step 5

In the Import Terrain tab click browse and select the desired file.



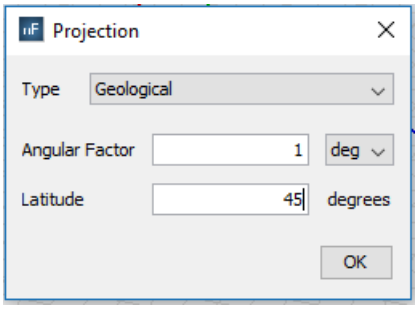
Step 6

As the file is `.asc` it's necessary to have a projection file. If it does not exist, clock OK in the pop-up window that will appear. Else, click on the **Edit PRJ** button next to browse.



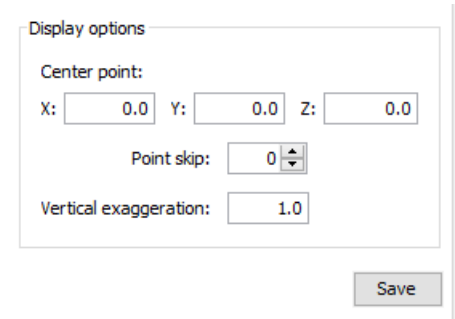
Step 7

Select the type Geological, factor of 1 degree and 45º of latitude. Then click OK.



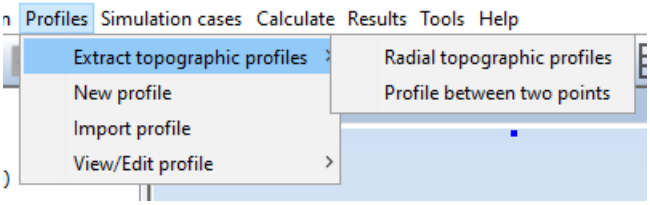
Step 8

Introduce the display options and click Save,



Step 9

When the terrain appears, click Profiles→Extract topographic profiles→Profile between two points.



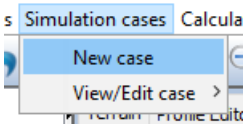
Step 10

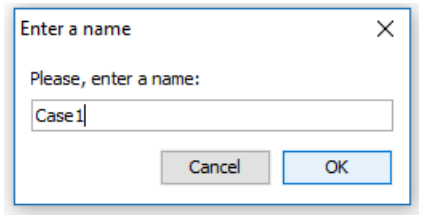
In the console, write the point "-3700 1600" and press enter, then do the same for the point "1100 -1600". This points can also be selected on the terrain.

```
command> profileBetweenPoints
Select start point [x y]: -3700 1600
Select end point [x y]: 1100 -1600
```

Step 11

Select Simulation cases→New case. Click New Simulation Case. Then choose a name and press OK.



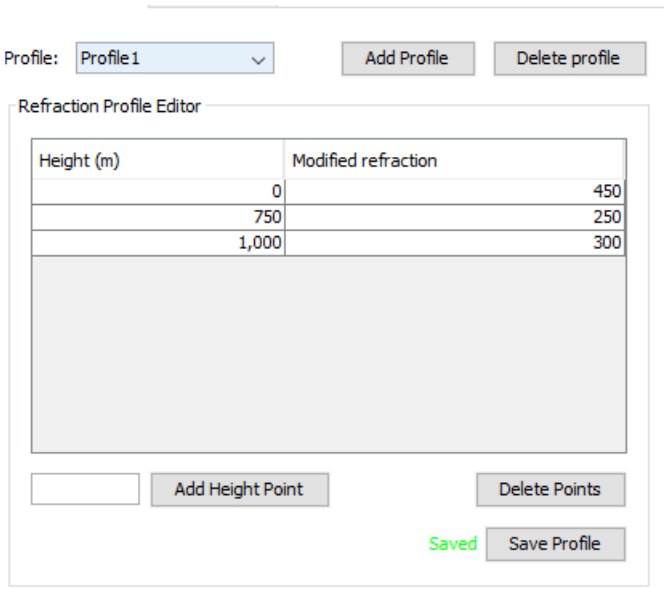


Step 12

Click Add profile and choose a name.

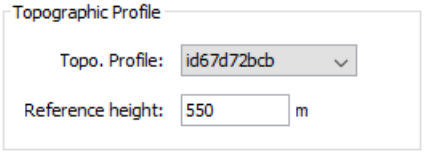
Step 13

Add height points at 750 and 1000 meters and then change the refraction to match the image below. Then save the profile.



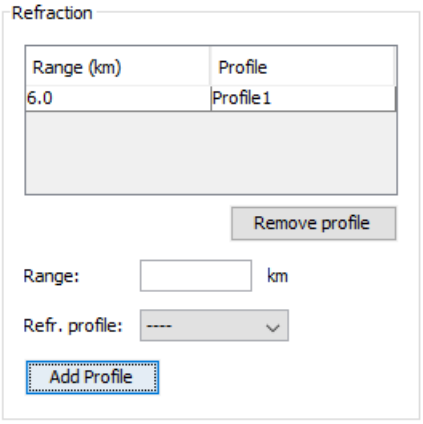
Step 14

Switch to the Case Definition tab. In the Topographic Profile section, choose the profile and set the reference height to 550 m.



Step 15

In the refraction section, select 6 km of range and the created refraction profile and press add.

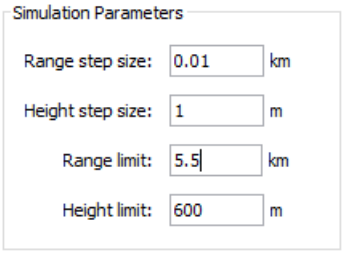


Step 16

Set the antenna parameters. In this case we will use the default values.

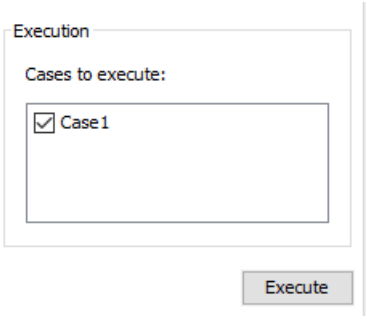
Step 17

In the simulation parameters, set the range step size to 0.01 km, the height step size to 1m, the range limit to 5.5 km and the height limit to 600. Then save the simulation case.



Step 18

Select Calculate→Execute on the menu bar. Select the case to process and press execute.

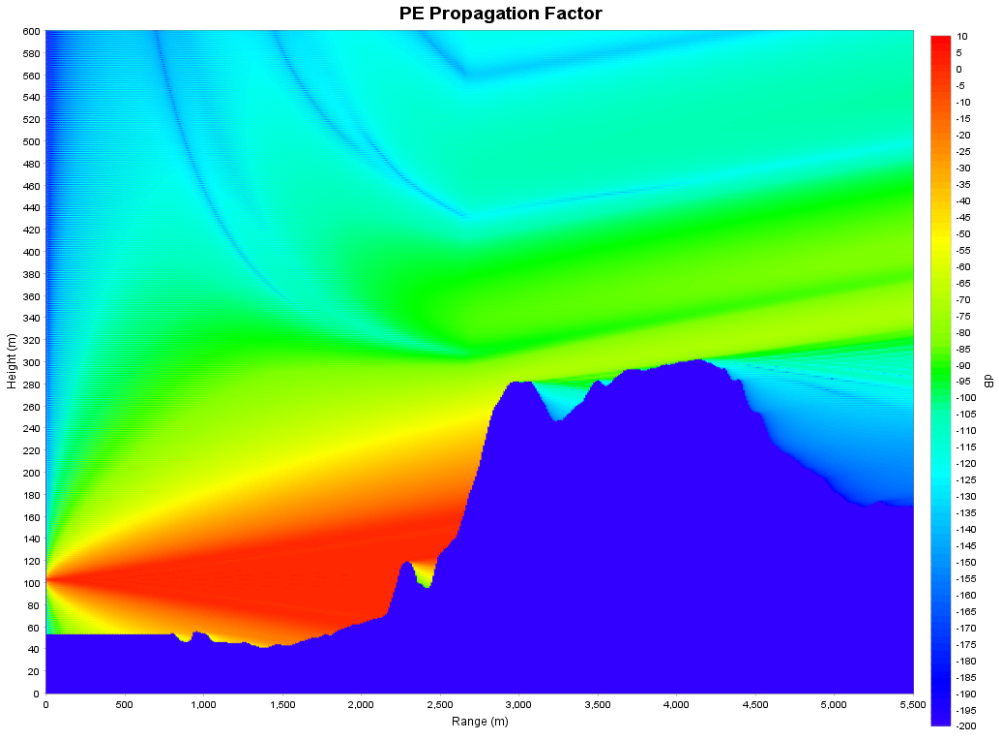


Step 19

When the simulation finishes we can visualize the simulation results. Click on Results→Propagation Factor→Propagation Factor Spectrum.

Step 20

Select the case and press add series.

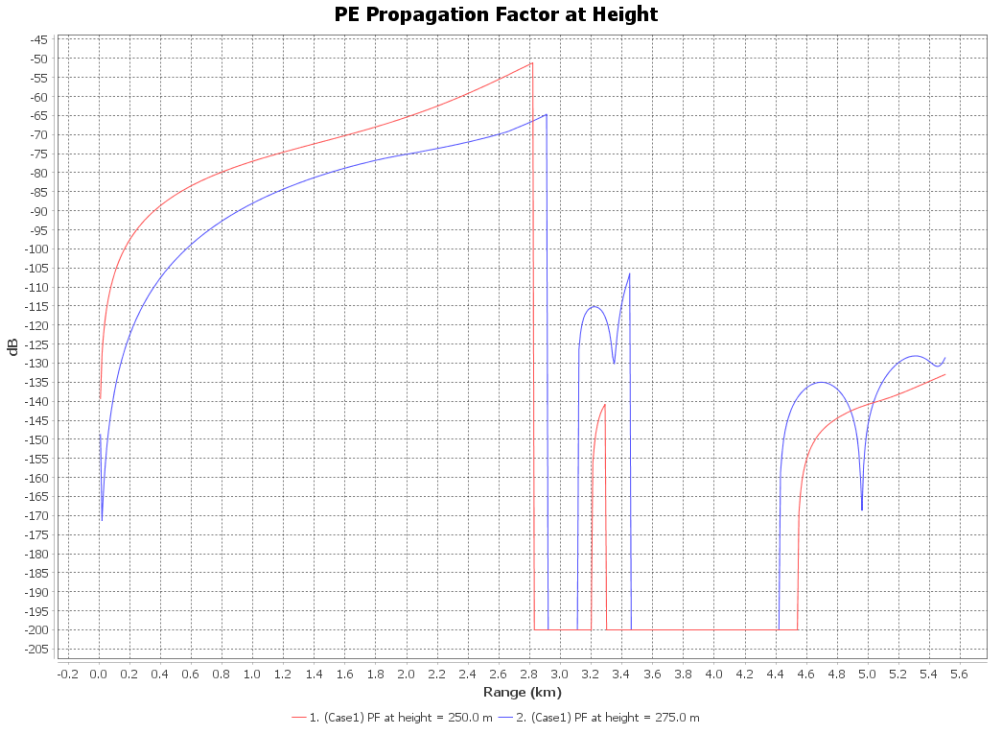


Step 21

Right click on any point at 250 m of height and select **Profile at selected height**.

Step 22

In the chart window add a series at 275 m to compare both values.


PE Propagation Factor at Height
dB
-45
-50
-55
-60
-65
-70
-75
-80
-85
-90
-95
-100
-105
-110
-115
-120
-125
-130
-135
-140
-145
-150
-155
-160
-165
-170
-175
-180
-185
-190
-195
-200
-205
-0.2 0.0 0.2 0.4 0.6 0.8 1.0 1.2 1.4 1.6 1.8 2.0 2.2 2.4 2.6 2.8 3.0 3.2 3.4 3.6 3.8 4.0 4.2 4.4 4.6 4.8 5.0 5.2 5.4 5.6
Range (km)
1. (Case1) PF at height = 250.0 m
2. (Case1) PF at height = 275.0 m

Ultrasound

14

Ultrasound is used for the 3D analysis of ultrasound emitting sources and time delay response simulation.

This chapter covers the following:

- 14.1 File Menu (p. 1541)
- 14.2 Edit Menu (p. 1542)
- 14.3 View Menu (p. 1543)
- 14.4 Geometry Menu (p. 1544)
- 14.5 Materials Menu (p. 1545)
- 14.6 Simulation Menu (p. 1546)
- 14.7 Source Menu (p. 1551)
- 14.8 Output Menu (p. 1556)
- 14.9 Meshing Menu (p. 1567)
- 14.10 Calculate Menu (p. 1568)
- 14.11 Show Results Menu (p. 1569)
- 14.12 Tools Menu (p. 1587)
- 14.13 Help Menu (p. 1588)
- 14.14 Training Examples (p. 1589)
- 14.15 Annex 1: DUS File Format (p. 1598)
- 14.16 Annex 2: Closed-Loop Simulation Support in the US Module (p. 1600)

14.1 File Menu

For information about the File menu, refer to File Menu.

14.2 Edit Menu

For information about the Edit menu, refer to Edit Menu.

14.3 View Menu

For information about the View menu, see View Menu.

14.4 Geometry Menu

For information about the Geometry menu, refer to Geometry Menu.

14.5 Materials Menu

For information about the Materials menu, refer to Materials Menu.

14.6 Simulation Menu

The first step when designing a US simulation case is to configure the simulation parameters. The options under the "Simulation" menu allows us to do so. The available options are shown in the following figure.

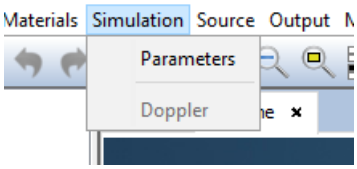


Figure 1837: Simulation menu

14.6.1 Parameters

When the user selects the "Parameters" option under the "Simulation" menu, the following panel will appear:

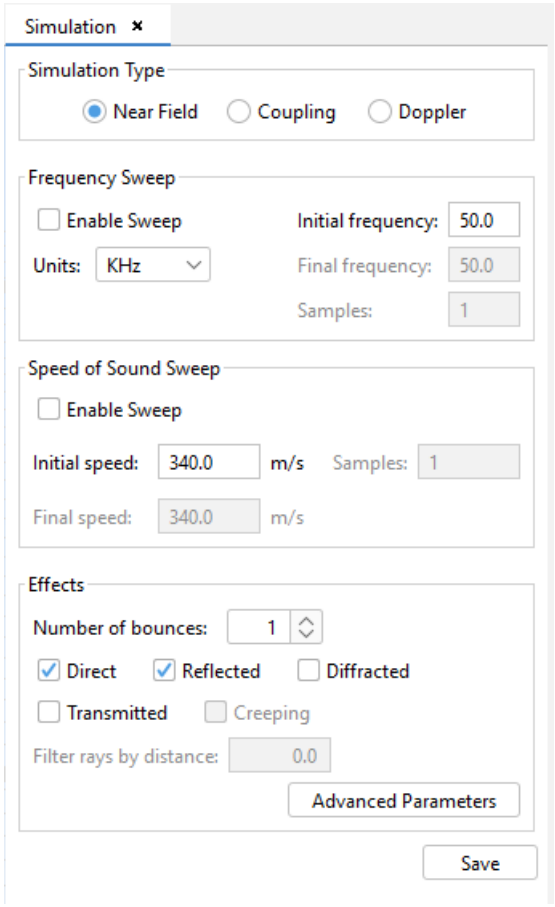


Figure 1838: Simulation parameters panel

This panel allows the user to select the **simulation type** for the current case. The "Near Field" simulation type consists on the computation of the sound pressure field in a set of points known as "observation points". The "Coupling" simulation type allows the calculation of coupling values between a set of sources and a passive receptor. The "Doppler" simulation type takes into account the relative speed and rotation of objects, observation points and sources to calculate the Doppler Shift.

The "Frequency Sweep" panel allows the configuration of the **frequency** of sound waves considered for the simulation. There are two possibilities:

- **Single frequency** When the "Enable Sweep" check box is unchecked, the results will be computed only for the frequency specified in the "Initial frequency" field.
- **Frequency sweep** When the "Enable Sweep" check box is checked, the results will be computed for a set of frequencies bounded by the specified values in the "Initial frequency" and "Final frequency" fields. The number of frequencies used for the frequency sweep is given in the "Samples" field. The sampled frequencies in the frequency sweep will be evenly-spaced.

The "Speed of Sound Sweep" panel allows the configuration of the **speed of the sound waves** in the simulation. As with the frequency, there are two options. If the "Enable Sweep" check box is unchecked, only the speed given in the "Initial speed" will be used in the simulation. Otherwise, the user can specify a range of speeds to compute the results for, as well as the number of samples within that range that will be used.

The sound pressure field is computed by taking into account the multiple paths the sound waves make in their way from the ultrasound source to each observation point. The "Effects" panel allows the user to set up which **effects** are considered when computing these paths. The available options are the following:

- **Number of bounces**: The maximum number of bounces considered in path calculation. Paths which exceed the number of bounces specified here are not computed.
- **Direct**: If this option is selected, the simulation will be performed taking into account direct rays.
- **Reflected**: If this option is selected, the simulation will be performed taking into account reflected rays.
- **Diffracted**: Determines whether diffracted rays are computed.
- **Creeping**: Determines whether creeping rays are computed. This option isn't available at the moment.
- **Transmission**: Determines whether transmitted rays are computed.

The **Filters rays by distance** is only enabled when the coupling option is set up. It determines the minimum distance from which the rays will be considered for the coupling calculation.

The **Advanced Parameters** button allows the user to configure parameters that are closer to the computation algorithm. When the user presses this button, the following dialog will appear:

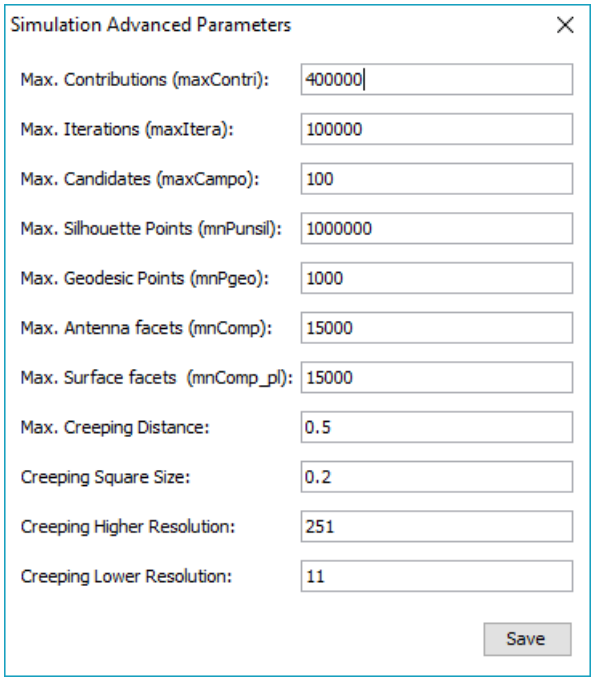


Figure 1839: Simulation advanced parameters dialog

All these parameters are detailed below:

- **Max. Contributions** maximum number of contributions to taken into account. A contribution is for instance that due to a given double ray-path.
- **Max. Iterations** maximum number of contributions that are taken into account.
- **Max. Candidates** maximum number of surfaces that can shadow other surfaces.
- **Max. Silhoutte Points** maximum number of points for the silhoutte when creeping wave paths are computed.
- **Max. Geodesic Points** maximum number of geodesic points when computed a creeping wave path.
- **Max Antenna facets** maximum number of facets to be stored in the anxel of Z-Buffer matrix of the source.
- maximum number of facets to be stored in the anxel of Z-Buffer matrix of every surface.
- **Max Creeping Distance** maximum distance between a surface and the antenna to increase the creeping sample resolution.
- **Creeping Square Size** size edge of the square where the creeping sample resolution is increased near the antenna.
- **Creeping Higher Resolution** higher sampling resolution per surface used to compute the creeping silhoutte.
- **Creeping Lower Resolution** lower sampling resolution per surface used to compute the creeping silhoutte.

After setting up the simulation parameters, the user may press the Save button to save the configuration.

