ALTAIR

ONLY FORWARD

Altair Feko 2025.1

Getting Started Guide

Updated: 05/22/2025

Intellectual Property Rights Notice

Altair® Inspire™ Print3D ©2021-2025

Altair® Inspire™ Render ©1993-2025

Altair® Inspire™ Studio ©1993-2025

Altair® Material Data Center™ ©2019-2025

Altair® Material Modeler™ ©2019-2025

Altair® Model Mesher Director™ ©2010-2025

Altair® MotionSolve® ©2002-2025

Altair® MotionView® ©1993-2025

Altair® Multi-Disciplinary Optimization Director™ ©2012-2025

Altair® Multiscale Designer® ©2011-2025

Altair® newFASANT™ ©2010-2020

Altair® nanoFluidX® ©2013-2025

Altair® NLView™ ©2018-2025

Altair® NVH Director™ ©2010-2025

Altair® NVH Full Vehicle™ ©2022-2025

Altair® NVH Standard™ ©2022-2025

Altair® OmniV™ ©2015-2025

Altair® OptiStruct® ©1996-2025

Altair® PhysicsAI™ ©2021-2025

Altair® PollEx™ ©2003-2025

Altair® PollEx™ for ECAD ©2003-2025

Altair® PSIM™ ©1994-2025

Altair® Pulse™ ©2020-2025

Altair® Radioss® ©1986-2025

Altair® romAI™ ©2022-2025

Altair® RTLvision PRO™ ©2002-2025

Altair® S-CALC™ ©1995-2025

Altair® S-CONCRETE™ ©1995-2025

Altair® S-FRAME® ©1995-2025

Altair® S-FOUNDATION™ ©1995-2025

Altair® S-LINE™ ©1995-2025

Altair® S-PAD™ ©1995-2025

Altair® S-STEEL™ ©1995-2025

Altair® S-TIMBER™ ©1995-2025

Altair® S-VIEW™ ©1995-2025

Altair® SEAM® ©1985-2025

Altair® shapeAI™ ©2021-2025

Altair® signalAI™ ©2020-2025

Altair® Silicon Debug Tools™ ©2018-2025

Altair® SimLab® ©2004-2025

Altair® SimLab® ST ©2019-2025

Altair® SimSolid® ©2015-2025

Altair® SpiceVision PRO™ ©2002-2025

Altair® Squeak and Rattle Director™ ©2012-2025

Altair® StarVision PRO™ ©2002-2025

Altair® Structural Office™ ©2022-2025

Altair® Sulis™ ©2018-2025

Altair®Twin Activate® ©1989-2025

Altair® UDE™ ©2015-2025

Altair® ultraFluidX® ©2010-2025

Altair® Virtual Gauge Director™ ©2012-2025

Altair® Virtual Wind Tunnel™ ©2012-2025

Altair® Weight Analytics™ ©2013-2022

Altair® Weld Certification Director™ ©2014-2025

Altair® WinProp™ ©2000-2025

Altair® WRAP™ ©1998-2025

Altair® HPCWorks®, a HPC & Cloud Platform

Altair® Allocator™ ©1995-2025

Altair® Access™ ©2008-2025

Altair® Accelerator™ ©1995-2025

Altair® Accelerator™ Plus ©1995-2025

Altair® Breeze™ ©2022-2025

Altair® Cassini™ ©2015-2025

Altair® Control™ ©2008-2025

Altair® Desktop Software Usage Analytics™ (DSUA) ©2022-2025

Altair® FlowTracer™ ©1995-2025

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

Creating a Rectangular Horn 1

The example is intended for users with no or little experience with CADFEKO. It makes use of a completed rectangular horn model to familiarise yourself with model creation in CADFEKO and viewing the simulated results in POSTFEKO.

This chapter covers the following:

1.1 Example Overview

This example shows a completed rectangular horn model to familiarise yourself with the Feko components and workflow. The main elements and terminology in the CADFEKO and POSTFEKO graphical user interface are discussed.

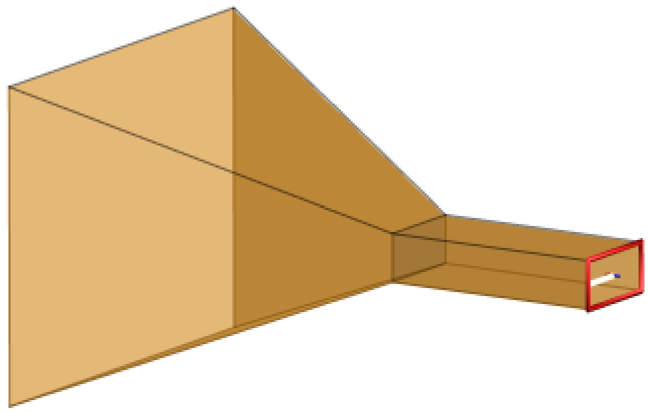


Figure 1: Illustration of the horn antenna.

Tip: Watch the demo video before working through this example. The model in the demo video is similar to the horn model used in this example.