14.6.2 Doppler

If the Doppler simulation type has been selected in the Simulation Parameters panel, the Doppler option will be available. When this option is selected, the following panel will appear:

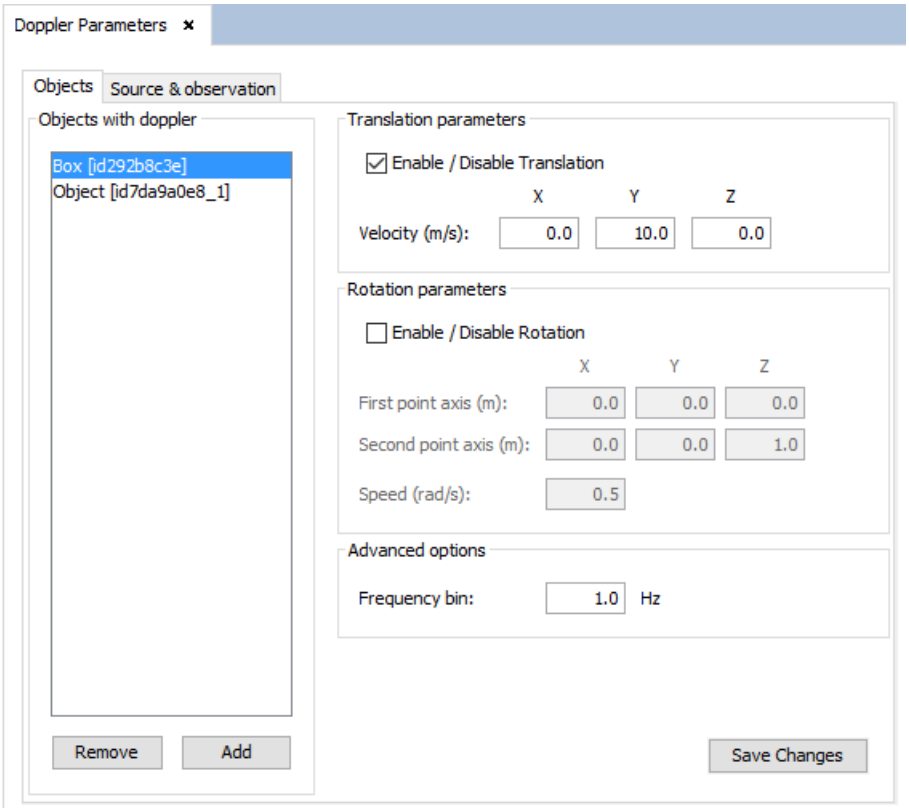


Figure 1840: Doppler parameters panel (Objects tab)

In the "Objects" tab, the user can add Doppler effect to objects in the geometry. To do this, select an object from the geometry view and press the **Add** button. The selected object will be added to the "Objects with doppler" list. It is possible to remove the Doppler effect from an object by selecting it from the list and pressing the **Remove** button.

- **Translation Parameters**. Any object in the list can have a translation movement. Using this panel the user can define the linear velocity of the object. The **Enable / Disable Translation** check box must be selected.
- **.**Any object in the list could have a rotation movement. The **Enable / Disable Rotation** check box must be selected. Using this panel the user can define:
- ◦ **First point axis** first point of the rotation axis.
  ◦ **Second point axis** second point of the rotation axis.
  ◦ **Speed** rotation speed.
- **.** The user can also define the frequency bin.

Once the parameters of the movement of one object have been defined, it is necessary to click on "Save Changes" button in order to save this information and assign it to the selected object.

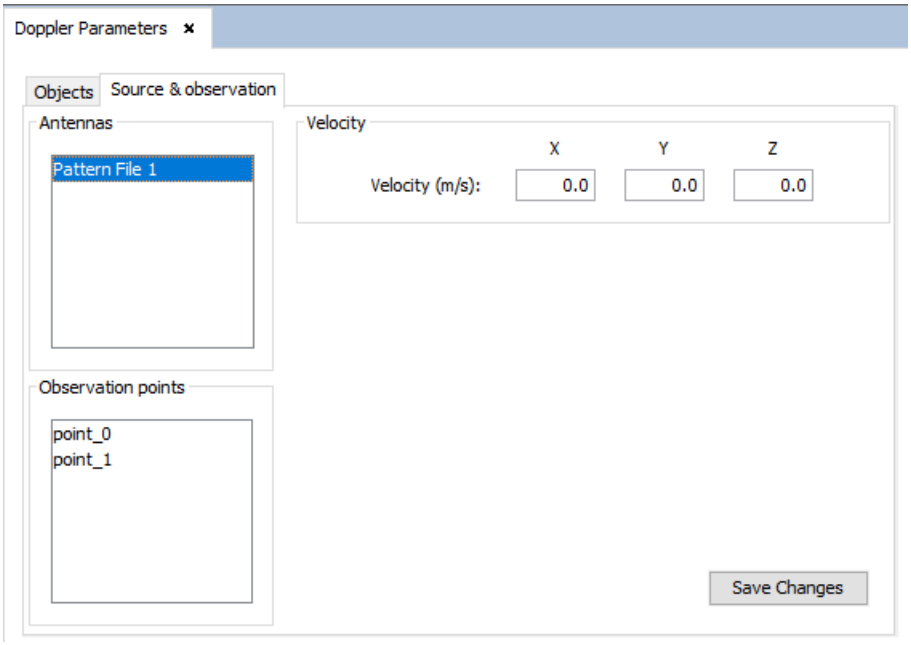


Figure 1841: Doppler parameters panel (Source & Observation tab)

In the "Source & Observation" tab, the user can set Doppler velocities to individual sources and observation points. To do this, the user should select an item from the "Antennas" or the "Observation points" lists and fill the Velocity text boxes with the desired velocity. It is possible to select multiple sources or observation points by holding the Ctrl key and clicking on each source or observation point.

Once the velocity for the selected antennas or observation points has been specified, the user needs to press the "Save Changes" button in order to confirm the changes.

14.7 Source Menu

The "Source" menu contains the options needed to define the ultrasound sources of the simulation. The "Source" menu contains the following options:

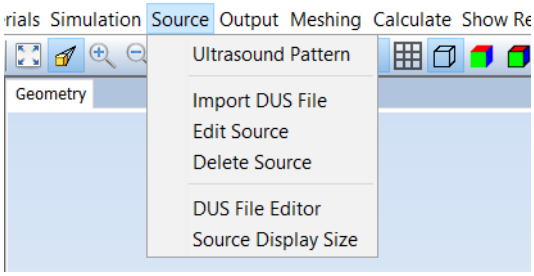


Figure 1842: Source menu

14.7.1 Ultrasound Pattern

This option allows the user to define an ultrasound source defined a ultrasound pattern file (also called DUS files). A DUS file is a text file format that defines the amplitude and phase of the sound waves emitted by a source in different directions.

Please note that **it is necessary to import a DUS file into the project** before defining an ultrasound pattern source. In order to so, the user needs to select the **Import DUS file** option under the **Source** menu and load a DUS file. The user can create their own DUS files by using the **DUS Files** option under the **Tools** menu.

When the user selects the **Ultrasound Pattern** option, the following panel will be shown:

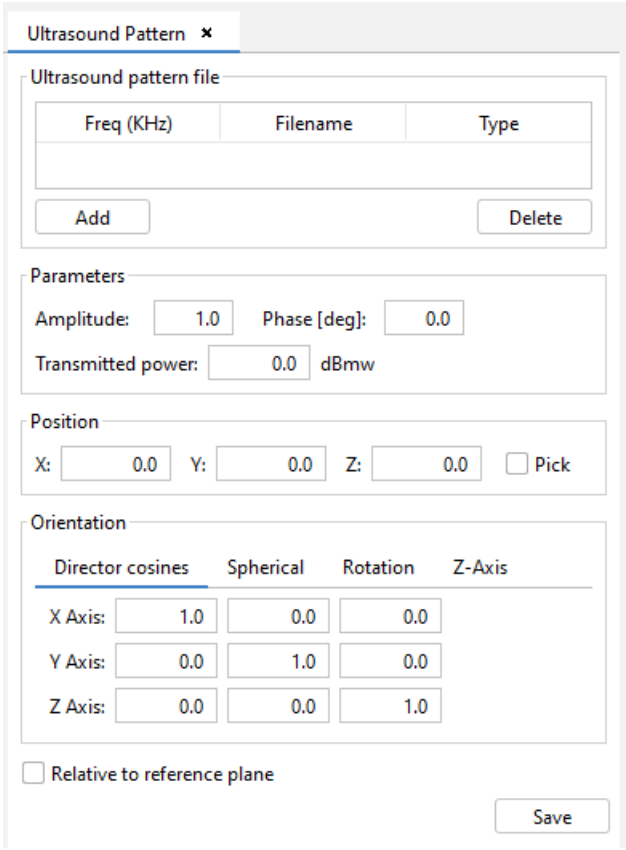


Figure 1843: Ultrasound source panel

In the "Set ultrasound pattern file" panel, the user can select multiple **DUS files** (ultrasound pattern files), from the ones imported, to be used for the ultrasound source. The user can specify several ultrasound pattern files to be used for different frequency values. The table will show, for each added DUS file, the frequency of operation (editable), the file name and whether it's a 3D pattern file (3DE) or a revolution pattern file (REV).

In the **Parameters** panel, the user can set the **Amplitude** factor for the emitted sound waves so that the amplitude of the resulting waves are multiplied by this factor. The user can also introduce a **Phase** variation, in degrees, to the sound waves defined in the ultrasound pattern. Finally, the user can set up the **Gain**, **Max directivity** and **Transmitted power** of the source. Note that **Gain** and **Max directivity** are in natural units.

The **Position - Coords** panel allows the user to specify the **location** of the ultrasound source. By checking the **Pick** check box, the user can pick a point by clicking in the desired position in the geometry panel.

The **Orientation** panel allows the user to configure the **orientation** of the ultrasound source using different methods.

- The **Director cosines** tab allows the user to explicitly provide the rotation matrix.
- The **Spherical** tab allows to specify the rotation using the theta and phi angles from spherical coordinates, as well as a alpha rotation -- the rotation around the source's local Z-axis.
- The **Rotation** tab allows the user to specify the angles of rotation around each axis.

- The **Z-Axis** tab allows the user to give the coordinates of the local Z-axis vector of the source, as well as the alpha rotation -- the rotation around the source's local Z-axis.

The **Relative to reference plane** check box controls whether the coordinates of the created source will be relative to the local coordinate system defined by the reference plane (checked) or they will be considered with respect to the absolute coordinate system (unchecked).

14.7.2 Import DUS File

This option allows the user to add DUS (ultrasound pattern) files to the current project. When the user selects this option, the following panel will appear:

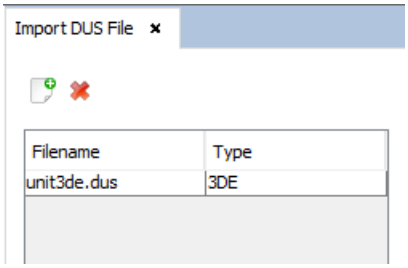


Figure 1844: Import DUS panel

The user can import a new DUS file by pressing the button. This will show a dialog that allows the user to import a DUS file (.dus extension). The imported files will appear in the table, showing their filenames and types (REV for revolution defined patterns, and 3DE por 3D patterns).

It is also possible to remove an already added DUS file by selecting its row in the table and pressing the button.

14.7.3 Edit Source

This option allows the user to modify the parameters of a previously created source. To edit an ultrasound source, the user first needs to select it in the geometry panel (selected sources will be colored with a bright yellow color) and then select this option. It is also possible to edit a source by right-clicking it and selecting the "Edit" option in the pop-up menu that appears.

The displayed panel for editing a source is the same as the one shown for adding a new source. Please, refer to previous sections for more details on the parameters of ultrasound sources.

14.7.4 Delete Source

This option allows the user to remove the selected ultrasound sources from the simulation. To delete ultrasound sources, the user needs to select them in the geometry panel (selected sources will appear colored with a bright yellow color) and then select this option. It is also possible to delete sources by selecting the "Delete" option in the pop-up menu shown when right-clicking a source.

14.7.5 DUS File Editor

The "DUS File Editor" under the "Source" menu allows the user to create their own DUS (ultrasound pattern) files. In addition, it is possible to edit already existing DUS files. When the user selects this option, the following panel is shown:

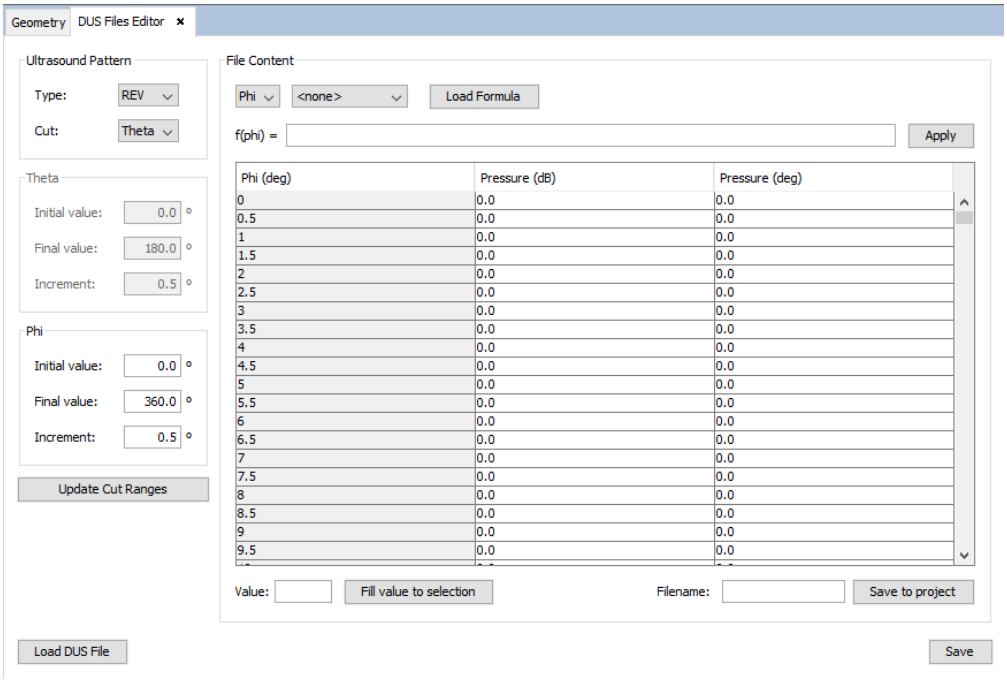


Figure 1845: DUS File editor panel

The "Ultrasound Pattern" panel allows the user to select the type of DUS file to be created, being able to select whether they want a revolution-defined ultrasound pattern or a 3D pattern. In the revolution-defined case, the user also needs to select whether it's a theta cut or a phi cut.

The "Theta" and "Phi" panels allow the user to specify the ranges of values used for the theta and phi spherical coordinates, respectively. In the case of a 3D pattern, both panels are enabled as both spherical coordinates are used at the same time. When the revolution-defined type (REV) is selected, the "Theta" panel will be enabled only when the "Phi" cut type is selected and vice versa. When the user makes an update to the range of either theta or phi values, they needs to press the "Update Cut Ranges" in order to update the table model and take into account the new ranges.

The table on the right of the panel allows the user to manually enter the desired amplitudes and phases for the sound waves along each direction. However, it is also possible to make bulk updates to multiple cells at once in one of the following ways:

- By entering a **formula** that depends either on theta (if a REV with phi cut is selected), on phi (if a REV with theta cut is selected) or both (if a 3D pattern is selected). In order to apply the formula to multiple cells of the table, the user needs to select the cells they want to update and then press the "Apply" button. The functions than can be used in formulas can be viewed in the drop-down list next to the "Load Formula" button. The user can select a function from the list and press this button to load it into the formula text box.

- By entering a value that will be entered into multiple cells. In order to do this, the user needs to enter a value into the text box below the table, select the cells they want to fill in, and then press the "Fill value to selection" button.

The "Load DUS File" option allows the user to load an already existing DUS file into the editor, allowing them to visualize and edit the values of this DUS file. This can be done by pressing the button and browsing the DUS file to be loaded.

When the user has finished the edition of the DUS file, they can save it into a file by pressing the "Save" button. A dialog will be shown to the user asking them to select the location where they want to save the file. Alternatively, the user can introduce a file name and press the "Save to project" button to automatically save and import the DUS file into the current open project (note that this option is only available for US projects).

14.7.6 Source Display Size

When this option is selected, the following panel is shown:

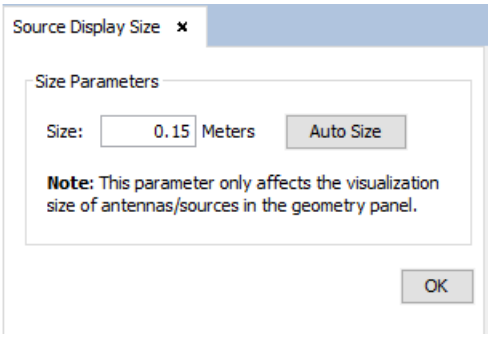


Figure 1846: Source Display Size panel

This option allows the user to scale the sources represented in the geometry view. The user needs to indicate the factor of scale. Pressing **OK** will cause the sources in the geometry view (and the new created sources) to be scaled. The **Auto Size** button allow the user to perform this task automatically. Note that this parameters only affects the visualization of sources in the geometry view and has no impact in the simulation.

14.8 Output Menu

This menu allows the user to set different parameters that determine what results are calculated in the simulation.

14.8.1 Observation Points

This option allows the user to specify the points where the results will be calculated. Please note that this option is available only in "Near Field" and "Doppler" simulations. If the user selects the "Observation Points" option, the following panel will appear:

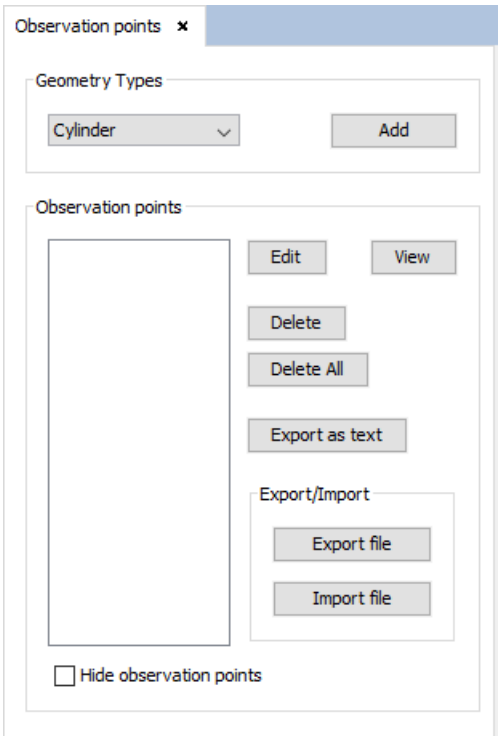


Figure 1847: Observation points panel

The user can add near field observation points in different ways. The next options are available:

- **Geometry Types** observation points may be inserted as a group that defines different shapes. Select the desired type entity and click on the "Add" button to set its parameters. The available **Geometry Types** are detailed in the following subsections. Before adding new elements, check that the **Name** of the new entity to be added is different from the already existing ones; otherwise, an error will be shown.
- **Observation points** this section contains several options to work with observation points.
- ◦ The left list contains all the observation shapes that have been added as observation points.
  ◦ The "Edit" button allows editing an observation points shape selected on the left list.
  ◦ The "Delete" button allows removing the observation entities selected on the left list.
  ◦ Use the "Delete All" button to remove all the observation points shapes on the left list.

  - The "Export as Text" button allows saving all the generated observation points. Every point is defined in a different line by its X, Y and Z coordinates.
  - The "Import File" button allows the user to add a previously saved observation points file. Only .obs files are supported, which are the internal format generated by the application.
  - The "Export File" button allows the user to save the current observation points configuration in a new file. It must be saved as .obs file, which is the internal format generated by the application.
- **Hide observation points** this option controls whether the observation points are shown in the geometry panel.

Cylinder

The observation cylinder may be defined as a closed cylinder (circular section) or only a cylindrical sector (arc section). The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Base Center** coordinates where the base of the cylinder will be placed (in the XY plane).
- **Radius** radius of the cylinder
- **Height Coordinates** to define the hight sampling rate of the cylinder.
-
  - Number of points define the number of samples to be generated along the height of the cylinder (in the Z axis).
  - **Height** defines the total height of the cylinder.
- **Angle Coordinates** to define the circular sampling rate of the cylinder.
-
  - **Number of points** define the number of samples to be generated along the transversal section of the cylinder (in the XY plane).
  - **Initial angle** angle where the first circular sample is generated in the transversal section, in degrees.
  - **Final angle** angle where the last circular sample is generated in the transversal section, in degrees.

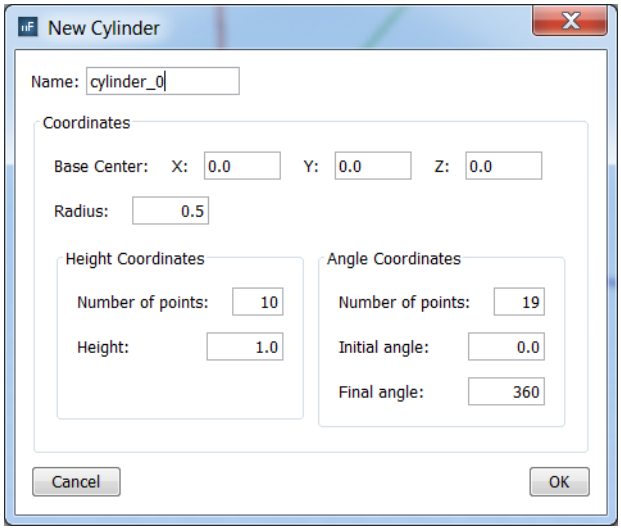


Figure 1848: New Cylinder parameters

Click on the "OK" button to insert the observation points from the cylinder, or abort the insertion by clicking on the "Cancel" button.

External File

The user can also import the near field points from a text file. This file can then be imported by selecting "External File" in the "Geometry Types" combo-box. Note that this option imports a file of points, but does not preserve the near field type (i.e. every point is imported as a separate type). In order to preserve the near field type, the "Export file" and "Import file" options must be used instead.

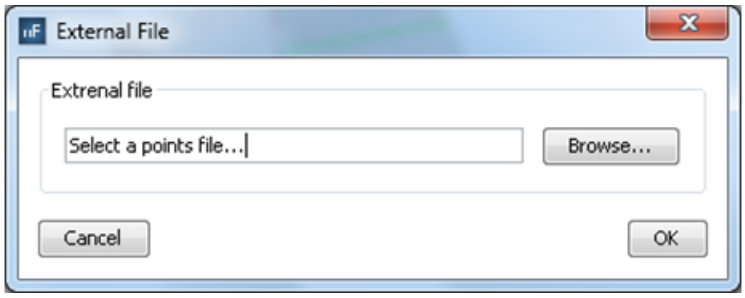


Figure 1849: Open External File dialog

The external file must contain a list of points given by its **X**, **Y** and **Z** coordinates, with every point specified in separated lines.

Click on "OK" button to insert the observation points from the file, or abort the insertion by clicking on "Cancel" button.

From Curve

This option lets you select a line or curve of the main panel and sampling it as observation points. Before clicking on the "Add" button having selected the "From Curve" type, the curve has to be selected in the Geometry panel. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Number of points** to specify the number of samples generated along the curve.

Click on the "OK" button to insert the observation points from the curve, or abort the insertion by clicking on the "Cancel" button.

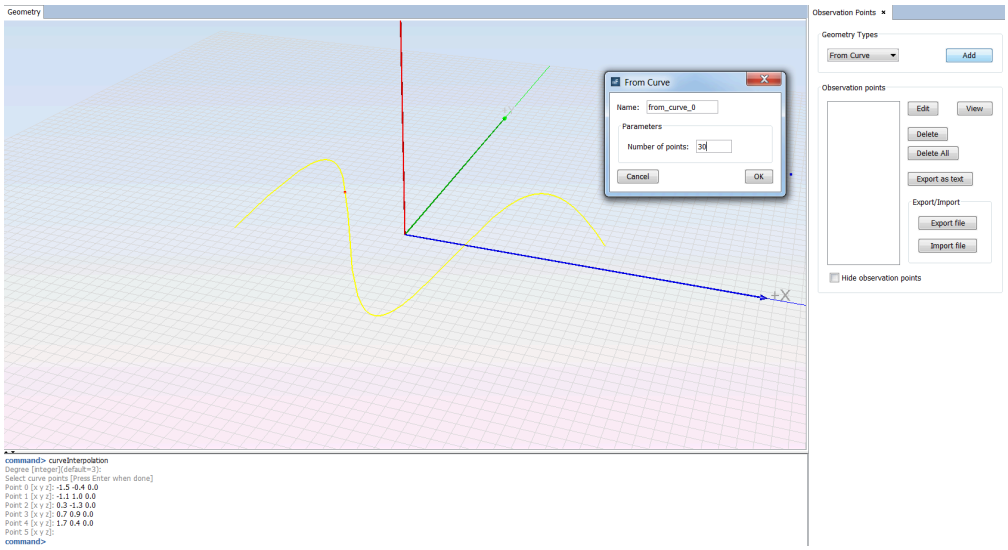


Figure 1850: The next figure shows that the grid points are added to the Observation points list, and represented together the geometry as a dotted grid.

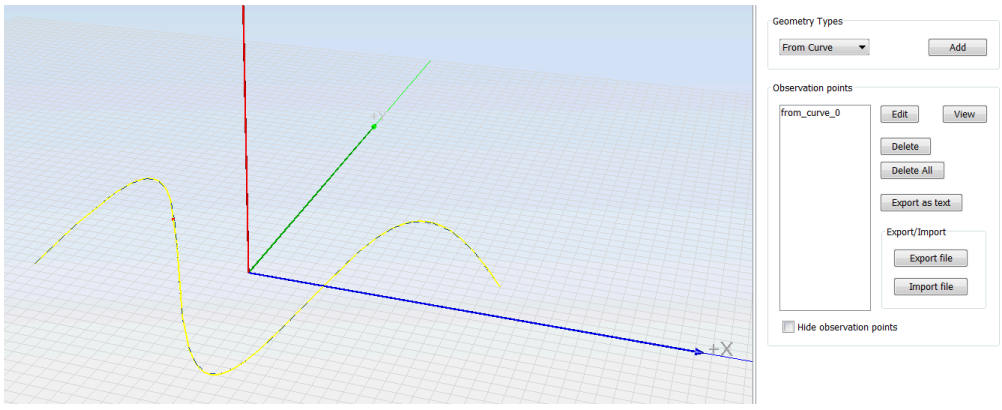


Figure 1851: Observation points generated on the curve

From Surface

This option allows the user to select a surface of the main panel and sample it as observation points. Before clicking on the "Add" button having selected the "From Surface" type, the surface has to be selected in the **Geometry** panel. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Parameters** the generated observation entity will be a matrix of U x V points obtained by sampling the surface.
-
  - **U-Dimension** to specify the number of samples generated along the U-Dimension of the surface.
  - **V-Dimension** to specify the number of samples generated along the V-Dimension of the surface.

Click on the "OK" button to insert the observation points from the surface, or abort the insertion by clicking on the "Cancel" button.

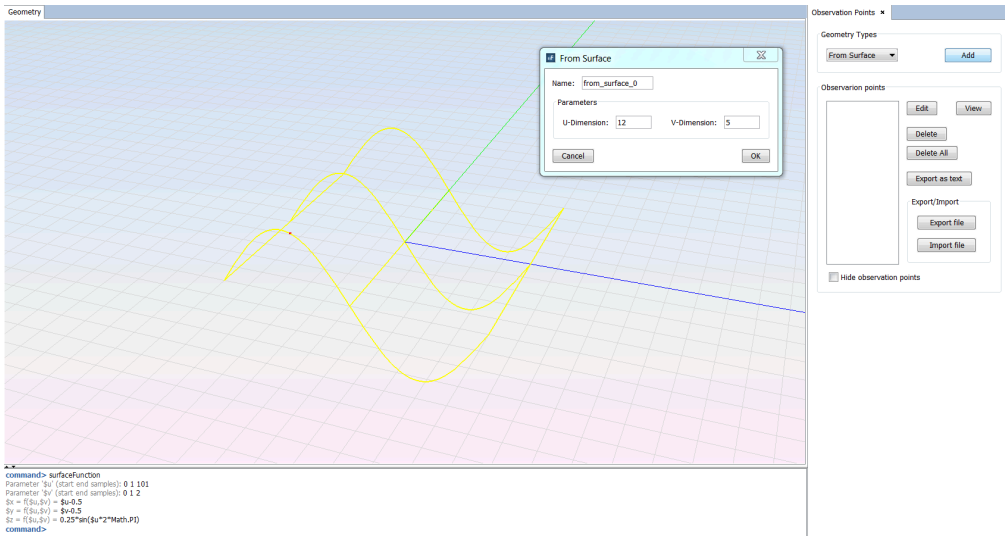


Figure 1852: Adding a surface of observation points

The next figure shows that the grid points are added to the **Observation points** list, and represented together the geometry as a dotted grid.

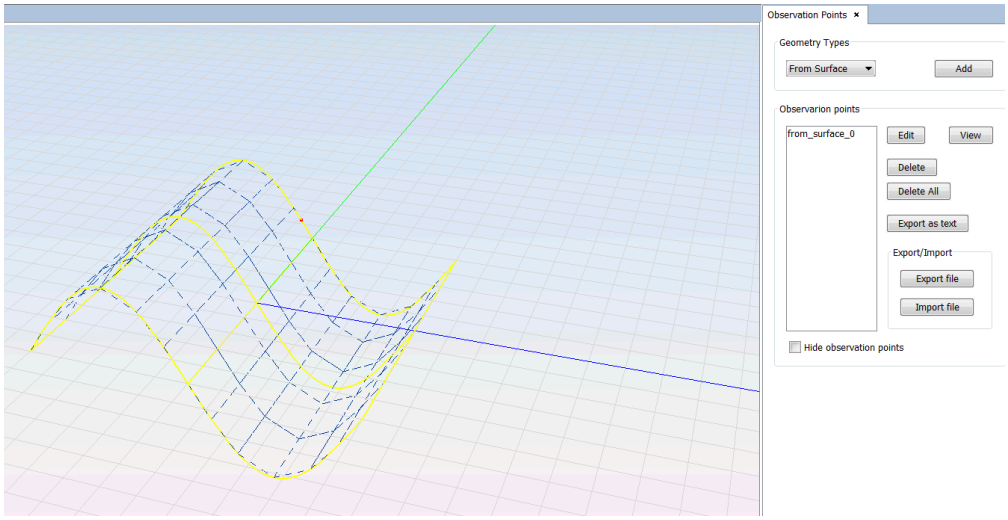


Figure 1853: Observation points generated on the surface

Line

Observation points also may be generated along a virtual straight line sampled between its extremes. As shown in the below figure, the following parameters are required:

- **Name** to define the name of this set of observation points. The name must be different of the already defined ones.
- **Coordinates** the line is defined by its side points.
- - **Initial Point** first end point of the line.
  - **Final Point** opposite end point of the line.

Click on the "OK" button to insert the observation points from the line, or abort the insertion by clicking on the "Cancel" button.

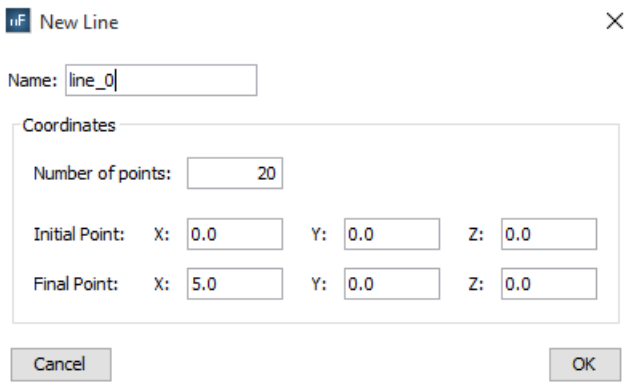


Figure 1854: New Line parameters

Parallelogram

Use this option to generate a parallelogram of coplanar observation points. The parameters required to its shape are:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Coordinates** the parallelogram is defined by three points that form its two main axis, so the generated observation entity will be a matrix of Side x Botton points obtained by sampling the parallelogram.
  - Side left axis is defined from **Point 1** to **Point 2**.
  - Bottom axis is defined from **Point 2** to **Point 3**.

Click on the "OK" button to insert the observation points from the parallelogram, or abort the insertion by clicking on the "Cancel" button.

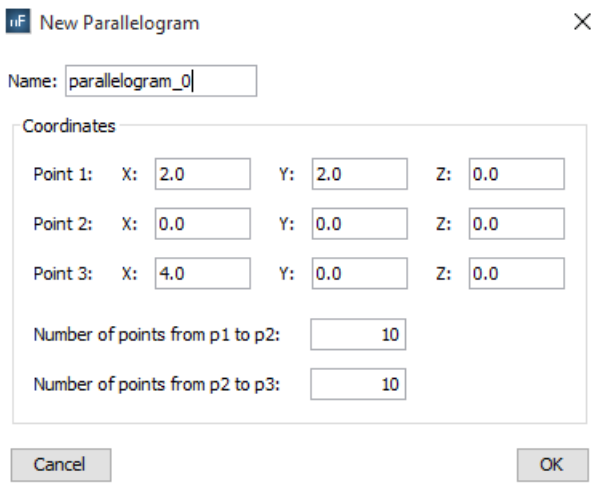


Figure 1855: New Parallelogram parameters

Plane

In this window, the user can create a set of observation points laid out in a grid pattern. To do so, the user needs to specify several parameters:

- **Name** A short name that will be used by the user to identify the created set of observation points.
- **Constant axis** The normal axis of the plane that will contain the grid of observation points.
- **Coordinate value** Value of the coordinate of the axis specified in the previous option.

For each of the axes different than the one selected in the "Constant axis" drop-down list, the user needs to specify the following parameters:

- **Initial value** Initial value of the coordinate along each axis.
- **Total size** The total length of the grid of observation points along the axis.
- **Number of points** The number of observation points along the axis.

Note The spacing between each pair of consecutive observation points along an axis will be given by the formula **spacing = (total size) / (number of points - 1)**. This value will be shown next to the label "Distance btw. points".

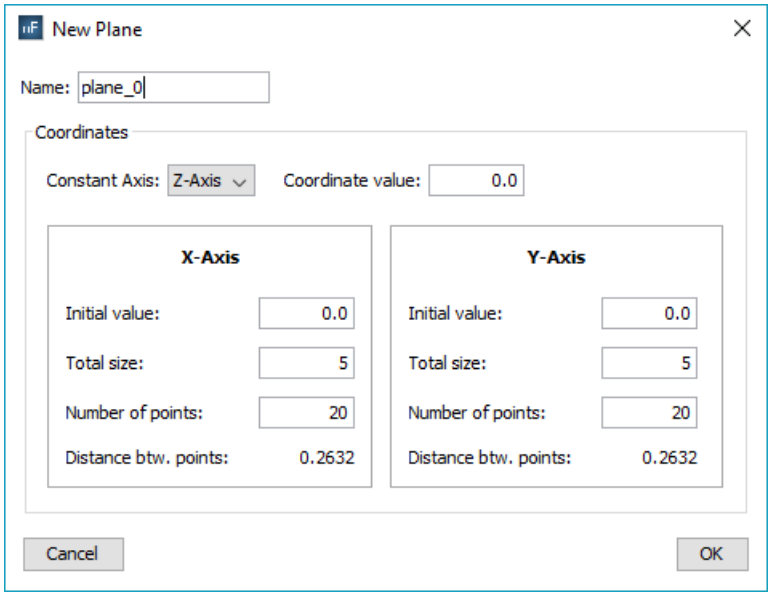


Figure 1856: New Plane parameters

Point

Isolated observation points may be generated by using this option. The parameters to specify are:

- **Name** to define the name of the observation point. The name must be different of the already defined ones.
- **Coordinates** position of the observation point, specified by its X, Y and Z coordinates.

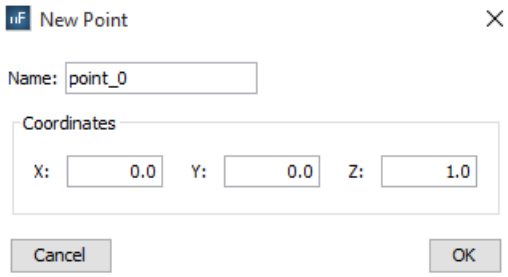


Figure 1857: New Point parameters

Click onthe "OK"button to insert the observation point, or abort the insertion by clicking on"Cancel"button.

Sphere

The observation sphere may be defined as a full closed sphere or only an spherical sector specified. Multiple concentric spheres (or spherical sectors) may be defined within the same entity. The following parameters are required:

- **Name** to define the name of this type of observation points. The name must be different of the already defined ones.
- **Parameters**:
  - **Center Point** X, Y and Z coordinates where the sphere will be centered on.
  - **Radius** radius of the sphere. A radius sweep may be defined to generate multiple concentrical spheres.
    - **Initial Radius** radius of the first generated sphere, which is the only one if the sweep is not configured.
    - **Increment Radius** step between two adjacent spheres. It is only considered when the sweep is configured.
    - **Radius Samples** number of concentric spheres to be generated, it must be an integer number. The radius sweep is enabled when this parameter is higher than 1.
  - **Theta Coordinates** to set the spherical range where te observation points are generated, in theta dimension.
    - **Number of points** defines the number of samples to be taken in theta dimension along the specified angular range.
    - **Initial angle** defines the angular angle (in degrees) of the first theta point to be generated.
    - **Final angle** defines the angular angle (in degrees) of the last theta point to be generated.
  - **Phi Coordinates** to set the spherical range where te observation points are generated, in phi dimension.
    - **Number of points** defines the number of samples to be taken in phi dimension along the specified angular range.
    - **Initial angle** defines the angular angle (in degrees) of the first phi point to be generated.
    - **Final angle** defines the angular angle (in degrees) of the last phi point to be generated.

The total number of observation points to be generated in the sphere is (**Radius Samples**) x (**Number of Points**) $_{\text{Theta}}$ x (**Number of Points**) $_{\text{Phi}}$.

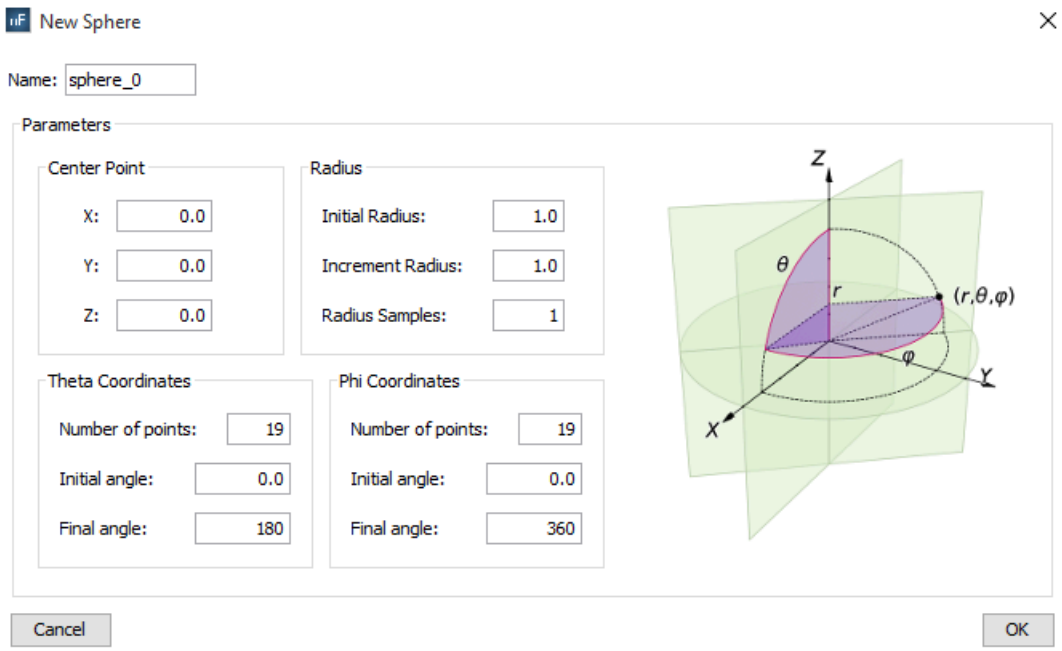


Figure 1858: Spherical observation points.

Click on the "OK" button to insert the observation points from the sphere, or abort the insertion by clicking on the "Cancel" button.

14.8.2 Coupling

This option allows the configuration of the receiver of ultrasound waves in "Coupling" simulations. In order to create a ultrasound receiver, a DUS file must have been imported into the project. Please, check section "Import DUS File" for more information.

When the user selects this option, the following panel appears:

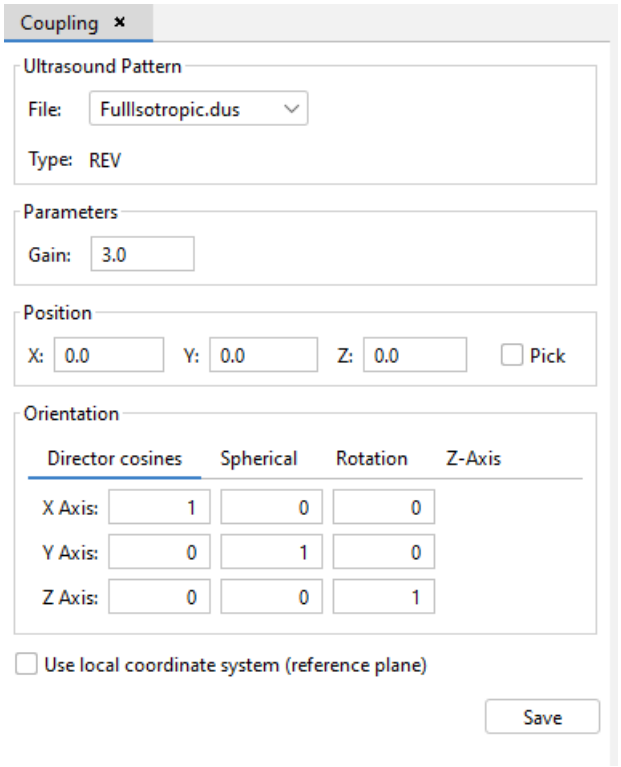


Figure 1859: Coupling panel

This panel allows the configuration of the new ultrasound receiver or the edition of the parameters of an already existing one. The **File** dropdown list allows the selection of one of the DUS file imported into the project. The **Gain** allows to set up the gain of the receiver (note that **Gain** is in natural units). The **Position** and **Orientation** panels allows to set the location and orientation of the receiver in the same way as in the **Ultrasound Source** panel.

It is possible to use the current local coordinate system (defined by the reference plane) so that the given coordinates and orientation are relative to it. To do this, the check box **User local coordinate system (reference plane)** must be selected.