Find the short demo video in the Altair installation directory, for example:

`Altair/2025.1/help/feko/videos/DemoExample.mp4` for Windows and

`Altair/2025.1/help/feko/videos/DemoExample.html` for Linux.

1.2 Topics Discussed in this Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- View the Feko general workflow.
- Launch CADFEKO.
- View the CADFEKO layout.
- View the POSTFEKO layout.
- View the far field results and near field results in POSTFEKO.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[3]:
>
> `GS 1: Rectangular Horn Antenna`

3. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

1.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

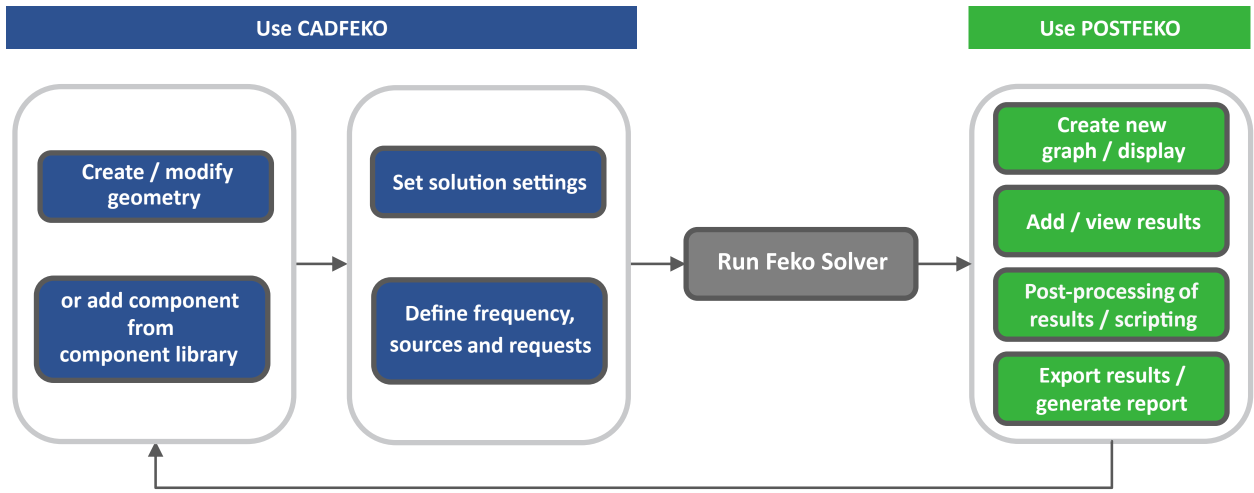
- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 30 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[4] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

4. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

1.4 Feko Components and Workflow

View the typical workflow when working with the Feko components.



CADFEKO

Create or modify the geometry (or model mesh) in CADFEKO, import geometry or mesh, or use a component from the component library. Apply solution settings, define the frequency, specify the required sources and request calculations.

When the frequency is specified or local mesh settings are applied, the automatic mesh algorithm calculates and creates the mesh to obtain a discretised representation of the geometry or model mesh. View the status of the model in the Notification centre[5]. If any warnings or errors are given, correct the model before running the Solver.

Solver

Run the Solver to calculate the specified output requests.

POSTFEKO

Create a new graph or 3D view and add results of the requested calculations on a graph or 3D view. Results from graphs can be exported to data files or images for reporting or external post-processing. Reports can be created that export all the images to a single document or a custom report can be created by configuring a report template.

After viewing the results, it is often required to modify the model again in CADFEKO and then repeat the process until the design is complete.

5. Notification centre is the panel to the right of the 3D view under **Model Status**.

1.5 Introduction to CADFEKO

Use CADFEKO to configure a solver-ready input file for Solver simulations.

CADFEKO is the Feko component that allows you to create complex CAD geometry using primitive structures (for example, cuboids and polygons) and to perform Boolean operations (for example, union and subtract) on the geometry. Complex geometry models and mesh models can be imported or exported in a wide range of industry standard formats. Reduce development time by using a component from the list of antennas and platforms in the component library.

In CADFEKO, you can request multiple solution configurations, specify calculation requests as well as specify the solution settings for the model. If an optimisation search is required, you can specify the optimisation parameters and goals.

1.5.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

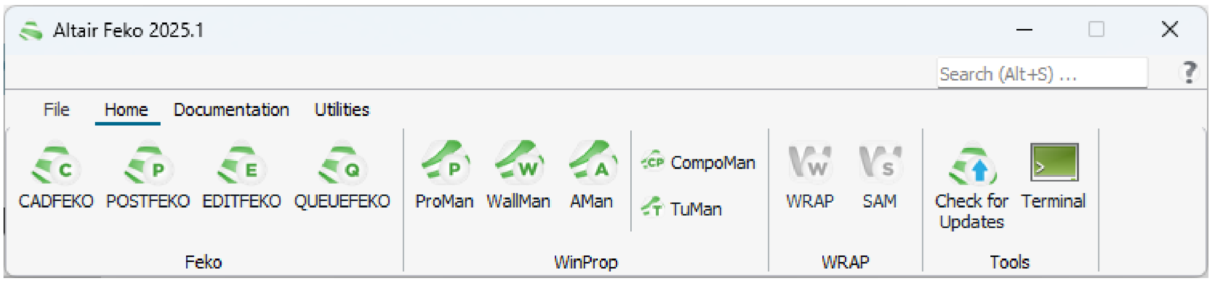


Figure 2: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[6] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

1.5.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the “initfeko” script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

6. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

1.5.3 Start Page

The Feko start page is displayed when starting a new instance (no models are loaded) of CADFEKO, EDITFEKO or POSTFEKO.