The user needs to press the **Save** button in order to add or edit the ultrasound receiver.

14.9 Meshing Menu

14.9.1 Create Visibility Matrix

Prior to running the simulation, it is necessary to run a preprocessing that takes into account the geometry of the simulation case and prepares the files needed to run the actual simulation. When the "Create Visibility Matrix" option is selected, the following panel is shown:

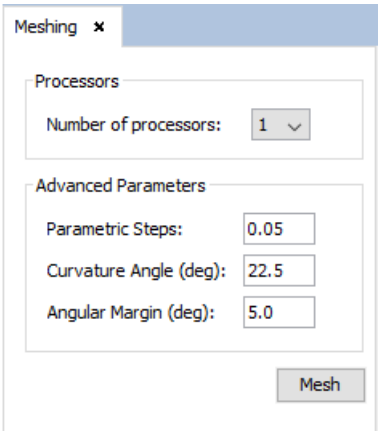


Figure 1860: Create Visibility Matrix panel

The following parameters can be configured:

- **Number of processors** The number of processors used to run the meshing process.
- **Parametric Steps** Accuracy in the curvature evaluation, normalized. Decrease this parameter to get a higher precision in the parametric discretization.
- **Curvature Angle** Maximum curvature considered on targets. It indicates the maximum curvature allowed in the input surfaces for the meshing step. Decrease this parameter to get a higher resolution mesh.
- **Angular Margin** maximum deviation allowed around the curvature reference angle. The curvature discretization is achieved by an approximated iterative method, so an angular error around the previous parameter is allowed. Decrease this parameter to increase the precision in the curvature discretization.

To start the meshing process, the user needs to press the "Mesh" button. This process may take a while (depending on the number of processors selected and the computing power of the computer running the simulation). While it is running, the status of the meshing process will be reported in the process log panel. From this panel we can abort the running meshing process, as well as save the log to a text file.

14.10 Calculate Menu

The calculate menu contains only one option, which allows the user to start the simulation process.

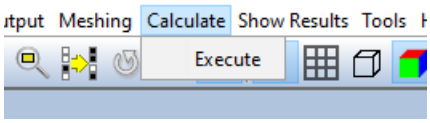


Figure 1861: Calculate menu

When the user selects the "Execute" option, the following panel will be shown:

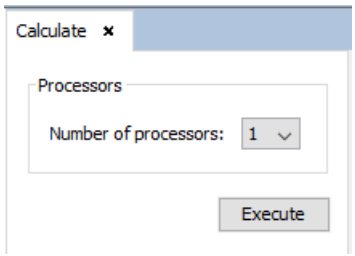


Figure 1862: Execute panel

In this panel, the user needs to select the number of processors that will be used to run the simulation. Before pressing the "Execute" button, it is advisable to check that all the simulation parameters are correct. If a problem is detected when trying to run the simulation, it will be notified to the user. Please note that the simulation can only be started if the meshing process has been executed successfully.

When the "Execute" button is pressed, the simulation process will start. The Process Log will appear and it will display the progress of the current simulation. It is possible to abort the simulation process while it is running by pressing the "Abort" button.

14.11 Show Results Menu

This menu allows the user to visualize the results of the simulation. The results are stored in external files that the user can access to obtain the information corresponding to that simulation.

Most of the plots show 2-D curves of a result field with respect to an input parameter. This type of graphics share additional features of zoom, axis, curves edition, and so on. To read more information about the chart options (when clicking on the right button over a chart) see Annex 1: Graphics Advanced Options.

14.11.1 View Ray Tracing

This option allows the user to view the rays emitted by the source of the simulation that end up reaching an observation point.

When open, the panel will display the rays for a particular step, order, source, observation point and speed. All the effects are selected by default. The possible configuration options are:

- The parametric step the results are visualized for.
- **Ray effect**: The user can filter the rays shown in the geometry panel by selecting the effects of the rays they want to visualize.
- **Order**: The order of the displayed rays. The order of a ray corresponds to the number of bounces it makes before reaching the observation point.
- **Source**: The ultrasound source that generates the rays the user wants to visualize.
- **Point**: The point where the visualized rays impact, or "All" to see the rays of all observation points.
- **Speed**: The speed of sound the results are visualized for.

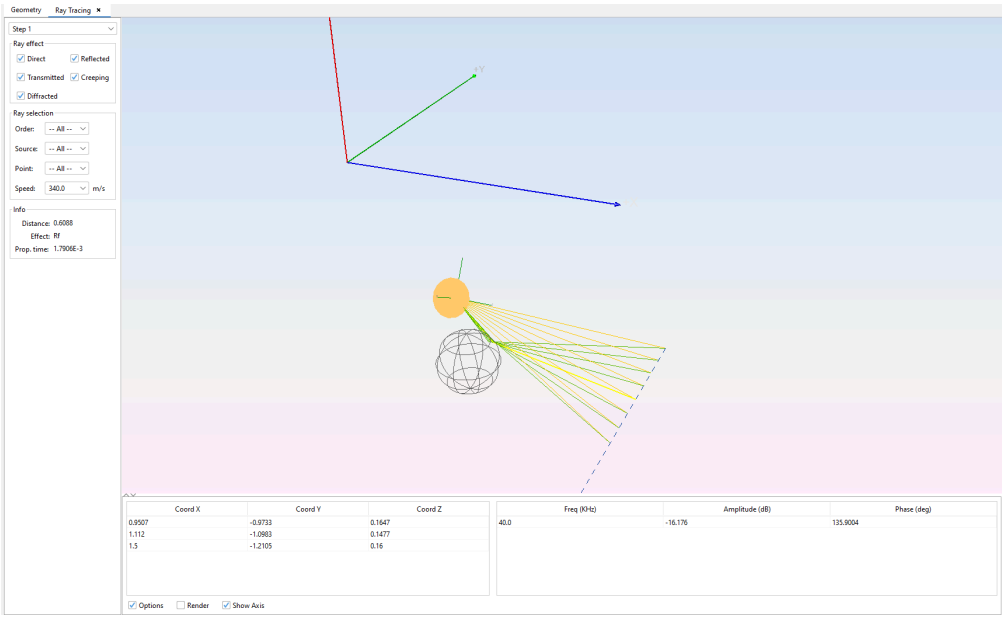


Figure 1863: Ray visualization panel

Selecting a ray will highlight it in yellow and its information will be shown in the Info panel (length of the ray, type of effect and propagation time). Other information will be shown in the bottom panel. This information comprises the frequency and its contribution to the total pressure at the corresponding observation point. The critical points of the ray are also displayed. These points consist on the starting point, the points where the ray bounces and the ending point (the observation point).

There are several visualization options at the bottom of the ray visualization panel:

- **Options** Controls whether the left (options) panel is shown.
- **Render** Controls whether the geometry in the geometry panel is fully rendered or shown as a wireframe model.
- **Show Axis** Controls whether the X, Y and Z axes are shown.

14.11.2 View Ray Tracing Text File

This option allows the user to visualize the information of the rays computed in the ray-tracing process in text format, allowing them to export these files in case they are needed for processing by external tools.

When the user selects this option, the following panel appears:

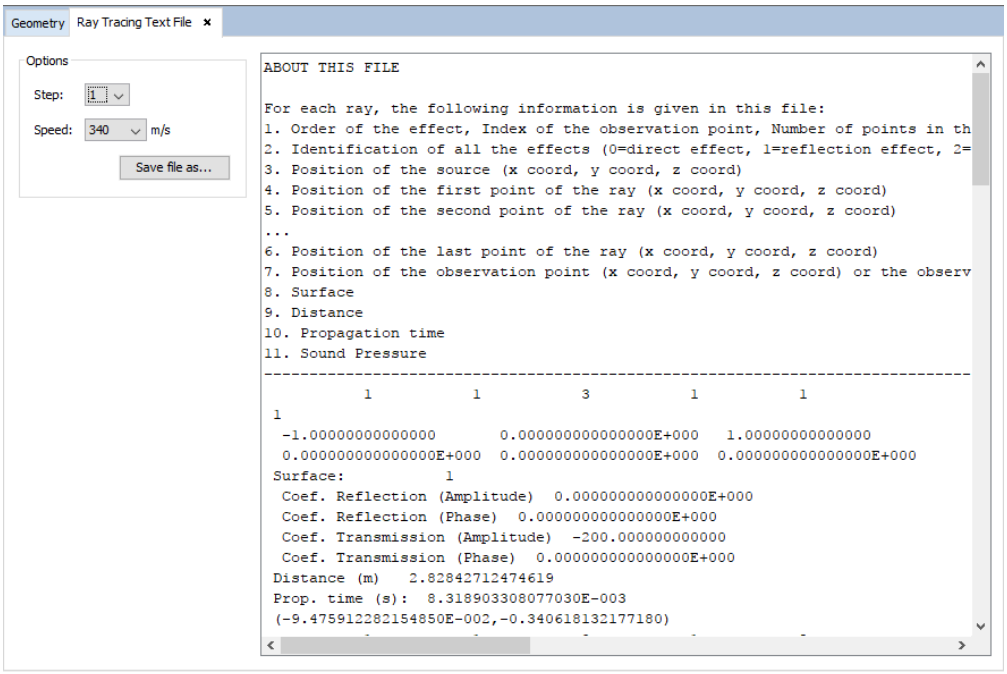


Figure 1864: Figure Ray-tracing text file panel

In the left side of the panel, the user can select the parametric step and the speed of sound to show the results for. When any of these options is modified, the text area will immediately show the updated file. The user can export the currently shown file to a location in their hard drive by pressing the "Save as..." button and selecting the location in the dialog that will appear.

The information that the ray-tracing file shows, for each ray, is the following:

- The first line contains the order (number of effects), the observation point, the number of critical points of the ray path, the source that generates the ray and the first effect (more info on how to interpret this field in the next point). All these values are separated with spaces.
- The second line shows each one of the effects of the ray. Each effect is associated with a specific number, i.e. 0 = direct, 1 = reflected, 2 = diffracted, 3 = transmitted, and 4 = creeping.
- The following lines contain the coordinates of each critical point of the ray. The first point always matches the coordinates of the source that emits the ray. Subsequent points are the ones where the ray is reflected, transmitted or diffracted. The last point always matches the coordinates of the observation point.
- The next line shows the index (starting in 1) of the surface where the ray is transmitted, reflected or diffracted (in the case of a direct ray, this value is 0). If the ray is transmitted, reflected or diffracted, the next four lines will show the reflection and transmission coefficients of the surface. If the ray has multiple effects, the index of the surface and the reflection and transmission coefficients will be shown for each effect.
- The next line shows the distance the ray travels between being emitted by the source and arriving at the observation point, in meters.
- The next line shows the propagation time of the ray, in seconds.
- The final line shows the contribution of the ray to the sound pressure at the observation point, as a complex value given in rectangular form.

This content is repeated for each one of the rays computed in the ray-tracing process.

14.11.3 View Text Files

This option allows the user to visualize the values of the sound pressure field in text format. These results are shown for each frequency and observation point at a given parametric step. When the user selects the **View Text Files** option, the following panel will appear:

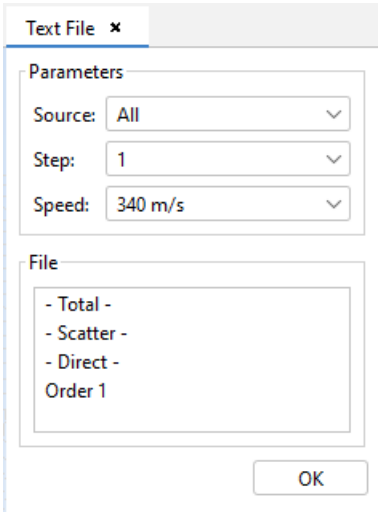


Figure 1865: Text File options panel

In this panel, the user needs to select the ultrasound source, the parametric step, the speed of sound and the order of the rays considered for the field calculation. When the user presses the **OK** button, a panel will appear showing the results:

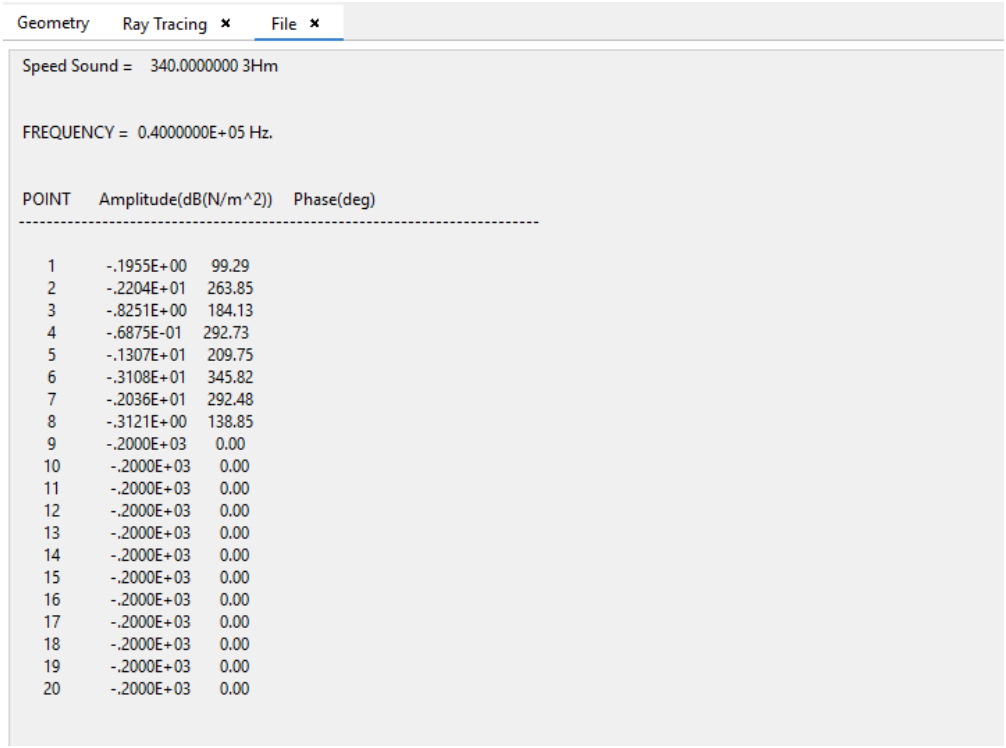


Figure 1866: View Text Files panel

For each frequency, a list of observation points with their respective field values (module and phase) are displayed. The user can export the text results to a file by pressing the **Save as** button and selecting the desired location and name for the file.

14.11.4 Near Field

The options in this menu allow the user to visualize the field values at the observation points defined in the simulation parameters.

The following options are available:

- **View Cuts**: Visualizes a plot of the field values with respect to the observation points cut.
- **View Cuts by Frequency**: Visualizes a plot of the field values at a given point, step and speed of sound with respect to frequency.
- **View Cuts by Step**: Visualizes a plot of the field values at a given frequency, point and speed of sound with respect to the parametric step.
- **View Cuts by Speed**: Visualizes a plot of the field values at a given frequency, point and step with respect to the speed of sound.
- **Near Field 3D**: Visualizes a 3D diagram showing the amplitude of field values at each observation point for a given step and frequency.

- **Delay Process**: Visualizes the temporal evolution of the signal emitted by the ultrasound sources at an observation point, allowing the user to choose the shape of the signal.
- **Power Process**: Visualizes the power of the sound waves emitted by the ultrasound sources as received by each observation point.

View Cuts

This option allows the user to visualize a plot of the sound pressure values at a given frequency, step and speed of sound with respect to the cut of observation points. When this option is selected, the following panel is shown:

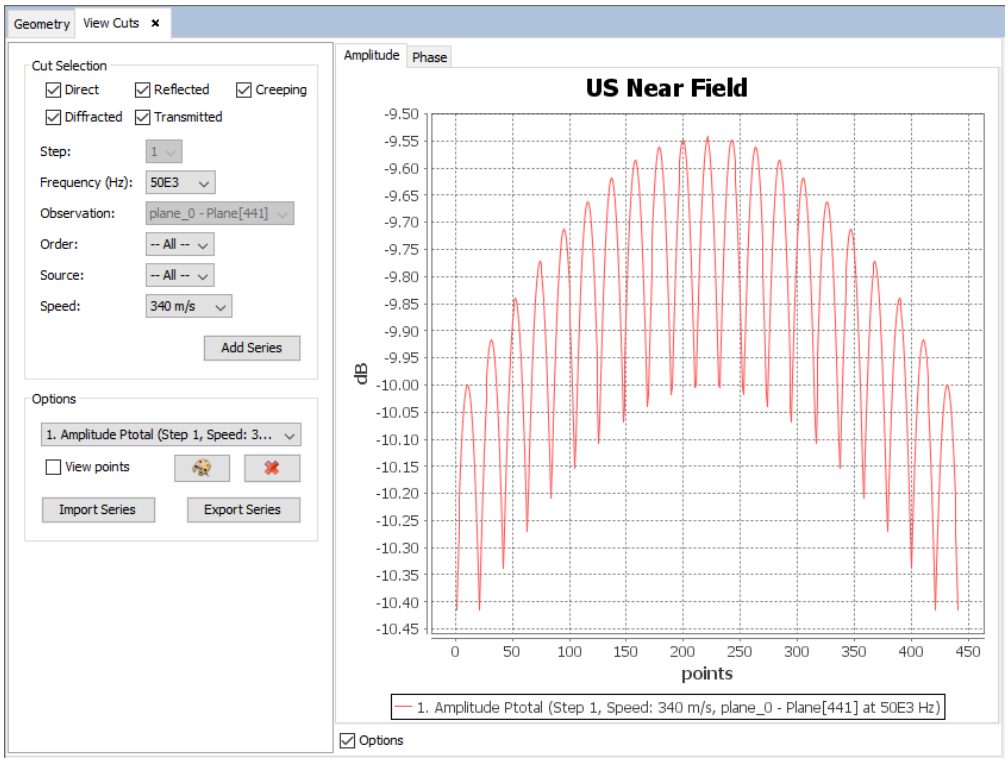


Figure 1867: View Cuts Panel

In this panel, the user can select the effects of the rays used to calculate the field, the parametric step, the frequency, the set of observation points, the order of the considered rays, the source and the speed of sound. By pressing the "Add Series" button, a new plot will be added to the panel.

It is also possible to visualize the phase of the field values by selecting the "Phase" tab in the chart panel.

The user can also manage the already added series, being able to do actions such as removing series, changing the color of series or importing and exporting series from/to text files.

View Cuts by Frequency

This option allows the user to visualize the field values at a given point, step and speed of sound with respect to the frequency. When the user selects this option, the following panel will be shown:

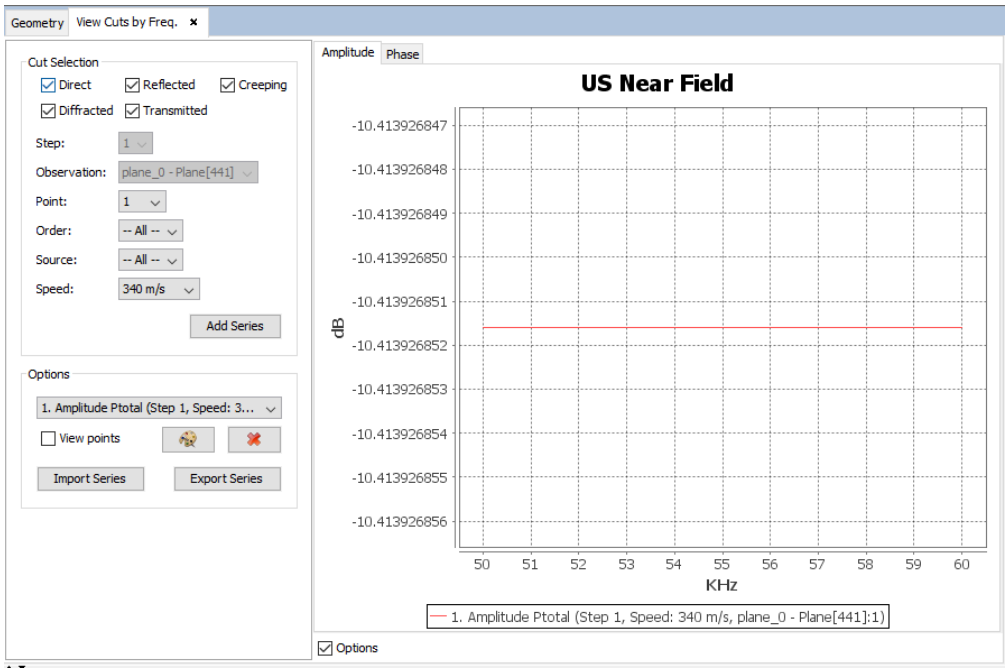


Figure 1868: View Cuts by Frequency panel

In this panel, the user can select the effects of the rays used to calculate the field, the parametric step, the set of observation points, the point, the order of the considered rays, the source and the speed of sound. By pressing the "Add Series" button, a new plot will be added to the panel.

It is also possible to visualize the phase of the field values by selecting the "Phase" tab in the chart panel.

The user can also manage the already added series, being able to do actions such as removing series, changing the color of series or importing and exporting series from/to text files.