The start page provides quick access to **Create a New Project**, **Open an Existing Project**, and a list of **Recent models**.

Links to the documentation (in PDF format), introduction videos and website resources are available on the start page. Click the ? icon to launch the Feko help.

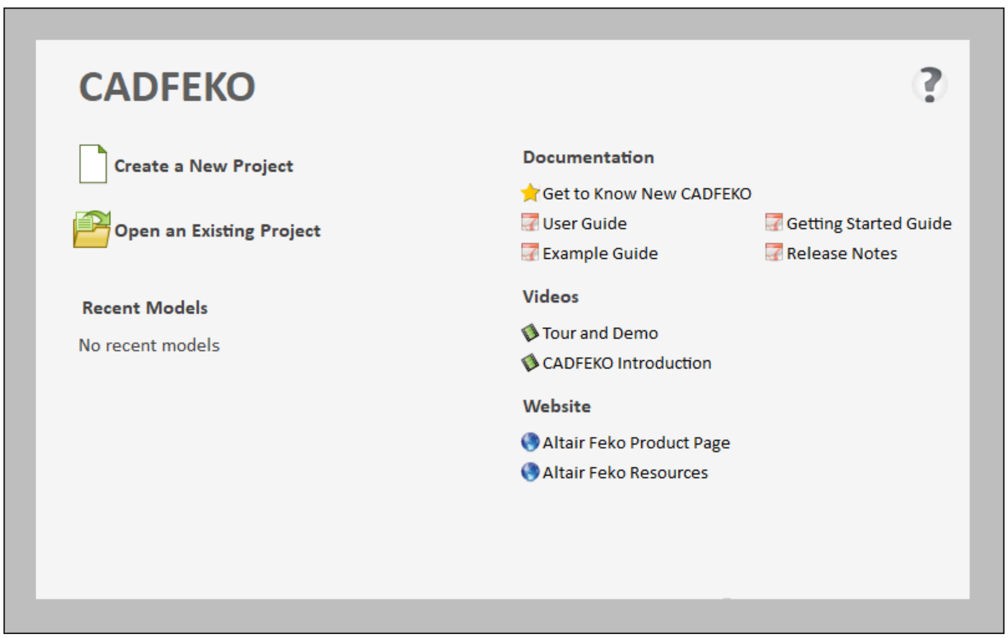
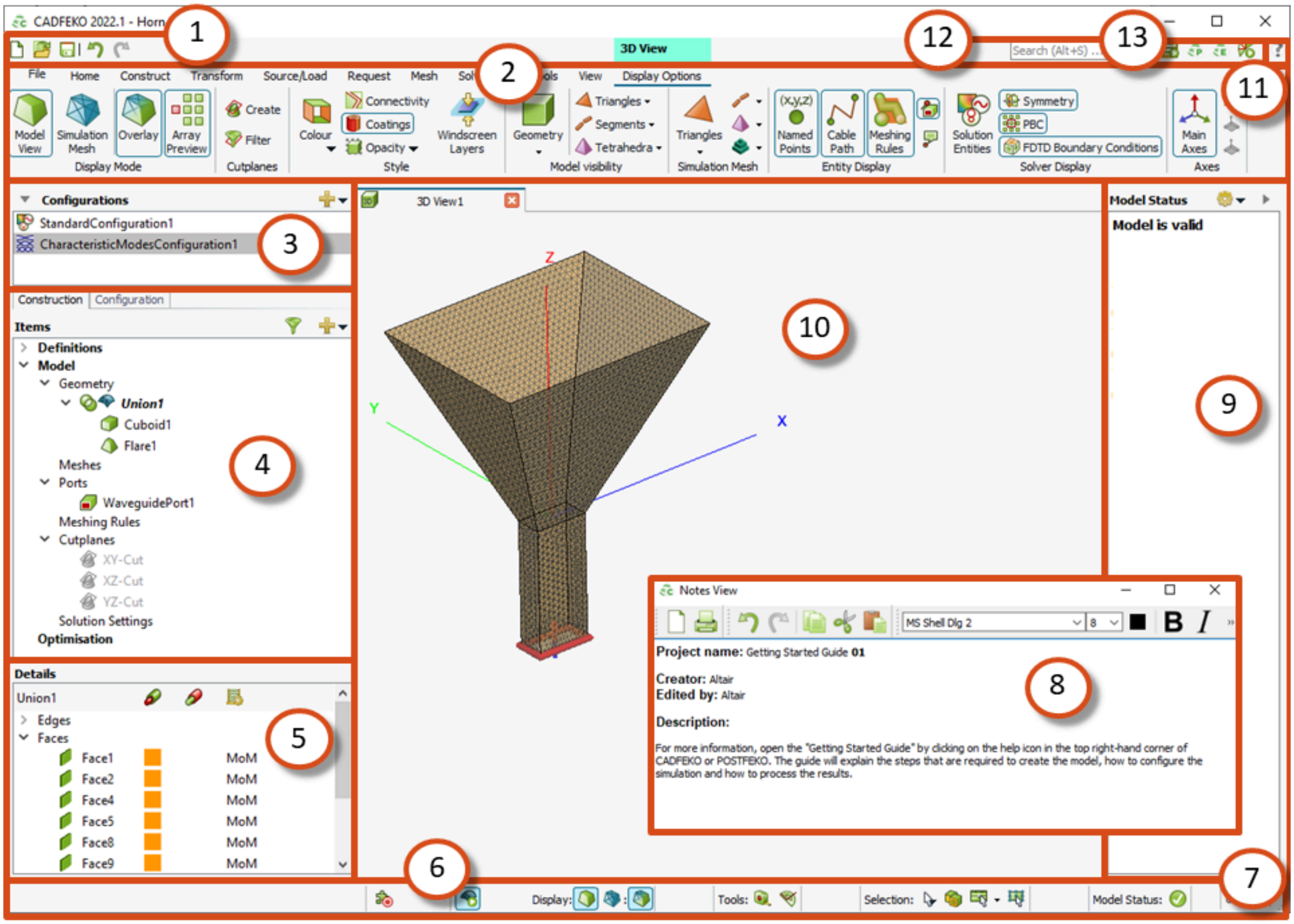