View Cuts by Step

This option allows the user to visualize the field values for a given point, frequency and speed with respect to the parametric step. When the user selects this option, the following panel is shown:

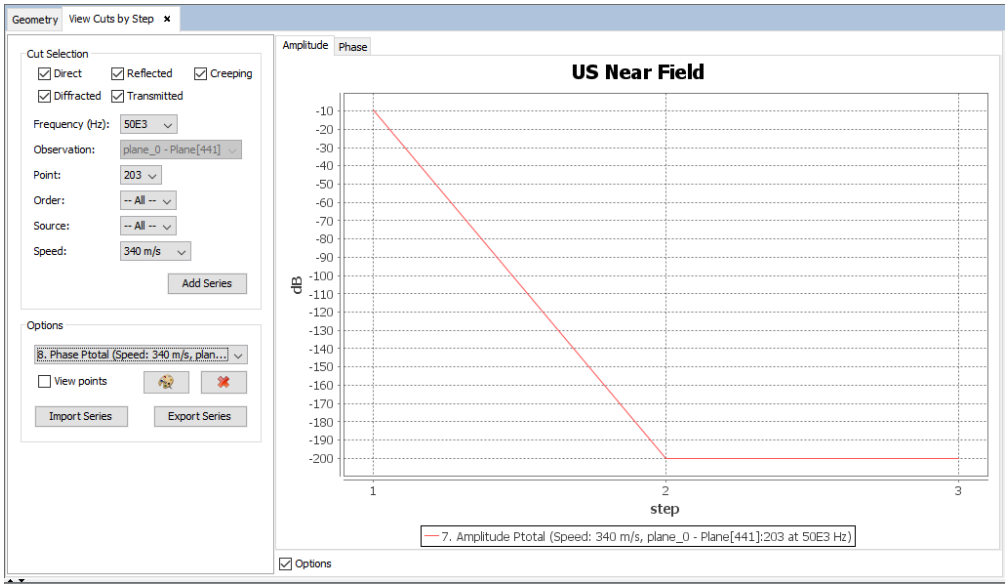


Figure 1869: View Cuts by Step panel

In this panel, the user can select the effects of the rays used to calculate the field, the frequency, the set of observation points, the point, the order of the considered rays, the source and the speed of sound. By pressing the "Add Series" button, a new plot will be added to the panel.

It is also possible to visualize the phase of the field values by selecting the "Phase" tab in the chart panel.

The user can also manage the already added series, being able to do actions such as removing series, changing the color of series or importing and exporting series from/to text files.

View Cuts by Speed

This option allows the user to visualize the field values for a given point, frequency and step with respect to the speed of sound. When the user selects this option, the following panel is shown:

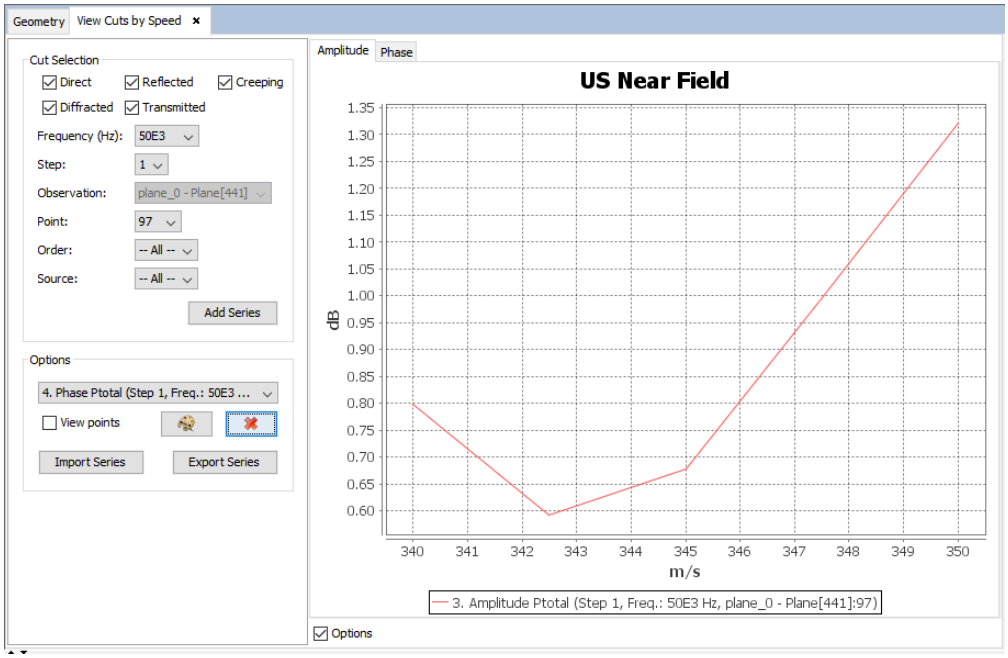


Figure 1870: View Cuts by Speed panel

In this panel, the user can select the effects of the rays used to calculate the field, the frequency, the step, the set of observation points, the point, the order of the considered rays and the source. By pressing the "Add Series" button, a new plot will be added to the panel.

It is also possible to visualize the phase of the field values by selecting the "Phase" tab in the chart panel.

The user can also manage the already added series, being able to do actions such as removing series, changing the color of series or importing and exporting series from/to text files.

Near Field 3D

This option allows the user to visualize a 3D diagram displaying the amplitude of the field values at each observation point, given the step and frequency. When this option is selected, the following panel appears:

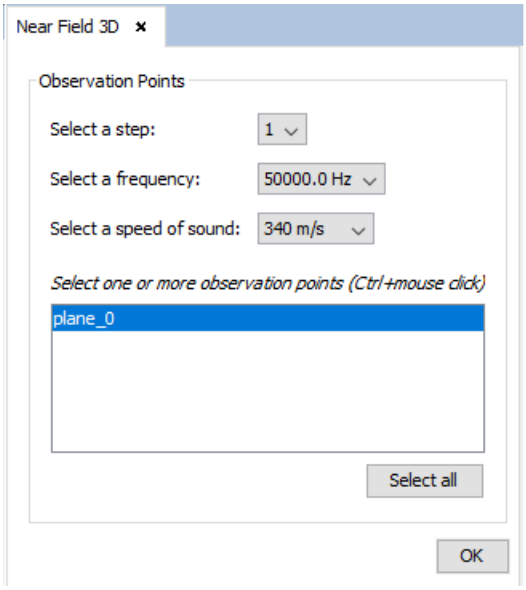


Figure 1871: Near Field 3D options

In this panel, the user needs to select the step, frequency and speed of sound they want to visualize the 3D diagram for, as well as the group or groups of observation points they want to compute the near field for. After the user has configured the parameters and pressed the "OK" button, the following panel appears:

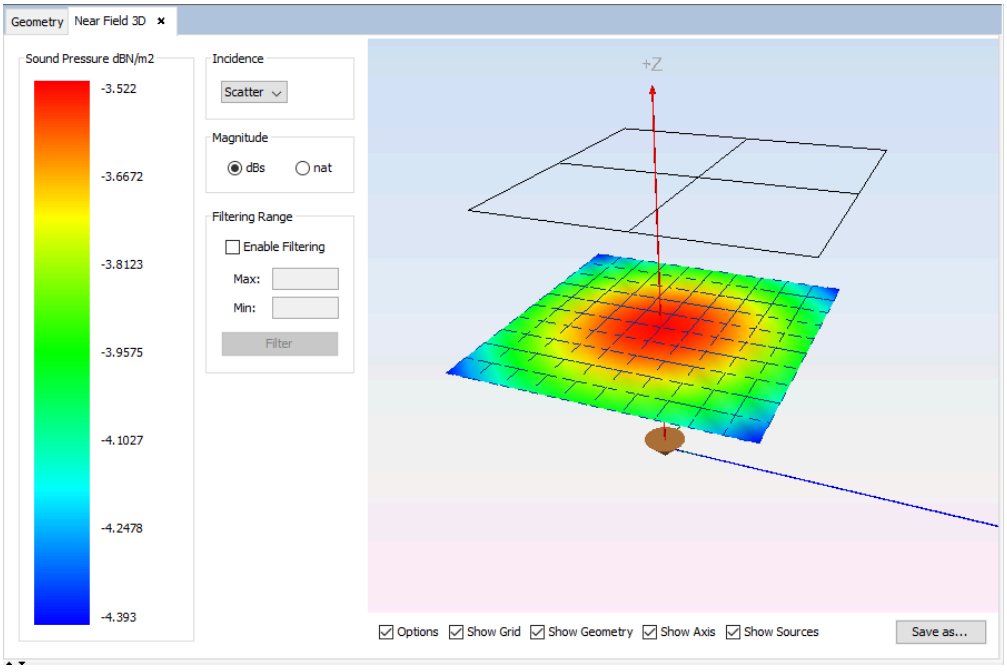


Figure 1872: Near Field 3D diagram

The user can select whether they want to visualize the contributions of the direct rays to the field, the contribution of the scatter rays, or the total field. They can also select whether they want to visualize the results in logarithmic units (dBPa - that is, sound pressure with respect to 1 pascal in dB) or

natural units (Pa). The user can also filter the displayed values within a range, by checking the "Enable Filtering" check box, entering the minimum and maximum values and pressing the "Filter" button.

Additionally, there are several options the user can use to alter the visualization of the results, such as possibility of hiding/showing the left panel, the observation points grid, the geometry, the axes or the sources. The user can also save the 3D diagram to an image file in a given location in their computer.

Delay Process

This option allows the user to visualize the Delay Process, that is, a visualization of the signal emitted by the ultrasound sources in the time domain, as it is received at an observation point. When the user selects this option, the following panel is shown:

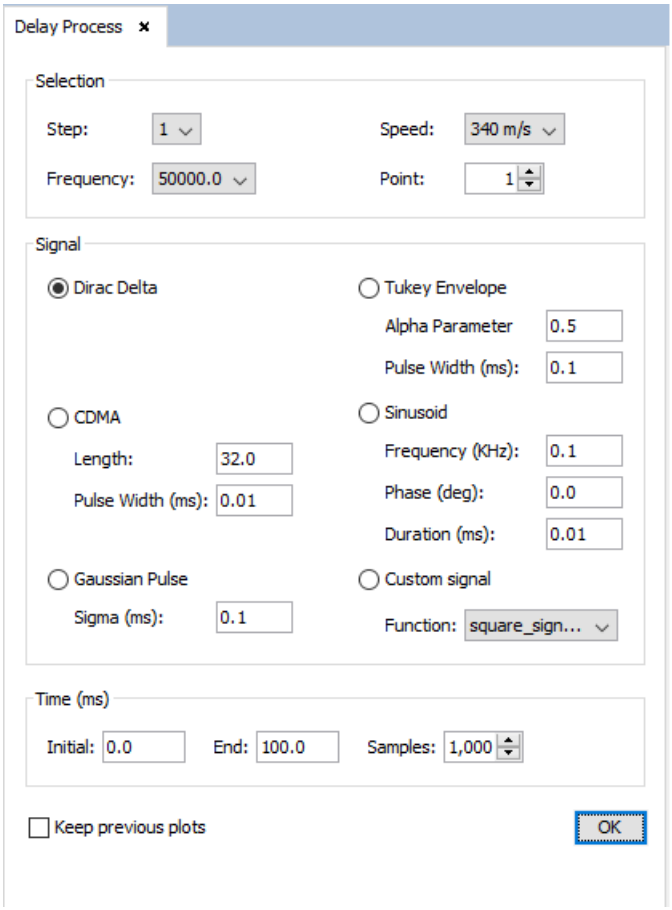


Figure 1873: Delay Process panel

In this panel, the user needs to select the step, frequency, observation point and speed of sound they want to show the results for. In addition, the user needs to select one of the multiple signal shapes:

- **Dirac Delta** A instantaneous pulse.
- **CDMA** A random distribution of pulses centred with positive (high) a negative (low) levels. Note that a low or high level is considered as a Pulse. The "Length" parameter determines the number of pulses to be generated, while "Pulse Width" refers to the time duration of each pulse, in milliseconds.

- **Sinusoid** A sinusoidal signal. The "Frequency" is the frequency of the sinusoidal signal, in kHz. The "Phase" is the initial phase of the sinusoidal signal, in degrees. The "Duration" is the total length of the generated sinusoid in the time domain, in milliseconds.
- **Tukey Envelope** A Gaussian signal enveloped by a rectangular pulse. The "Alpha Parameter" defines the slope of the Gaussian curve. The higher is this parameter, the more enveloped by the rectangular pulse is the generated signal. The "Pulse Width" defines the total length of the generated signal in the time domain, in milliseconds.
- **Gaussian pulse** A Gaussian pulse defined by its **sigma** parameter, in milliseconds.
- **Custom signal** A user-defined signal (by a user function). In this drop-down list, only functions that return a double value and accept no arguments will be shown. The user function defines the base signal function, that must be centered at t=0. This function is then multiplied by the field value of each ray, translated in time by the propagation of the ray and finally the value of all functions are added at each point to obtain the Delay Process plot.

The following is the example of a user-function that can be used as a custom signal (a square signal of amplitude 1 and 3ms long):

```
double square_signal(){
 double tms = $t * 1000.0;
 double DURATION_MS = 3;
 if (tms <= -DURATION_MS/2 || tms >= DURATION_MS/2) {
  return 0.0;
 } else {
  return 1.0;
 }
}
```

The function used as a custom signal has access to three special variables, prefixed by the dollar symbol:

- $t The time value, in seconds, the function is sampled at.
- $freq The selected frequency, in Hz.
- $sound_speed The selected speed of sound, in m/s.

Additionally, the "Time (ms)" panel allows the user to specify the time interval they want to see the results for. The "Initial" and "Final" fields represent the bounds of the time interval, while the "Samples" field is the number of evenly-spaced points that will be calculated within the time interval and therefore the number of points of the generated plot.

The "Keep previous plots" option allows the user to choose whether they want to preserve the previously plotted graphs (which could be useful if the user wants to compare several plots) or they want to clear all previous plots. This option only has an effect for signals different than "Dirac Delta" (which are always shown in a different panel each time).

When the user has finished setting the parameters, they need to click on the "OK" button to plot the **Delay Process** results.

Only when the CDMA signal has been selected, the window represented below asks the selection of the Correlation Signal to be plotted with the following options.

- **Input (x[n])** the original CDMA signal is represented.
- **Output (y[n])** the received CDMA signal is represented

- **Correlation (c[n])** the correlation on the received signal is represented.

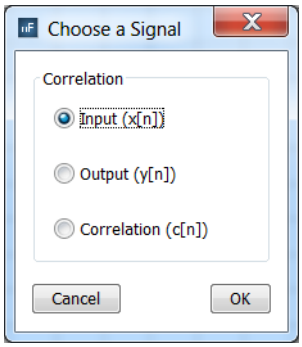


Figure 1874: Selection of signal to be plotted with CDMA

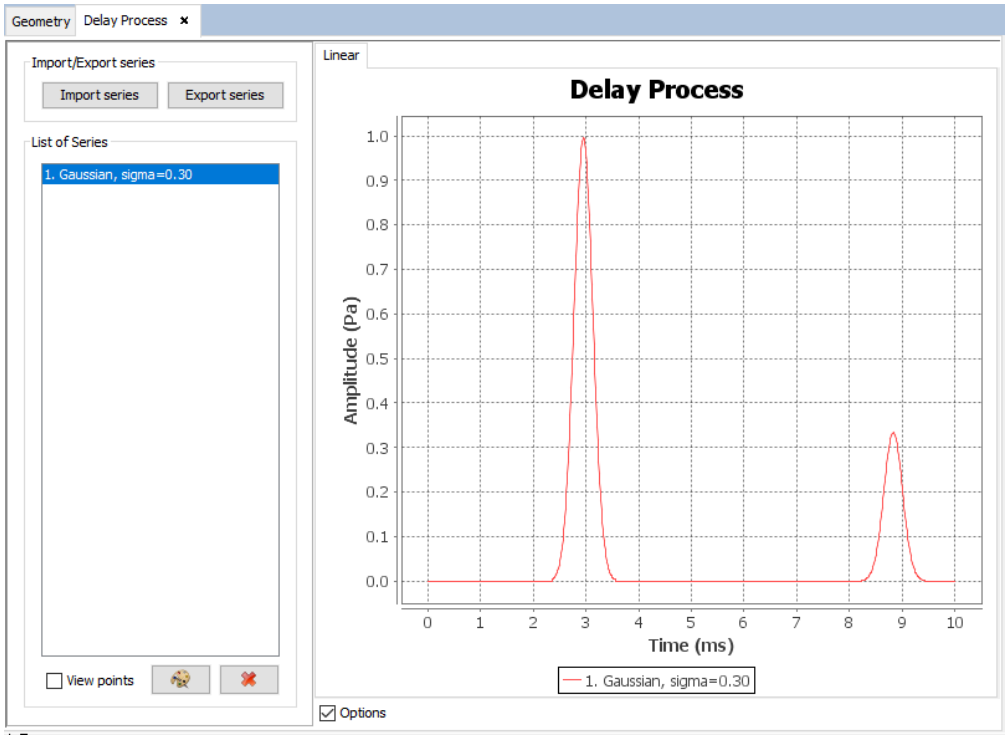


Figure 1875: Plot of the delay process with a Gaussian signal

In the plot viewer, the user can delete or change the color of the displayed series, as well as import or export series from/to text files.

Power Process

This option allows the user to visualize the power of the emitted sound waves as received at each observation point. When the user selects this option, the following panel is shown:

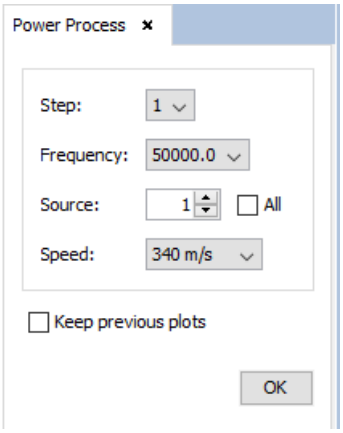


Figure 1876: Power process options panel

In this panel, the user needs to select the step, frequency and speed of sound the results are visualized for. The user can select a specific source or compute the sum of the results of each one by checking the "All" option.

The "Keep previous plots" option allows the user to choose whether they want to preserve the previously plotted graphs (which could be useful if the user wants to compare several plots) or they want to clear all previous plots.

When the user presses the "OK" button, a graph like the following will appear, showing the power of the sound waves for each observation point:

Figure 1877: Plot of the power process

14.11.5 Coupling

This menu allows the visualization of the coupling values between each source and receiver pair. As such, this submenu is only visible when a Coupling simulation has been executed.

The available options under this menu are the following:

- **Coupling Values** This options presents a table showing the coupling values for each source-receiver pair and each simulation frequency. The user can select the speed of sound and step to show the results for.
- **Coupling by Frequency** This option shows a plot of the coupling values by frequency of simulation. The step, speed of sound and source-receiver pair are set by the user.
- **Coupling by Step** This option shows a plot of the coupling values by parametric step. The frequency, speed of sound and source-receiver pair are set by the user.
- **Coupling by Speed** This option shows a plot of the coupling values by speed of sound. The frequency, step and source-receiver pair are set by the user.
- **Coupling Text Files** This option allows the visualization of the coupling values for a given step and speed of sound in text format.

Coupling Values

This option allows to visualize the coupling values for each source-receiver pair at each frequency, for a user-specified parametric step and speed of sound. The contents of the table will automatically reflect the changes in any of these parameters.