Figure 3: The CADFEKO start page.

1.5.4 User Interface Layout

View the main elements and terminology in the CADFEKO graphical user interface (GUI).



1. *Quick access toolbar*

   The quick access toolbar is a small toolbar that gives quick access to actions that are performed often. The actions available on the quick access toolbar are also available via the ribbon. The quick access toolbar includes: **New**, **Open**, **Save**, **Undo** and **Redo**.

2. *Ribbon*

   The ribbon is a command bar that groups similar actions in a series of tabs. The ribbon consists of the application menu, core tabs and contextual tab sets.

3. *Configuration list*

   The configuration list is a panel that displays all defined configurations in the model. A new model starts by default with a single standard configuration. The following configuration types are supported: Standard configuration, Multiport S-parameter configuration and Characteristic modes configuration.

> **Tip:** Multiple configurations allow you to perform efficient simulations using different configurations (different loads, sources, frequencies or power scaling) in a single model.

4. *Model tree*

   The model tree is a panel that organises the model-creation hierarchy and configuration-specific items of the model into two separate tabs at the top of the panel. A right-click context menu is available for all items in the model tree. Double-click on an item to open its properties.

   Predefined variables, named points, media, workplanes, field/current data, worksurfaces and cables are listed in both the **Construction** tab and the **Configuration** tab to provide quick access.

   a. *Construction tab*
      The **Construction** tab lists the model-creation hierarchy in a tree format.

      > **Note:** Select a geometry or mesh part on the **Construction** tab and in the details tree (5), modify its wire / edge / face / region properties, solution settings and custom mesh settings.

   b. *Configuration tab*
      The **Configuration** tab lists the configuration-specific items in a tree format.

      > **Note:** Select a configuration in the configuration list (3) and view its configuration-specific items in the **Configuration** tab.

5. *Details tree*
   The details tree is a panel that displays the relevant wires, edges, faces and regions for the geometry or mesh part selected in the **Construction** tab (4). From the right-click context menu specify the properties for its wire, edge, face or region properties in the details tree. You can modify the selected item's local mesh size, material definition or coating or solution properties that are specific to the selection.

6. *Status bar*
   The status bar is a small toolbar that gives quick access to macro recording, general display settings, tools, selection method and type, snap settings and the model unit.

7. *Model Status icon*
   The Model Status icon shows the current status of the model in the Notification centre. The Notification centre can be hidden but the Model Status icon in the status bar will still indicate the current status of the model.

8. *Notes view*

The notes view is a window where you can document model details. Add additional comments or information for future reference.

> **Tip:** The notes view is hidden by default, but can be enabled.
>
> On the **Home** tab, in the **Create view** group, click the **Notes** icon.

9. *Notification centre*

The Notification centre performs computational electromagnetic model (CEM) validation and shows the status of the model. When problems in the model are detected, it is highlighted in the Notification centre with hyperlinks to the problematic entities.

10. *3D view*

The 3D view window displays the geometry and mesh as well as solution requests (for example, a far field request).

> **Tip:**
>
> - Select the **Construction** tab (4.a) to view only CAD in the 3D view.
> - Select the **Configuration** tab (4.b) to view both CAD and solution requests in the 3D view.

11. *Help*

The **Help** icon gives quick access to the Feko manuals.