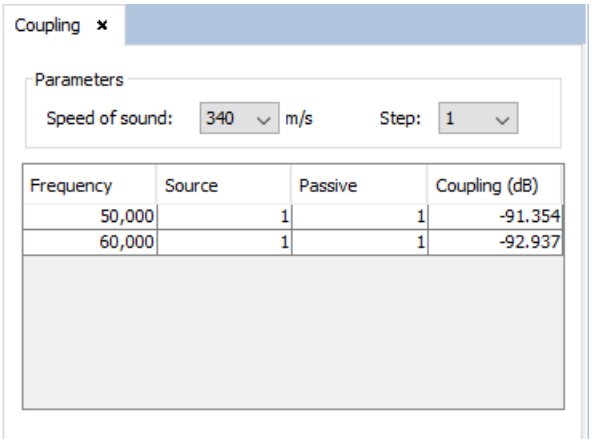


Figure 1878: Coupling values panel

Coupling Charts

The options "Coupling by Frequency", "Coupling by Step" and "Coupling by Speed" under the Results -> Coupling menu allow the user to analyze the dependency the coupling values have with the frequency, the geometry and the speed of sound, respectively, while keeping the other parameters constant. When selecting one of these options, a panel like the following will be shown:

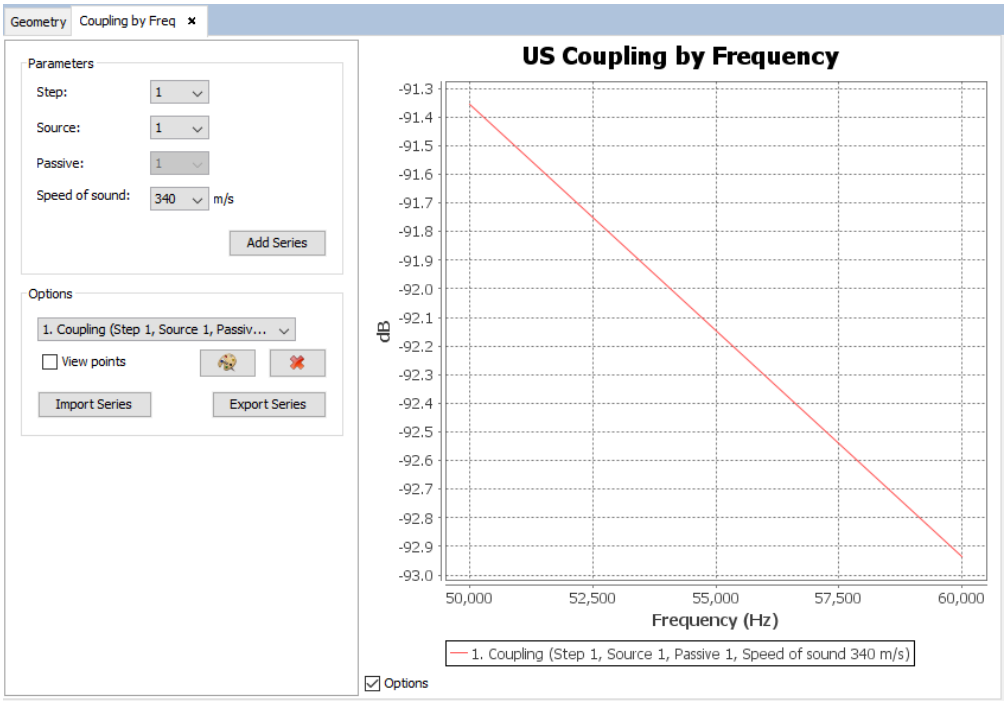


Figure 1879: Coupling by Frequency panel

In the left panel, the user can select the independent parameters (those other than the one the coupling values are plotted against -- in the previous figure, that would be the frequency). This allows the user to configure the series they want to plot.

The selected series can be added to the plot by pressing the **Add Series** button. It is also possible to modify the look of a series previously added to the chart. You can change the color of a particular series by selecting it in the list and pressing the **Change Colors**.

Series can be imported and exported. To save a series click on the **Export series** button. It is possible to load previously exported series by pressing the **Import Series** button and locating the file where the results were saved in.

Coupling Text Files

This option shows the coupling results as a text file. First, a panel will appear after selecting this option. In this panel, the user needs to select the parametric step and the speed of sound they want to show the results for. A table will be presented with the coupling values. To save the results click on the **Save as...** button below the text file.

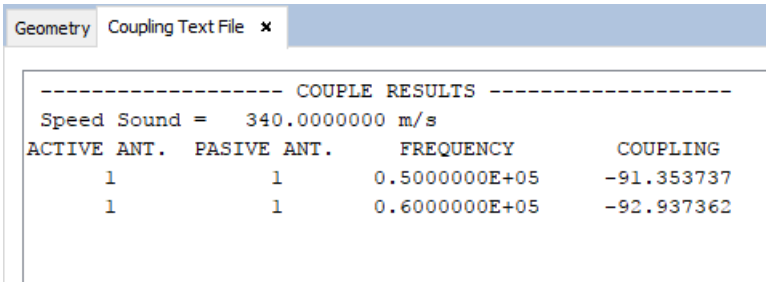


Figure 1880: Showing the result text file

14.11.6 Doppler Spectrum

This menu offers various options to analyze the Doppler Spectrum through the observation points previously defined in the Output section.

The following options are available:

- **Chart** Display a plot that relates the Doppler spectrum.
- **Text Files** Displays the numeric results obtained in the simulation process in text format.

Chart

This command plots the Doppler frequency shift computed. Several cuts and frequencies can be plotted in the same graphic. To plot a graph, the user can select the source and observation point. A new graph will be plotted with the button ' **Add Series**'. In the case that the user has set up a parametrized geometry, it is possible to select the step to visualize the results for.

It is also possible to delete a series that is selected by the user from the list with the **' Remove Series '**. The display option also allows changing the color of the series and display points. The buttons ' **Import Series**' and ' **Export Series**' are used for importing and exporting the selected series in 'List of Series' to a text file.

This panel displays a plot with the results and several options for adding and removing extra series from the plot.

When initially open, the panel will display a default series in the plot that will represent the measurements for a particular component, antenna, observation point and step. More series can be added, by selecting component and step in the control panel that appears on the left side of the plot.

- **Step**: the parametric step to select. These should have been defined prior to the calculation when designing the simulation.
- **Antenna group** The antenna to show the doppler spectrum results for. The user can select a specific antenna group or they can choose to show the results for all antennas.
- **Observation group** The observation points group that contains the point that the user wants to see the results for.
- **Observation point** The observation point to visualize the results for. This drop-down list shows the observation points included in the selected observation group.

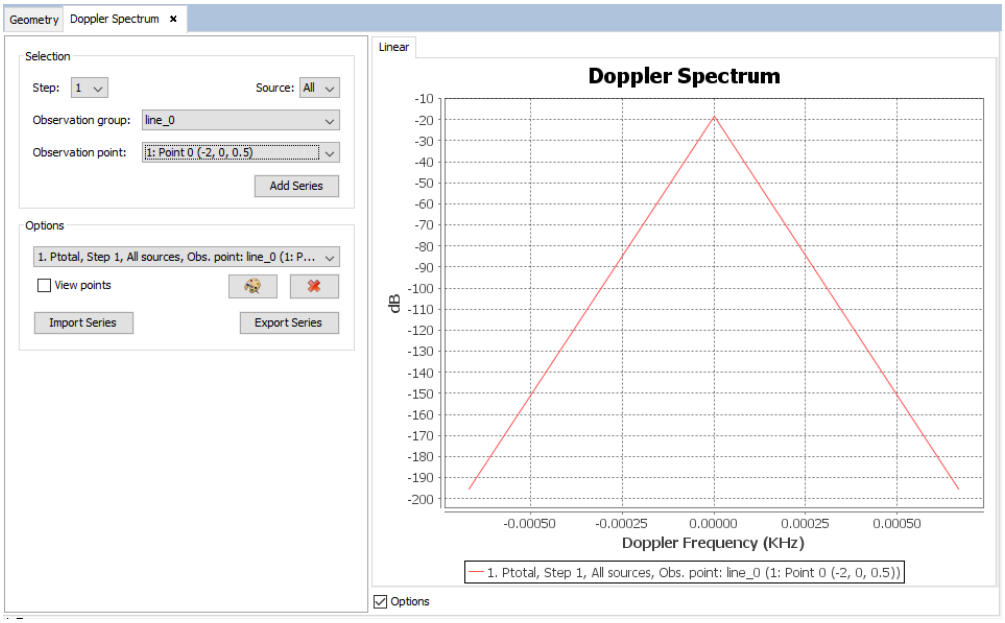


Figure 1881: Doppler Spectrum panel

Text Files

When the user selects this option, the following panel will appear:

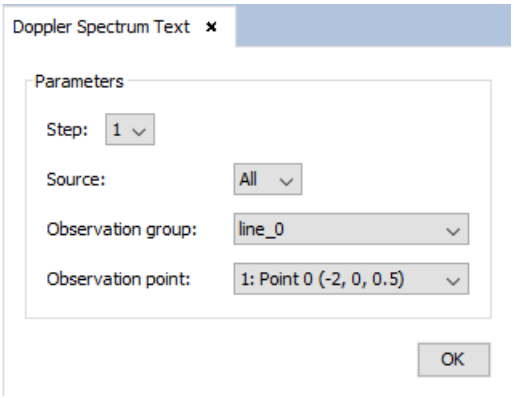


Figure 1882: View Text Files panel

In this panel, the user can select the step, the source and the observation point the results will be shown for. Then, the text file is generated and opened with newFASANT text file viewer.

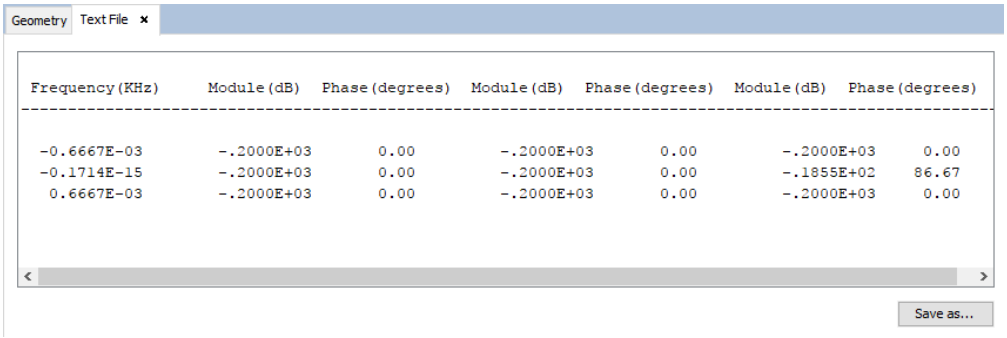


Figure 1883: Showing the result text file

The user can export the text file by selecting the "Save as..." button.

14.12 Tools Menu

For information about the Tools menu, refer to Tools.

14.13 Help Menu

For information about the Help menu, refer to Help Menu.

14.14 Training Examples

14.14.1 Example 1: Getting Started

In this section, a simple example of a Near Field simulation will be shown. This simulation will be performed considering a cube as the geometry. An Ultrasound Source will be defined as an ultrasound pattern file (also called Annex 1: DUS File Format).

Step 1: The first step consists of creating a new data file where all the parameters of the simulation will be saved. Use the command **New** from the **File** menu. Select **US** type.

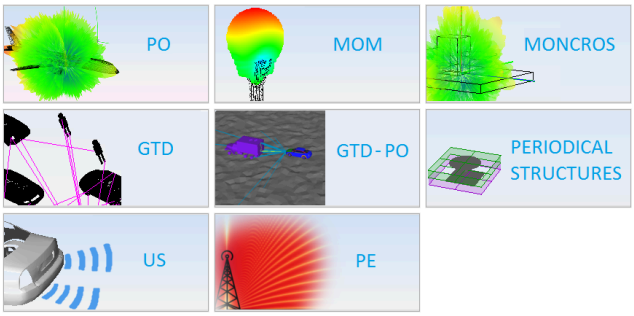


Figure 1884: Selection Method dialog box

Step 2: The second step is to generate the box. Select **GeometrySolidBox**. It will be located at (0,0,0) and the dimensions are 1x1x1 m.

```
command> box
First corner of base [x y z]: 0 0 0
Base size [width depth]: 1 1
Height [double]: 1
command>
```

Figure 1885: Create box

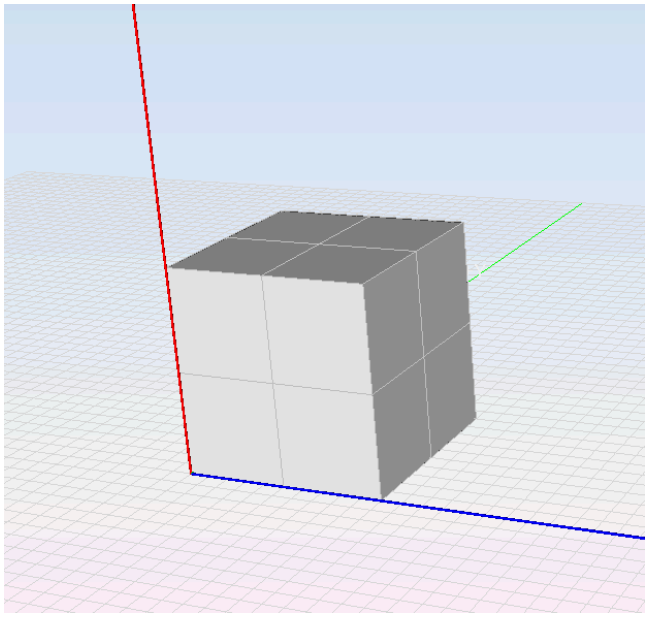


Figure 1886: Box situated at (0,0,0)

Step 3: The next step is to set the main parameters of the simulations, which are located in **Simulation Parameters Menu** > **Simulation**.

A dialog appears. In this menu, the frequency, kind of simulation, speed of sound sweep, effects to calculate, and other parameters must be defined before running the simulation. In this example, a radiation pattern file in far field is simulated. The user will introduce:

- **Near field** in **Simulation Type**.
- The number of frequency points and the frequency in this example 1 and 50 kHz.
- The **Speed of Sound Sweep** will be 340 m/s
- The effect order will be 1.

The rest of the options are the default ones.

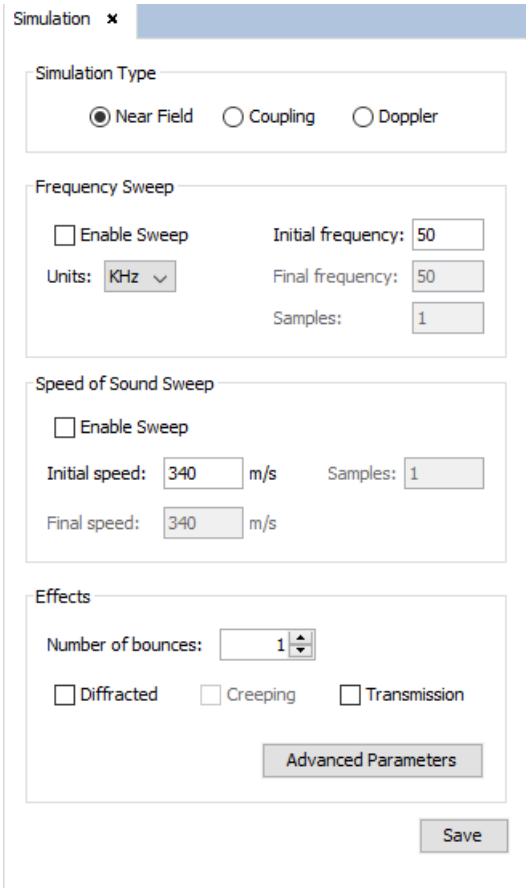


Figure 1887: Parameters dialog box

Step 4a: To create a new ultrasound pattern file (`.dus`), select the **Source** menu → **DUS Source Editor**. The user can find more info about DUS Source format at Annex 1: DUS File Format.

In this case, an omnidirectional pattern file has been edited.

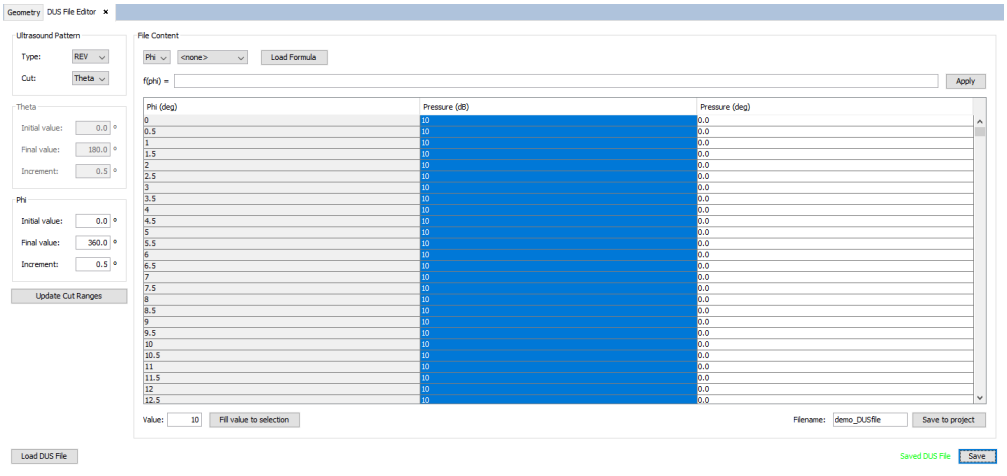


Figure 1888: DUS file Editor

Step 4b: Import the ultrasound pattern (DUS file) to the current project. In this example, the user has to import the `demo_DUSfile.dus` file.

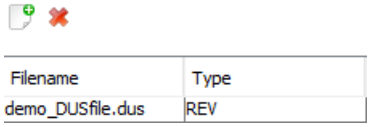


Figure 1889: Import DUS file

Step 5: Define the position of the ultrasound pattern over the cube clicking on **Source** > **Ultrasound Pattern**.

Set the ultrasound pattern file. Click on the add button and add the previous imported file.

Set the **Amplitude** and **Phase**.

Set the position of the source on (0.5, 0.5, 1.2).

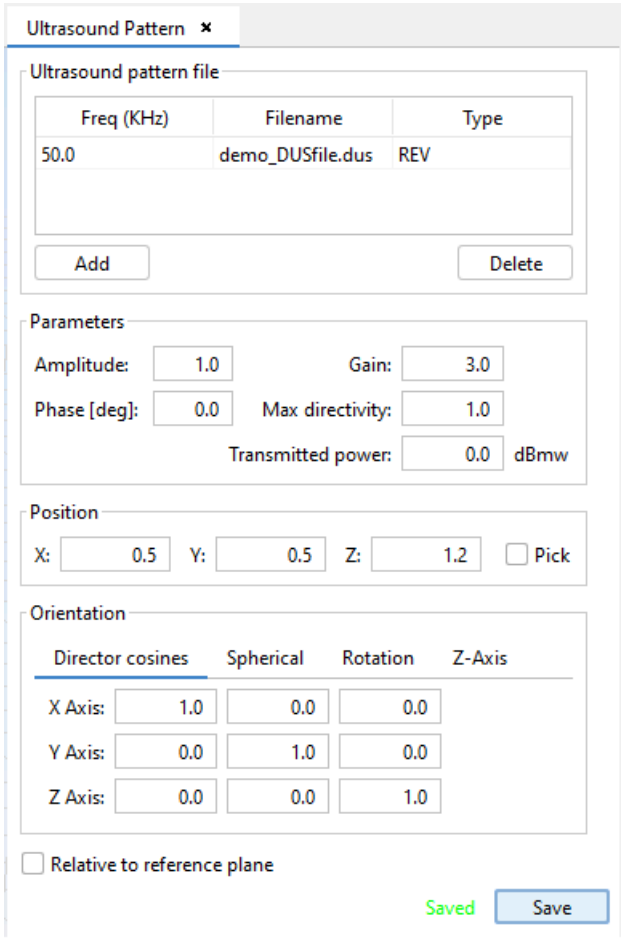
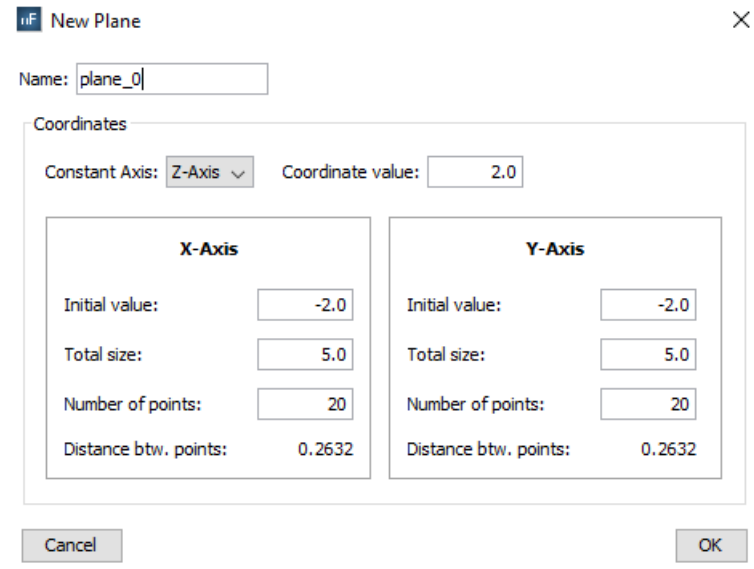
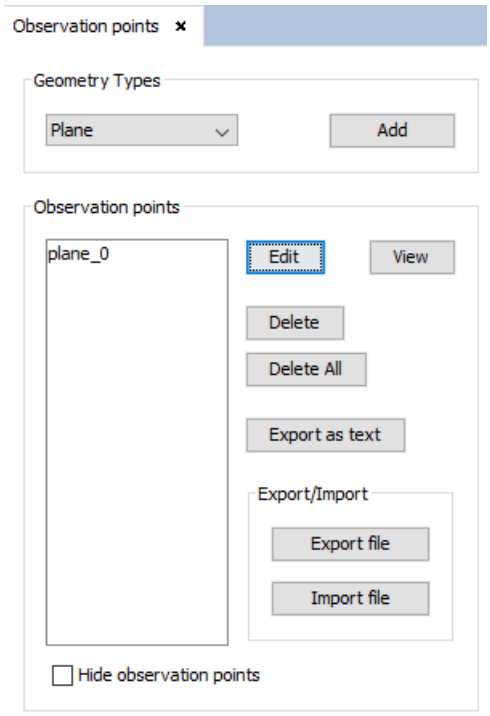


Figure 1890: Ultrasound Pattern menu

Step 6: The next step is to define the position of the observations points. Select **Observation points** from **Output** menu. In **Geometry Types** dialog, select **Plane** and click on the **Add** button.