> **Tip:** Press F1 to access context-sensitive help.

12. *Search bar*

The search bar is a single-line textbox that allows you to enter a keyword and search for relevant information in the GUI. Entering a keyword in the search bar will populate a drop-down list of actions as well as the location of the particular action on the ribbon or context menu. Clicking on an item in the list will execute the action.

13. *Application launcher*

The application launcher toolbar is a small toolbar that gives quick access to other Feko components.

1.5.5 Ribbon

The ribbon is a command bar that groups similar actions in a series of tabs.

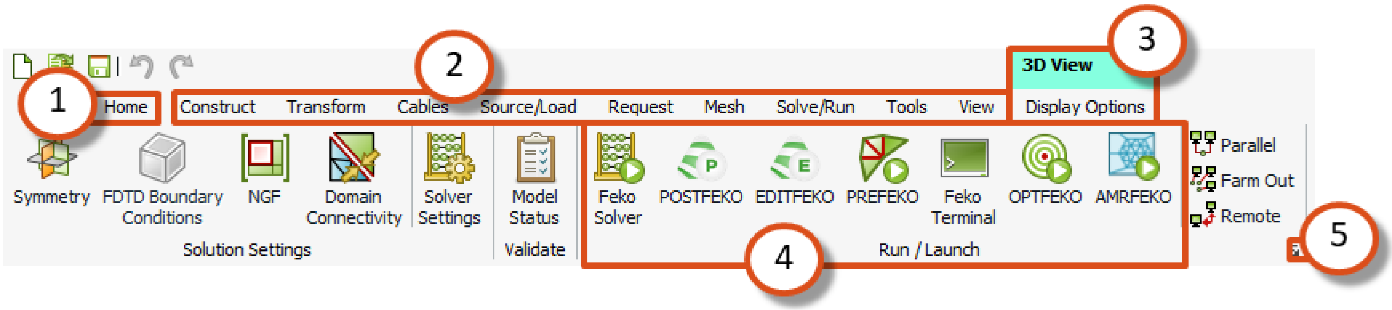


Figure 4: The ribbon in CADFEKO.

1. File menu
: The **File** menu is the first item on the ribbon. The menu allows saving and loading of models, import and export options as well as giving access to application-wide settings and a recent file list.

2. Core tabs
: A tab that is always displayed on the ribbon, for example, the **Home** tab and **Construct** tab.

  The **Home** tab is the first tab on the ribbon and contains the most frequently used commands for quick access.

3. Contextual tab sets
: A tab that is only displayed in a specific context.

  For example, the **Schematic** contextual tab set contains the **Network Schematic** contextual tab. Contextual tabs appear and disappear as the selected items such as a view or item on a view, change.

4. Ribbon group
: A ribbon tab consists of groups that contain similar actions or commands.

5. Dialog launcher
: Click the dialog launcher to launch a dialog with additional and advanced settings that relate to that group. Most groups don't have dialog launcher buttons.

Keytips
: A keytip is the keyboard shortcut for a button or tab that allows navigating the ribbon using a keyboard (without using a mouse). Press F10 to display the keytips. Type the indicated keytip to open the tab or perform the selected action.

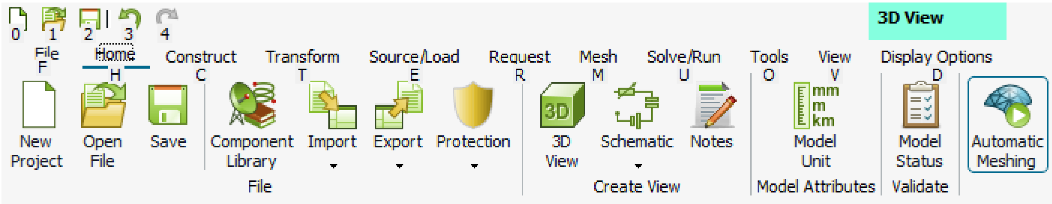


Figure 5: An example of keytips.

1.5.6 Construction Tab

The Construction tab contains the geometry and mesh representation of the current model in a tree structure. It also lists ports and the optimisation configuration.

The tree contains a **Definitions** branch, **Model** branch and **Optimisation** branch.