Step 7: To mesh the geometry click on **Meshing** > **Create visibility matrix** and select the number of processors.

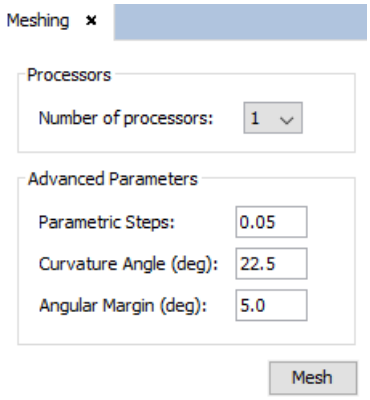


Figure 1891: Processors option for meshing

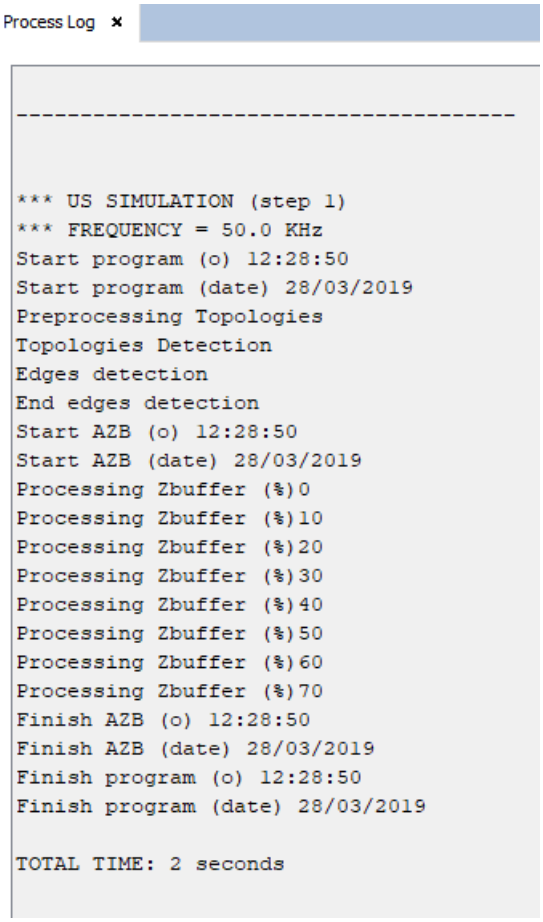


Figure 1892: Mesh

Step 8: Execute the simulation by selecting **Calculate** > **Execute** and introduce the number of processors to be used.

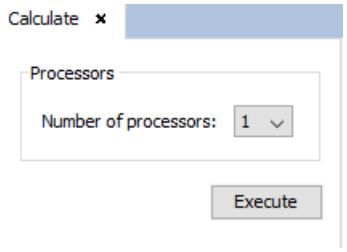


Figure 1893: Processors options for Execute

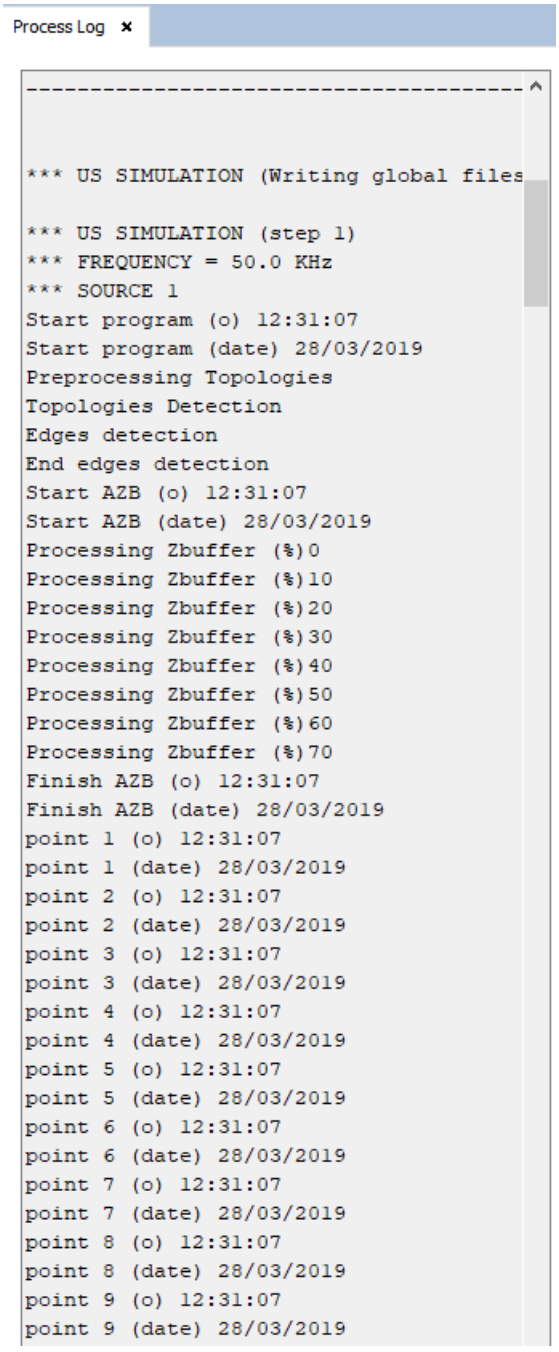


Figure 1894: Execute

Step 9: After obtaining the results, choose view ray from show results. It is possible to visualize order 1 rays. It is possible to represent the ray tracing interacting with the geometry.

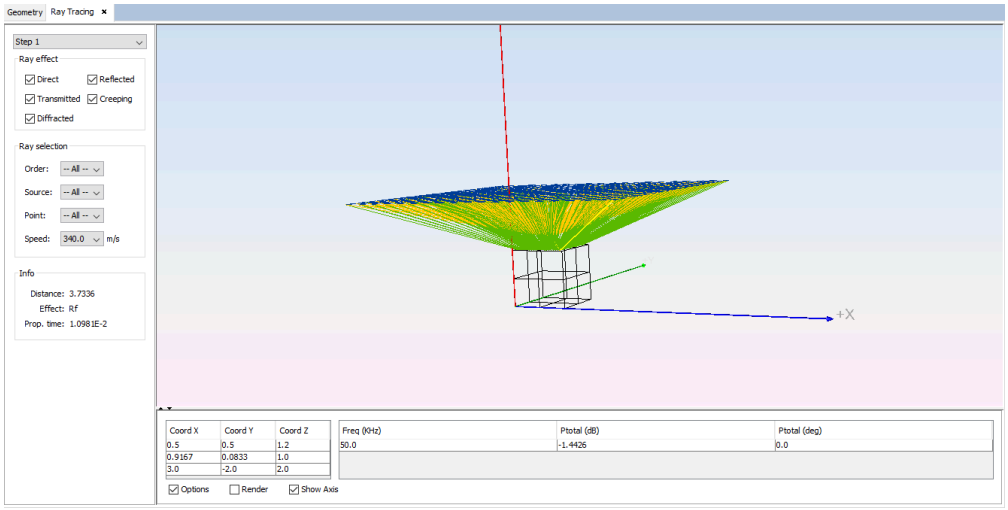


Figure 1895: Ray Viewer dialog box

Step 9: It is possible to select a ray to show its information. Clicking in one of the ray will fill its information in the bottom of the panel.

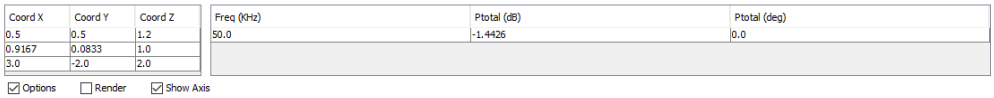


Figure 1896: Viewing the ray information

Step 10: To visualize the charts click on **Show Results** > **Near Field** > **View Cuts**, after which the following dialog will be shown:

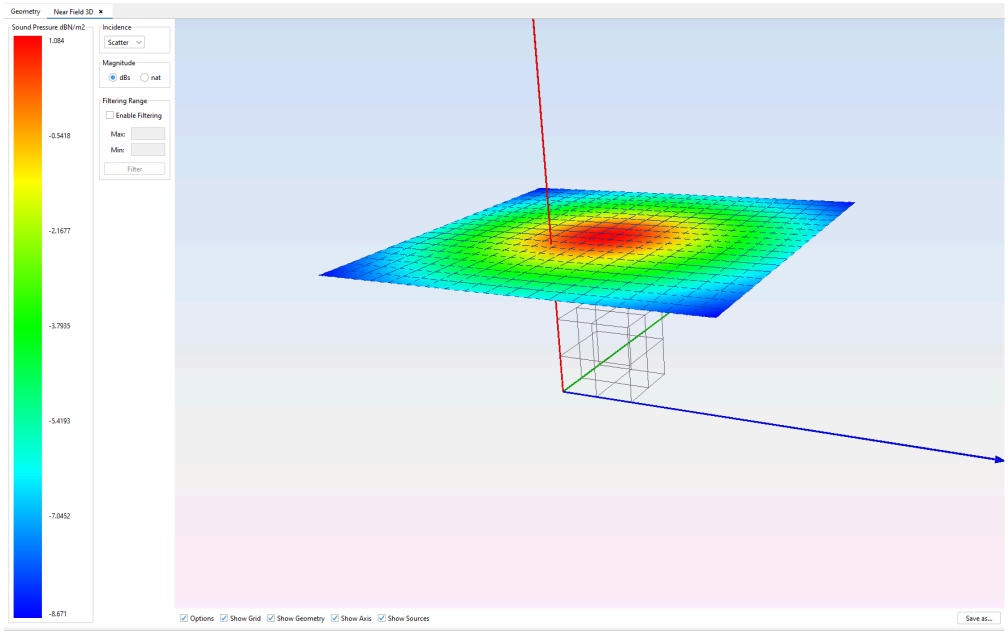


Figure 1897: Near field

14.15 Annex 1: DUS File Format

DUS files are ultrasound pattern files, and their function is analogous to DIA files (radiation pattern) used by other modules of newFASANT. DUS files are used to define the sound waves emitted by an ultrasound source in different directions. These directions are specified in spherical coordinates and, depending on whether the pressure sound field emitted is constant on either phi or theta coordinates, the DUS file format is different. On the one hand, when the ultrasound pattern depends only on theta or phi, it is said to be a revolution (REV) DUS file, defined as a phi or theta cut respectively. On the other hand, an ultrasound pattern with an emitted field depending on both theta and phi is said to be a 3D DUS file.

In the following subsections, the format of both types of DUS files is defined.

Note It is not necessary to know the DUS file format in order to work with this kind of files in newFASANT. The “DUS Files” option under the “Tools” menu allows the visualization and edition of the information contained in DUS files.

14.15.1 REV DUS Files

Revolution defined DUS files can be classified into two types those defined by a phi cut and those defined by a theta cut. The cut determines the coordinate upon which the emitted field remains constant. Thus, when a REV DUS file is defined by theta cut, the field does not depend on the theta coordinate but only on the phi coordinate, and vice versa.

The file format for a REV DUS file is very simple and consists of the following information (for each list item, a line is required):

- A header line usually reserved for information on the program that generated the DUS file and information about the file.
- The **type of DUS file**. In this case, it is the string "REV" (without quotes).
- The **type of cut**, "TH" if it is a theta cut or "PH" if it is a phi cut.
- The component of the field that is defined in this file. It always is "Ptotal" (without quotes).
- A comment line, ignored by the software, that can hold any content.
- Several values **separated by whitespaces**. These values are, in order:
  - Initial theta value.
  - The number of theta samples.
  - The theta increment (the difference between two consecutive theta samples).
  - Initial phi value.
  - The number of phi samples.
  - The phi increment (the difference between two consecutive phi samples).
- Please note that either the first three values or the last three values will be ignored by the program if the file is defined as a theta cut or a phi cut, respectively. However, all the values must be provided in the file.
- Another comment line that can hold any content.

- For each theta/phi sample (depending on whether is a phi or theta cut, respectively), three values, separated by whitespace, are provided the theta/phi value, the amplitude of the emitted sound pressure wave (in dBPa) and the phase of the sound pressure wave (in degrees).

14.15.2 3D DUS Files

3D DUS files are very similar to the revolution-defined DUS files. However, in this case, the sound pressure wave depends on both theta and phi coordinates and therefore:

- There is only one type of 3D DUS files, unlike in REV files.
- It is necessary to specify both theta and phi values for each sample of the sound pressure emitted field.

In consequence, the 3D DUS file format is defined as follows (each list item corresponds to one line in the file):

- A header line usually reserved for information on the program that generated the DUS file and information about the file.
- The type of DUS file. In this case, it is the string "3DE" (without quotes).
- The component of the field that is defined in this file. It always is "Ptotal" (without quotes).
- A comment line, ignored by the software, that can hold any content. Several values separated by whitespaces. These values are, in order:
  - Initial theta value.
  - The number of theta samples.
  - The theta increment (the difference between two consecutive theta samples).
  - Initial phi value.
  - The number of phi samples.
  - The phi increment (the difference between two consecutive phi samples).
- Another comment line that can hold any content.
- For each combination of theta and phi samples, four values, separated by whitespace, are provided the theta value, the phi value, the amplitude of the emitted sound pressure wave (in dBPa) and the phase of the sound pressure wave (in degrees).

14.16 Annex 2: Closed-Loop Simulation Support in the US Module

Besides the usual GUI execution mode and server execution mode, newFASANT includes a third model of execution closed-loop execution. This mode allows the user to use the capabilities in the program without the need of using the graphical user interface (except for creating the geometries, as explained in the following sections). This mode of execution is intended as a way of using the simulation capabilities of newFASANT in headless machines (without graphical environments) or as an aid to external simulation workflows such as closed-loop simulations, where it is needed to automate the generation of simulation cases.

The following sections explain the usage of the "closed-loop simulations" feature.

14.16.1 Simulation Workflow

When using newFASANT in closed-loop mode, the user needs to provide a path to a NFIL file that contains the **specification** of the simulation case. This file consists of a series of directives and commands that are used for setting up the simulation case (i.e simulation parameters, geometry, sources and observation points). An example of NFIL file is provided below:

```
@version 1
@log stdout
@set_module us
set_simulation_type doppler
# Geometry.
add_geom box.nur box.mat
# Sources.
add_sound_source source.mat 1.0 0.0 50.0 unit.dus
# Simulation parameters.
set_frequency 50.0
#set_freq_sweep 50.0 60.0 3
set_speed_sound 340.0
set_bounces 1
set_effects Df Tx
# Observations.
add_point_observation obs.dis 0.0 0.0 0.0
# Doppler.
set_doppler_freq_bin 1.0
set_doppler_velocity_obj box 1.0 0.0 0.0
set_doppler_velocity_source 1 -1.0 0.0 0.0
set_doppler_velocity_obs 1 2.0 0.0 0.0
# Calculation.
set_processors 4
```

NFIL files are processed line by line. Empty lines and lines starting with the "#" character (comments) are ignored by the parser. Every other line consists in the name of a command followed by zero or more arguments, the number of which depends on the specific command or directive. **Commands whose name begins with "@" are called directives**, and the purpose of this type of commands is to configure details relating to how the closed-loop simulation is run rather than the simulation itself. As such, directives MUST always appear at the start of the file, and therefore no regular command can appear before any directive. Any file that does not comply with this rule will be considered invalid.

NFIL files are designed to be written only once even if multiple simulations are run within the same closed-loop simulation workflow. That happens because NFIL files only encode "static" information about the simulation geometry, observation points, sources, receivers and simulation parameters. The parameters of the simulation that are subject to change between simulations, such as the position and rotation of geometry, observation points and sources, are factored out into files other that the NFIL files DIS files and MAT files, which are the ones that are supposed to be modified by the external tool between simulations (usually in response to the results of the previous simulation). DIS files only encode information about the translation of a particular object, while MAT files encode translation, rotation and/or scaling by specifying an affine transformation matrix. Depending on the command, it will accept either a DIS file or a MAT file. Please check the "Command Reference" section for more details.

An example of the contents of a DIS file would be the following. This file encodes a translation of two meters along the X axis:

```
2.0 0.0 0.0
```

On the other hand, an example of the contents of a MAT file is presented below. This file represents a scaling along the X axis (by a factor of 2, as defined by the top-left 3x3 submatrix) and a translation of 3 units along both the X axis and the Y axis (as defined by the last column of the matrix):

```
2.0 0.0 0.0 3.0
0.0 1.0 0.0 3.0
0.0 0.0 1.0 0.0
0.0 0.0 0.0 1.0
```

The geometry for the simulation case provided to the add_geom command **must be in newFASANT's NUR file format**. Therefore, the user needs to use newFASANT GUI to design and export the geometry as a NUR file in order to be used in a closed-loop simulation workflow. The NUR file contains the information of the materials assigned to each surface of the geometry, so these do not need to be configured in the NFIL file.

After the NFIL file and all companion (NUR, DIS and MAT) files have been created, the user can run newFASANT in closed-loop mode by running the following command (using the command-line):

```
C:\newFASANT\newFASANT6.exe "C:\PathToNfil\spec.nfil"
```

The above command assumes that the installation directory of newFASANT is "C:\newFASANT" and that "C:\PathToNfil\spec.nfil" is the path where the NFIL file is located. Please note that **all relative paths referenced from the NFIL file will be relative to the folder where the NFIL is located** (in the previous example, that would be the "C:\PathToNfil" directory). This encourages a folder structure where all the files relating to the closed-loop simulation are located in the same directory.

When the above command is run, newFASANT will parse the NFIL file and build the simulation using the commands specified in this file. After the file is parsed, newFASANT will run the meshing and calculation process one after another. After the simulation is run, newFASANT will output the result files in the "mesh" and "result" folders inside the NFIL file folder.

To sum up, the following process should be followed when running a closed-loop simulation with newFASANT:

- Create an empty folder that will contain the files used in the simulation.

- Design and export as NUR the geometry to be used in the simulation. It is advisable to export each different object in the simulation as a separate NUR file so that these different objects can be transformed independently between simulations. These NUR files should be saved to the folder created in the previous step.
- Create the NFIL file that will be the specification of the simulation. Check the "Command Reference" section for detailed information about the available commands for use in NFIL files.
- The previous NFIL file will contain references to DIS and MAT files. Create these files with the translations and transformations desired for each of the objects, sources and observation points they refer to in the NFIL file.
- Add the additional files referred to by the NFIL file, such as DUS files, to the NFIL folder.
- Using the command-line, call the newFASANT6.exe executable with the path to the NFIL file.
- Inspect the result files in the "mesh/" and "result/" folders inside the NFIL folder and, if desired, modify the DIS and MAT files and re-run the simulation. Repeat this step as many times as needed.

14.16.2 Command Reference

The current section contains the detailed description of the directives and commands that can be used within NFIL files.

Directives

Directives are special commands that configure the way simulations are run, rather than the simulations themselves. In this sections, the different directives are described:

Directive @version

Description Specifies the version of the NFIL file format used. This directive is intended to be used for backwards-compatiblity reasons. At the moment, it should be passed the argument "1".

Example:

```
@version 1
```

Directive @set_module

Description Sets the module of the simulation. Currently, only US module simulations are supported.

Arguments This directive only takes one argument the name of the module that will be used.

Example:

```
@set_module us
```

Directive @log

Description Allows to configure the logging of messages during the simulation.

Arguments The first argument can be "off" if the user does not need logging, "stdout" if the user wants to see the messages of the simulation in the standard output, or "file" if the user wants to

output messages to a file. If the first argument is "file", the directive will accept an additional argument specifying the path to the output log file.

Examples:

```
@log off
```

```
@log stdout
```

```
@log file log.txt
```

Directive @output_nfp

Description Allows the creation of a NFP (newFASANT project) file containing the generated simulation. This NFP can later be opened with the newFASANT GUI. This functionality is intended to be used with debugging or troubleshooting purposes, as the generation of the NFP project file takes some time.

Arguments This directive only takes one argument either "off" if the user wants to disable this functionality (although it is disabled by default) or the path to the resulting NFP file.

Examples:

```
@output_nfp off
```

```
@output_nfp project.nfp
```

Commands

In this section, the available commands available for US simulations are documented. Please note that all paths are relative to the folder where the NFIL file is located.

Command add_geom

Description Adds the geometry from a NUR file to the simulation.

Arguments The add_geom commands accepts two arguments:

- The path to the NUR file with the geometry to be added to the simulation.
- The path to the MAT file that defines the affine transformation applied to the geometry.

Example:

```
add_geom car.nur car.mat
```

Command add_sound_source

Description Adds a new ultrasound source to the simulation.

Arguments This command accepts the following arguments, in order:

- The path to the MAT file that defines the affine transformation applied to the source.
- The amplitude of the ultrasound source.
- The phase of the ultrasound source.

- The frequency of operation of the DUS file given in the next argument, in kHz.
- The DUS file that the defines the ultrasound pattern.

The later two arguments can be repeated if the user wants to specify DUS pattern files for different frequencies.

Examples:

```
add_sound_source source.mat 1.0 0.0 50.0 pattern.dus
```

```
add_sound_source source.mat 1.0 0.0 50.0 pattern50khz.dus 60.0 pattern60khz.dus
```

Command set_simulation_type

Description Sets the simulation type of the current US simulation. Note that some other commands will only work when the simulation is of a certain type.