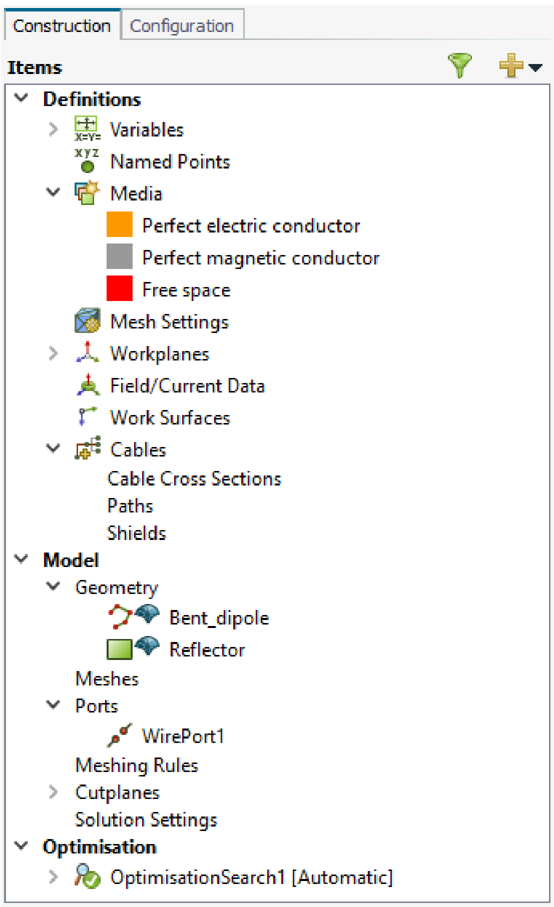


*Figure 6: The **Construction** tab in the model tree.*

Definitions Branch

The **Definitions** branch contains by default the predefined variables, named points, media, mesh settings, workplanes, field/current data, worksurfaces and cables.

Model Branch

The **Model** branch is mainly a visualisation of the geometry and mesh creation hierarchy. Where geometry or mesh objects are derived from existing ones, the original (parent) objects are removed from the top level of the model and listed as sub-levels (children) under the new object.

> **Note:** The highest-level items in the model are referred to as "parts".

For example, **Cone1** and **Cuboid1** (parent objects) were unioned and the result is that they have become children of the new object **Union1**. **Union1** is the highest-level item and referred to as a part.

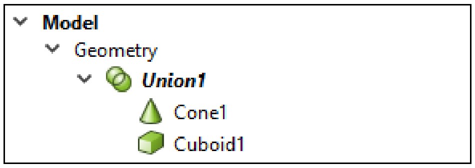


*Figure 7: The **Construction** tab in the model tree showing the part, **Union1**.*

The **Model** branch also contain the ports, meshing rules, cutplanes and solution settings.

Optimisation Branch

The **Optimisation** branch contains the optimisation searches, associated masks, parameters and goal functions defined for the model.

> **Note:** The **Optimisation** branch is only displayed if the model contains an optimisation search or mask.

1.5.7 Configuration Tab

The Configuration tab contains the global and configuration-specific model settings and requests of the current model in tree form.

The tree contains a **Definitions** branch, **Global** branch and **Configuration specific** branch.

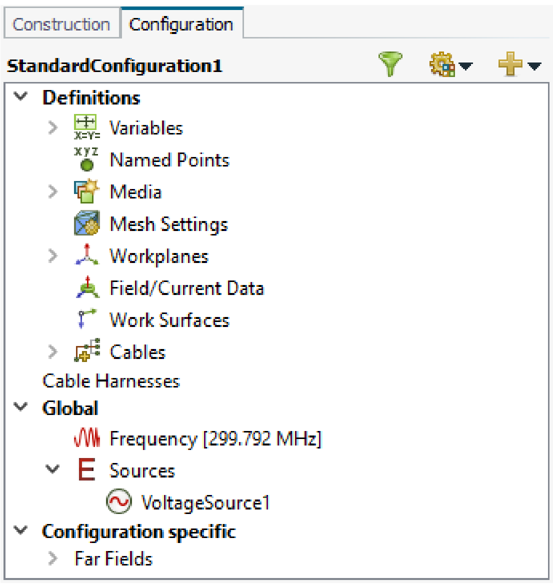


Figure 8: The ***Configuration*** *tab in the model tree.*

Definitions Branch

The **Definitions** branch contains by default the predefined variables, named points, media, mesh settings, workplanes, field/current data, work surfaces and cables.

Global Branch

The **Global** branch contains the global specific model settings. From the right-click context menu define solver settings, specify the global frequency, sources, loads, networks and power.

Configuration specific Branch

The **Configuration specific** branch contains configuration specific settings. From the right-click context menu define requests per configuration, frequency per configuration, sources per configuration, loads per configuration and power per configuration.

1.5.8 Notification Centre

The Notification centre performs computational electromagnetic model (CEM) validation and shows the status of the model and notifications.

The Notification centre lets you stay informed of the model status at all times. When problems in the model are detected, it is highlighted in the Notification centre with hyperlinks to the problematic entities.

The Notification centre can be hidden but the **Model Status** icon in the status bar will still indicate the current status of the model.

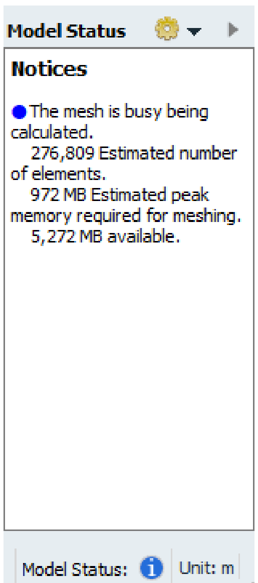


Figure 9: The Notification centre in CADFEKO. Note the Model Status icon at the bottom that shows the current status of the model.