Arguments This command only takes one argument the simulation type. The simulation type can either be "near" for Near Field simulations, "coupling" for coupling simulations or "doppler" for doppler simulations.

Examples:

```
set_simulation_type near
```

```
set_simulation_type coupling
```

Command set_frequency

Description Sets the frequency of the simulation.

Arguments This command only takes one argument the frequency in kHz.

Example:

```
set_frequency 50.0
```

Command set_freq_sweep

Description Sets a frequency sweep for the simulation.

Precondition The simulation type must not be Doppler.

Arguments This command takes the following arguments, in order:

- The initial frequency of the simulation, in kHz.
- The final frequency of the simulation, in kHz.
- The number of frequency samples in the interval of frequencies.

Example:

```
set_freq_sweep 50.0 60.0 3
```

Command set_speed_sound

Description Sets the speed of sound of the simulation.

Arguments This command only takes one argument the speed of sound in m/s.

Example:

```
set_speed_sound 340.0
```

Command set_speed_sweep

Description Sets a sweep of speeds of sound for the simulation.

Precondition The simulation type must not be Doppler.

Arguments This command takes the following arguments, in order:

- The initial speed of sound, in m/s.
- The final speed of sound, in m/s.
- The number of samples in the interval of speeds.

Example:

```
set_speed_sweep 340.0 350.0 3
```

Command set_bounces

Description Sets the number of effects per ray that will be calculated in the ray-tracing.

Arguments This command only takes one argument the number of desired effects.

Example:

```
set_bounces 2
```

Command set_effects

Description Sets the types of effects that will be taken into account when calculating rays in the ray-tracing. Note that direct and reflected rays are ALWAYS calculated.

Arguments This command takes the list of effects to be calculated, or "none" if the user does not want to calculate any effects other than direct and reflected. The available effects are "Tx" (transmission) and "Rf" (refraction).

Examples:

```
set_effects Tx Df
```

```
set_effects none
```

Command add_passive

Description Adds a new receiver to the simulation.

Precondition Simulation must be of type Coupling.

Arguments This command takes the following arguments:

- Path to the MAT file that defines the position and orientation of the receiver.
- Path to the DUS file that defines the ultrasound pattern of the receiver.

- The effective area of the receiver, in m^2.

Example:

```
add_passive passive.mat passive.dus 0.01
```

Command set_doppler_freq_bin

Description Sets the Doppler frequency bin value.

Precondition Simulation must be of type Doppler.

Arguments This command takes only one argument the Doppler frequency bin, in Hz.

Example:

```
set_doppler_freq_bin 1.0
```

Command set_doppler_velocity_obj

Description Sets the doppler velocity for a geometry object.

Precondition Simulation must be of type Doppler.

Arguments This command takes the following arguments:

- The name of the object. The name of an object is the name set to it before exporting to NUR.
- The three components (X, Y, Z) of the vector that defines the velocity, in m/s.

Example:

```
set_doppler_velocity_obj car 26.0 0.0 0.0
```

Command set_doppler_rotation_obj

Description Sets the doppler rotation for a geometry object.

Precondition Simulation must be of type Doppler.

Arguments This command takes the following arguments:

- The name of the object. The name of an object is the name set to it before exporting to NUR.
- The three components (X, Y, Z) of the first point of the rotation axis.
- The three components (X, Y, Z) of the second point of the rotation axis.
- The rotation speed, in rad/s.

Example:

```
set_doppler_rotation_obj box 0.0 0.0 0.0 0.0 0.0 1.0 0.5
```

Command set_doppler_velocity_source

Description Sets the doppler velocity for an ultrasound source.

Precondition Simulation must be of type Doppler.

Arguments This command takes the following arguments:

- The index (starting at 1) of the source the user wants to apply the velocity to. Indices are assigned sequentially to sources when they are created with the add_sound_source command.
- The three components (X, Y, Z) of the vector that defines the velocity, in m/s.

Example:

```
set_doppler_velocity_source 1 1.0 2.0 0.0
```

Command set_doppler_velocity_obs

Description Sets the doppler velocity for an observation point group.

Precondition Simulation must be of type Doppler.

Arguments This command takes the following arguments:

- The index (starting at 1) of the observation point group the user wants to apply the velocity to. Indices are assigned sequentially to observation point groups when they are created.
- The three components (X, Y, Z) of the vector that defines the velocity, in m/s.

Example:

```
set_doppler_velocity_obs 1 1.0 2.0 0.0
```

Command set_meshing_processors

Description Sets the number of processors that will be used for meshing.

Arguments This command only takes one argument the number of processors that will be used.

Example:

```
set_meshing_processors 4
```

Command set_executing_processors

Description Sets the number of processors that will be used for execution.

Arguments This command only takes one argument the number of processors that will be used.

Example:

```
set_executing_processors 4
```

Command set_processors

Description Sets the number of processors that will be used for meshing and execution. This command is a shortcut for setting the number of processors for both calculations without the need to call two commands.

Arguments This command only takes one argument the number of processors that will be used.

Example:

```
set_processors 4
```

Command add_point_observation

Description Adds an observation point to the simulation.

Precondition Simulation type must not be Coupling.

Arguments This command takes the following arguments:

- Path to the DIS file that defines the translation applied to this observation point.
- Coordinates (X, Y, Z) of the point. The result of applying the translation defined in the DIS file to this point will be the location of the observation point.

Example:

```
add_point_observation obs1.dis 2.0 1.0 0.0
```

Command add_cylinder_observation

Description Adds a cylinder (or cylindrical section) of observation points to the simulation.

Precondition Simulation type must not be Coupling.

Arguments This command takes the following arguments (all distances are expressed in meters):

- Path to the DIS file that defines the translation applied to this group of observation points.
- The coordinates (X, Y, Z) of the center of the cylinder's base.
- The radius of the cylinder.
- The number of points along the height of the cylinder.
- The height of the cylinder.
- The number of points along the circumference of the cylinder.
- The initial angle of the cylindrical section, in degrees.
- The final angle of the cylindrical section, in degrees.

Example:

```
add_cylinder_observation obs2.dis 0.0 0.0 0.0 2.0 11 20.0 10 0.0 180.0
```

Command add_line_observation

Description Adds a line of observation points.

Precondition Simulation type must not be Coupling.

Arguments This command takes the following arguments (all distances are given in meters):

- The path to a MAT or DIS file defining an affine transformation or a translation, respectively.
- The number of points of the line.
- The coordinates (X, Y, Z) of the initial point of the line.
- The coordinates (X, Y, Z) of the final point of the line.

Examples:

```
add_line_observation obs3.dis 21 10.0 0.0 0.0 20.0 0.0 0.0
```

```
add_line_observation obs3.mat 11 0.0 0.0 0.0 0.0 10.0 0.0
```

Command add_parallelogram_observation

Description Adds a parallelogram of observation points.

Precondition Simulation type must not be Coupling.

Arguments The command takes the following arguments (all distances are given in meters):

- The path to a MAT or DIS file defining an affine transformation or a translation, respectively.
- The coordinates (X, Y, Z) of three points defining the vertices of the parallelogram. The fourth point is calculated by using the other three points.
- The number of points along the direction from the first to the second point.
- The number of points along the direction from the second to the third point.

Example:

```
add_parallelogram_observation obs4.mat 2.0 2.0 0.0 0.0 0.0 0.0 4.0 0.0 0.0 11 11
```

Command add_plane_observation

Description Adds a plane of observation points to the simulation.

Precondition Simulation type must not be Coupling.

Arguments The command takes the following arguments (all distances are given in meters):

- The path to a DIS path defining the translation applied to this group of observation points.
- The constant coordinate of the plane (can either be "x", "y" or "z").
- The value of the constant coordinate of the plane.
- The initial value of the first non-constant coordinate of the plane.
- The total size of the plane along its first non-constant coordinate.
- The number of points along the first non-constant coordinate of the plane.
- The initial value of the second non-constant coordinate of the plane.
- The total size of the plane along its second non-constant coordinate.
- The number of points along the second non-constant coordinate of the plane.

Example:

```
add_plane_observation obs5.dis y 2.0 1.0 5.0 5 2.0 6.0 5
```

Command add_sphere_observation

Description Adds a sphere of observation points to the simulation.

Precondition The simulation type must not be Coupling.

Arguments The command takes the following arguments (all distances are given in meters):

- The path to the DIS file with the translation applied to this group of observation points.
- The coordinates (X, Y, Z) of the center of the sphere.
- The initial radius of the sphere.
- The increment between two consecutive radius values.
- The number of radius values.

- The number of points along the theta direction.
- The initial theta value, in degrees.
- The final theta value, in degrees.
- The number of points along the phi direction.
- The initial phi value, in degrees.
- The final phi value, in degrees.

Example:

```
add_sphere_observation obs6.dis 2.0 0.0 0.0 10.0 1.0 3 11 0.0 90.0 21 0.0 180.0
```

List of Acronyms and Abbreviations

15

View the list of commonly used acronyms in newFASANT.

This chapter covers the following:

- 15.1 Acronyms (p. 1612)

15.1 Acronyms

AZB	Angular Z-Buffer
BiCGStab	Biconjugate Gradient Stabilized Method
CBFM	Characteristic Basis Function Method
CFIE	Combined Field Integral Equation
EFIE	Electric Field Integral Equation
FSS	Frequency Selective Surface
GMRES	Generalized Minimal Residual Method
GO	Geometrical Optics
GTD	Geometrical Theory of Diffraction
IR	Infrared
ISAR	Inverse Synthetic Aperture Radar
MBF	Macro Basis Function
MFIE	Magnetic Field Integral Equation
MLFMA	Multilevel Fast Multipole Algorithm
MoM	Method of Moments
NURBS	Non-Uniform Rational B-Spline
PEC	Perfect Electric Conductor
PO	Physical Optics
PSO	Particle Swarm Optimization
PTD	Physical Theory of Diffraction
RAM	Radar Absorbing Material
RCS	Radar Cross Section
SVP	Space Volumetric Partitioning
UTD	Uniform Geometrical Theory of Diffraction

Index

A

acronyms *1612*
analyzeCurvature *437*
analyzeEdges *438*
analyzeNakedBorders *439*
angle *440*
antenna *670*
    delete *679*
    display size *679*
    edit *678*
    passive *1056*
antenna coupling *724*
antenna coupling by frequency *725*
antenna coupling by step *726*
antenna coupling text files *727*
antenna feeding
    import *680*
arc *441*
area *443*
array *72*, *444*
    3D *72*
ArrayOnSuerface *73*
arrayOnSurface *446*
axis *451*

B

batch *256*
batch utility *27*
bentDipoles *452*
bistatic RCS
    box *913*
boolean operations *68*
booleanDifference *453*
booleanInside *456*
booleanIntersection *458*
booleanOutside *461*
booleanSplit *464*
booleanUnion *467*
booleanXOR *469*
box *472*

C

calculator *230*, *236*

capacitance *474*
circle *475*
circularHole *476*
command
- analyzeCurvature *437*
- analyzeEdges *438*
- analyzeNakedBorders *439*
- angle *440*
- arc *441*
- area *443*
- array *444*
- arrayOnSurface *446*
- axis *451*
- bentDipoles *452*
- booleanDifference *453*
- booleanInside *456*
- booleanIntersection *458*
- booleanOutside *461*
- booleanSplit *464*
- booleanUnion *467*
- booleanXOR *469*
- box *472*
- capacitance *474*
- circle *475*
- circularHole *476*
- cone *477*
- connect2Surfaces *479*
- connectCurves *482*
- coons *483*
- copy *486*
- cross *488*
- crossHole *490*
- curve *491*
- curveFunction *493*
- curveInterpolation *495*
- cylinder *497*
- delete *499*
- disk *500*
- distance *501*
- duplicate *502*
- ellipse *503*
- ellipsoid *505*
- exit *507*
- explode *508*
- export *509*
- extendHole *510*
- extrude *513*

extrudeNormal *515*
extrudeSurface *517*
flare *519*
group *521*
help *522*
hexagon *523*
hyperbola *524*
hyperboloid *525*
import *526*
interdigitalLines *527*
invertNormals *528*
length *530*
line *531*
localTransform *532*
meshFromSurfaces *535*
meshToSurfaces *536*
move *537*
ogive *539*
openRing *542*
parabola *543*
paraboloid *544*
perpendicular *545*
pipe *548*
planarSurface *550*
plane *552*
point *554*
pointCloud *555*
pointsFromCurve *557*
pointsFromCurveByDistance *558*
pointsFromSurface *559*
polygon *561*
project *563*
rebuildCurve *566*
rebuildSurface *567*
rectangle *570*
rectangularHole *572*
rename *573*
revolve *574*
rhomboid *577*
ring *579*
rotate *580*
scale *582*
scale1D *584*
scale2D *587*
scale3D *590*
scaleNonUniform *593*
script *596*

segment *597*
semicircle *598*
set *599*
skinned *601*
sphere *603*
spiral *605*
spiralCross *606*
splitByProjection *607*
splitCurves *610*
splitRing *612*
splitSurfaceByIsoparam *613*
splitSurfaceByIsoparamGrid *615*
splitSurfaces *616*
stripCross *618*
stripHexagon *619*
stripTrifilar *620*
surface *621*
surfaceCornerPoints *623*
surfaceEdges *624*
surfaceEllipse *626*
surfaceFromEdges *627*
surfaceFunction *630*
surfaceInterpolation *632*
surfaceIsoparamCurve *634*
surfaceSpiral *636*
sweep *637*
symmetric *638*
tangent *641*
tCross *642*
threeDipoles *643*
threeDipolesUnion *644*
torus *645*
trifilar *647*
trim *648*
unset *651*
untrim *652*
command-line shell *32*
cone *477*
connect2Surfaces *479*
connectCurves *482*
console *32*
coons *483*
copy *63*, *486*
coupling *724*, *859*, *919*
create mesh *861*
creeping *848*
cross *488*

crossHole *490*
curve *104*, *491*
  arc *115*
  circle *116*
  connect *107*
  control points *105*
  ellipse *117*
  function *107*
  hyperbola *118*
  interpolate points *106*
  parabola *118*
  perpendicular *110*
  polygon *112*
  rebuild *108*
  rectangle *111*
  regular polygon *113*
  rhomboid *114*
  segment *111*
  semicircle *116*
  spiral *119*
  star polygon *114*
  tangent *109*
curveFunction *493*
curveInterpolation *495*
cylinder *497*

D

delay process *716*
delete *63*, *499*
delete all *63*
DIA
  files *210*
DIA file
  examples *216*
  import *678*
  structure *214*
diagram correlation *227*
difference *69*
dipole *670*
dipole antenna *670*
dipole array *672*
disk *500*
displayaxes *54*
distance *501*
Doppler *667*, *848*

Doppler spectrum *721*, *923*
chart *722*
text files *723*
drag and drop *38*
duplicate *65*, *502*

E

edit
project parameters *36*
edit menu *62*
ellipse *503*
ellipsoid *505*
example
MIMO *757*, *763*
execute *1076*
exit *507*
explode *67*, *508*
export *95*, *509*
extendHole *510*
extrude *513*
extrudeNormal *515*
extrudeSurface *517*

F

far field *706*
view cuts *706*
view cuts by frequency *708*
view cuts by step *710*
files
DIA *210*
flare *519*
FMCW *933*

G

graphics
advanced options *259*
group *66*, *521*
GUI
overview *32*

H

help *258*, *522*
hexagon *523*
high performance computing (HPC) *29*
history *62*

HPC *29*
hyperbola *524*
hyperboloid *525*

I

ignore edges *699*
import *95*, *526*
    pattern files *856*
inside *71*
interdigitalLines *527*
intersection *69*
invert normals *91*
inverting normals *269*
invertNormals *528*

J

Java *240*
Java function *233*

L

layers *92*
length *530*
line *531*
load *254*
localTransform *532*

M

menu
    geometry *95*
mesh
    create visibility matrix *701*
    ignore edges *699*
    manual divisions *866*
    to surfaces *187*
    view *864*
meshFromSurfaces *535*
meshToSurfaces *536*
method of moments (MoM) *1046*
MIMO
    example *757*, *763*
MIMO results *730*
    channel correlation *730*
    channel transfer matrix *730*
monostatic RCS
    ship *906*

sphere / box *895*
truck *929*
move *64*, *537*
multiple points *99*
multipole antenna *677*, *853*
mutual coupling *769*

N

near field *711*, *715*
view cuts *712*
view cuts by frequency *713*
view cuts by step *714*
newFASANT script *252*

O

object properties *93*
observation directions *682*, *857*, *1057*
observation points *683*, *1058*
coupling *692*
curve *686*
cylinder *684*
external file *685*
line *688*
parallelogram *689*
plane *689*
point *690*
sphere *691*
surface *687*
ogive *539*
openRing *542*
outside *71*

P

pan *54*
parabola *543*
paraboloid *544*
parameter
define *207*
passive antenna *1056*
pattern file *674*
pattern file antenna *674*
pattern file array *676*
pattern files
import *856*
perpendicular *545*

pipe *548*
planarSurface *550*
plane *552*
plot viewer *227*, *236*
point *98*, *554*
pointCloud *555*
pointsFromCurve *557*
pointsFromCurveByDistance *558*
pointsFromSurface *559*
polygon *561*
post-process *275*
power angular spectrum *728*
power process *720*
primitive *97*
project *563*
  by direction *103*, *126*
  new *47*
  open *49*
  recent *50*
  save *51*
  save as *51*
  to closest point *102*, *125*
  wizard *47*
project folder *52*
project parameters
  edit *36*
project to surface *102*, *125*

Q

queueing *29*

R

radiation pattern *1086*
RCS *851*, *869*
RCS parameters *851*
rebuildCurve *566*
rebuildSurface *567*
rectangle *570*
rectangularHole *572*
redo *62*
reference plane *54*
reflectarray *239*
remote server *24*
rename *573*
render mode *60*
result *283*

revolve *574*
rhomboid *577*
ring *579*
rotate *53*, *86*, *580*

S

save *255*
scale *82*, *582*
    1D *82*
    2D *83*
    3D *84*
    non-uniform *85*
scale1D *584*
scale2D *587*
scale3D *590*
scaleNonUniform *593*
script *596*
    load *254*
    save *255*
segment *597*
semicircle *598*
server
    remote *24*
set *599*
set step *208*
simulation parameter *664*, *951*, *1042*
single point *98*
skinned *601*
solid
    box *178*
    cone *179*
    cylinder *180*
    ellipsoid *181*
    extrude surface *175*
    flare *184*
    from surfaces *187*
    ogive *182*
    sphere *185*
    torus *186*
source *852*
    pattern file *854*
sphere *603*
spiral *605*
spiralCross *606*
split *72*, *122*
    by curves *122*

by cutting objects *124*
by direction *173*
by isoparam *171*
cut object *170*
extrude normal *177*
parametric *123*
surface *169*
to closest point *172*
splitByProjection *607*
splitCurves *610*
splitRing *612*
splitSurfaceByIsoparam *613*
splitSurfaceByIsoparamGrid *615*
splitSurfaces *616*
statusbar *45*
stripCross *618*
stripHexagon *619*
stripTrifilar *620*
surface *127*, *621*
bent dipole *162*
capacitance *156*
circular hole *159*
connect *140*
control points *129*
coons *164*
corner points *128*
cross *148*
cross hole *161*
disk *142*
ellipse *144*
extend hole *138*
extrude curve *132*
extrude normal *134*
function *131*
hexagon *143*
hyperboloid *159*
interdigital line *155*
interpolate points *130*
open ring *146*
paraboloid *158*
pipe *168*
planar surface *165*
plane *141*
rebuild *140*
rectangular hole *160*
revolve *135*
ring *146*

skinning *166*
spiral *157*
spiral cross *149*
split ring *147*
strip cross *154*
strip hexagon *152*
strip trifilar *153*
sweep *167*
T cross *150*
three dipoles *163*
three dipoles union *164*
trifilar *151*
trim *136*
untrim *137*
surfaceCornerPoints *623*
surfaceEdges *624*
surfaceEllipse *626*
surfaceFromEdges *627*
surfaceFunction *630*
surfaceInterpolation *632*
surfaceIsoparamCurve *634*
surfaceSpiral *636*
sweep *637*
symmetric *638*
symmetry *86*

T

tangent *641*
tCross *642*
threeDipoles *643*
threeDipolesUnion *644*
tool
angle *188*
area *189*
border *189*
clean surface *205*
curvature *191*
distance *188*
duplicate elements *192*
edges *190*
length *189*
overlapped objects *197*
split surface *200*
toolbar *43*
torus *645*
trace *279*

trace options *279*
trace representation *280*
transform
    to reference plane *88*
tree *33*
trifilar *647*
trim *648*

U

undo *62*
union *68*
unset *651*
untrim *652*
user function *230*
    editor *230*
utility
    batch *23*
    queueing *23*
    remote server *23*

V

view
    reset *53*
view control points *89*
view mesh *864*
view ray *704*
view text files *705*
visibility matrix *701*

X

XOR *70*

Z

zoom in *53*
zoom out *53*
zoom selected *53*