Show or hide the Notification centre using one of the following workflows:

- Click the **Model Status** icon in the status bar.
- Drag the splitter from the right edge of the application to open the pane. To close, drag the splitter all the way to the right.
- On the **Home**, in the Validate group, click the **Model Status** icon.
- Click the icon in the Notification centre to hide the panel. Click the icon to show the panel again.

1.5.9 Dialog Error Feedback

CADFEKO provides error feedback for dialogs by showing a soft message bubble when validation fails on a dialog.

Click the icon to show or hide the message bubble or click elsewhere in CADFEKO to hide the message bubble. The error feedback is also shown per tab when the validation fails on a multi-tab dialog.

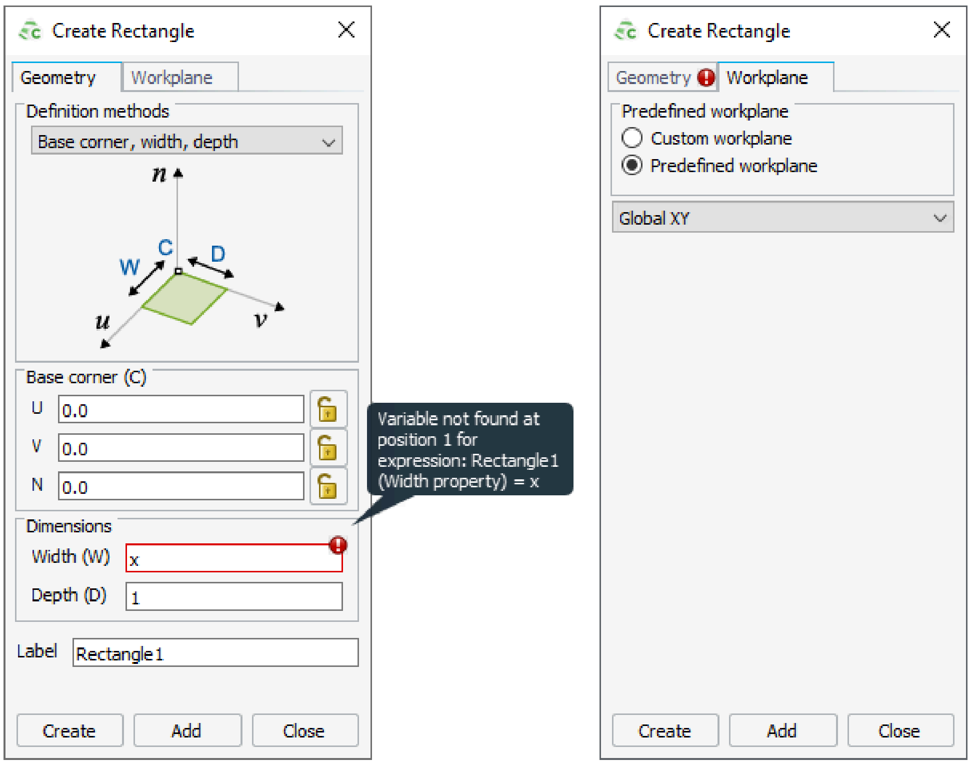


*Figure 10: The soft message bubble indicating that an undefined variable was used on the **Geometry** tab of the **Create Rectangle** dialog.*

1.5.10 Custom Keyboard Shortcut Settings

CADFEKO provides default keyboard shortcuts. To better fit your workflow and work style, you can reassign keyboard shortcuts to different commands.

To reassign keyboard shortcuts, click **File** > **Settings** > **Keyboard Shortcut Settings**.

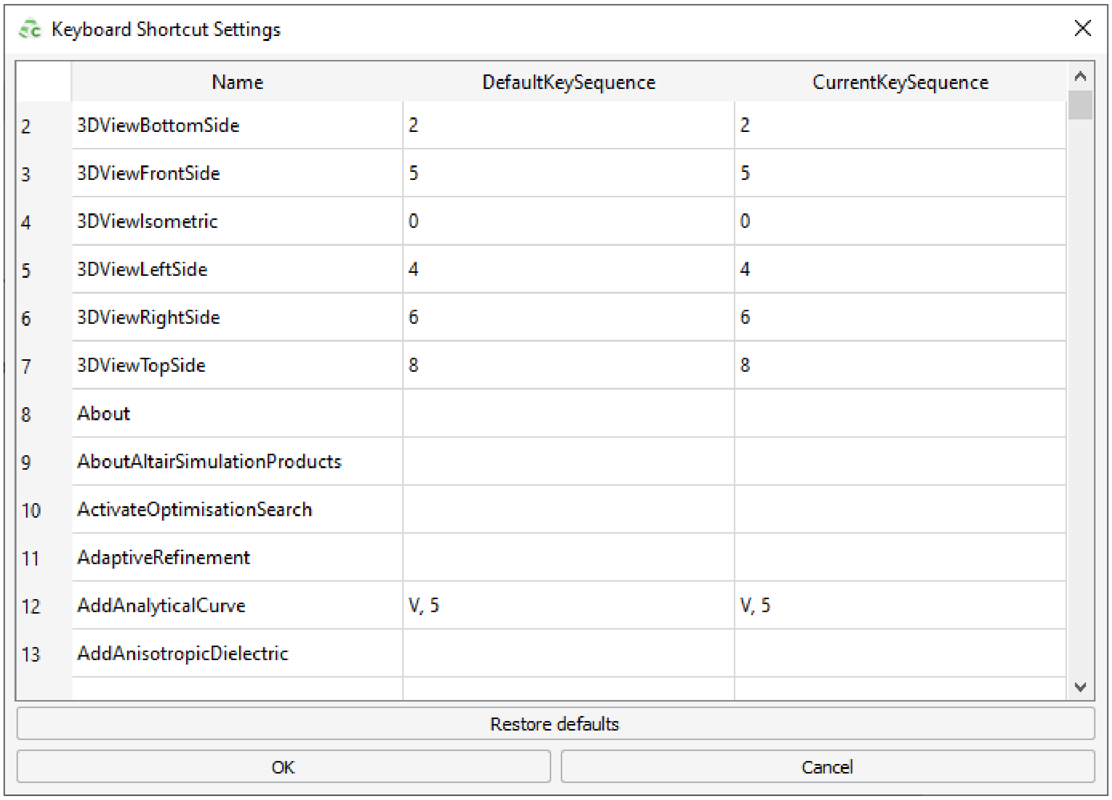


*Figure 11: The **Keyboard Shortcut Settings** dialog.*

For example, to change the shortcut key for the undo command, on the **Keyboard Shortcut Settings** dialog, click in the **CurrentKeySequence** column and enter the shortcut key that suits your work style.

1.5.11 Custom Mouse Bindings

CADFEKO provides default commands for all the mouse buttons. To better fit your workflow and work style, you can reassign mouse buttons to different commands.

To reassign mouse buttons, click **File** > **Settings** > **Mouse Binding Settings**.

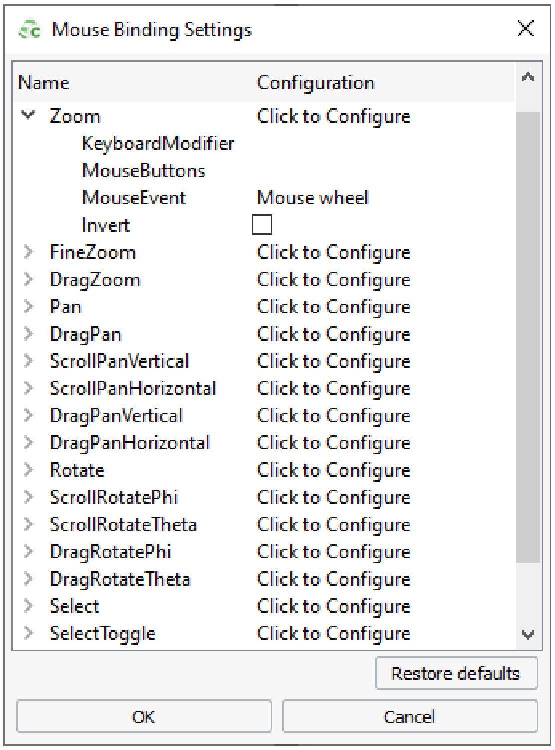


*Figure 12: The **Mouse Binding Settings** dialog.*

For example, to reverse the mouse wheel direction to better suit your workflow, on the **Mouse Bindings** dialog, click **Click to Configure**. On the **Zoom** dialog, select the **Invert** check box.

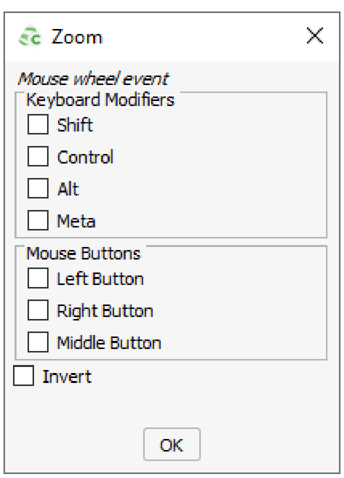


*Figure 13: The **Zoom** dialog.*

1.6 Introduction to POSTFEKO

Use POSTFEKO to validate meshed geometry and analyse and post-process results.

POSTFEKO is the component that allows you to verify that your model is constructed and configured correctly before starting a simulation and analyse the results after the simulation completes. The POSTFEKO component is particularly useful to verify models created using EDITFEKO, but it is just as relevant for CADFEKO model verification.

Result post-processing and analysis is the primary function of POSTFEKO. Once a model has been simulated, POSTFEKO can be used to display and review the results. It is easy to load multiple models in a single session and compare them on 3D views, Cartesian graphs, Smith charts, polar graphs and surface graphs. Various measurement and other data formats are supported for comparison to the simulated results. A powerful scripting interface makes it easy to post-process results, automate repetitive tasks and create plug-in extensions that customise the interface and experience.

1.6.1 Reviewing POSTFEKO and Launching OPTFEKO

Open POSTFEKO from within CADFEKO.

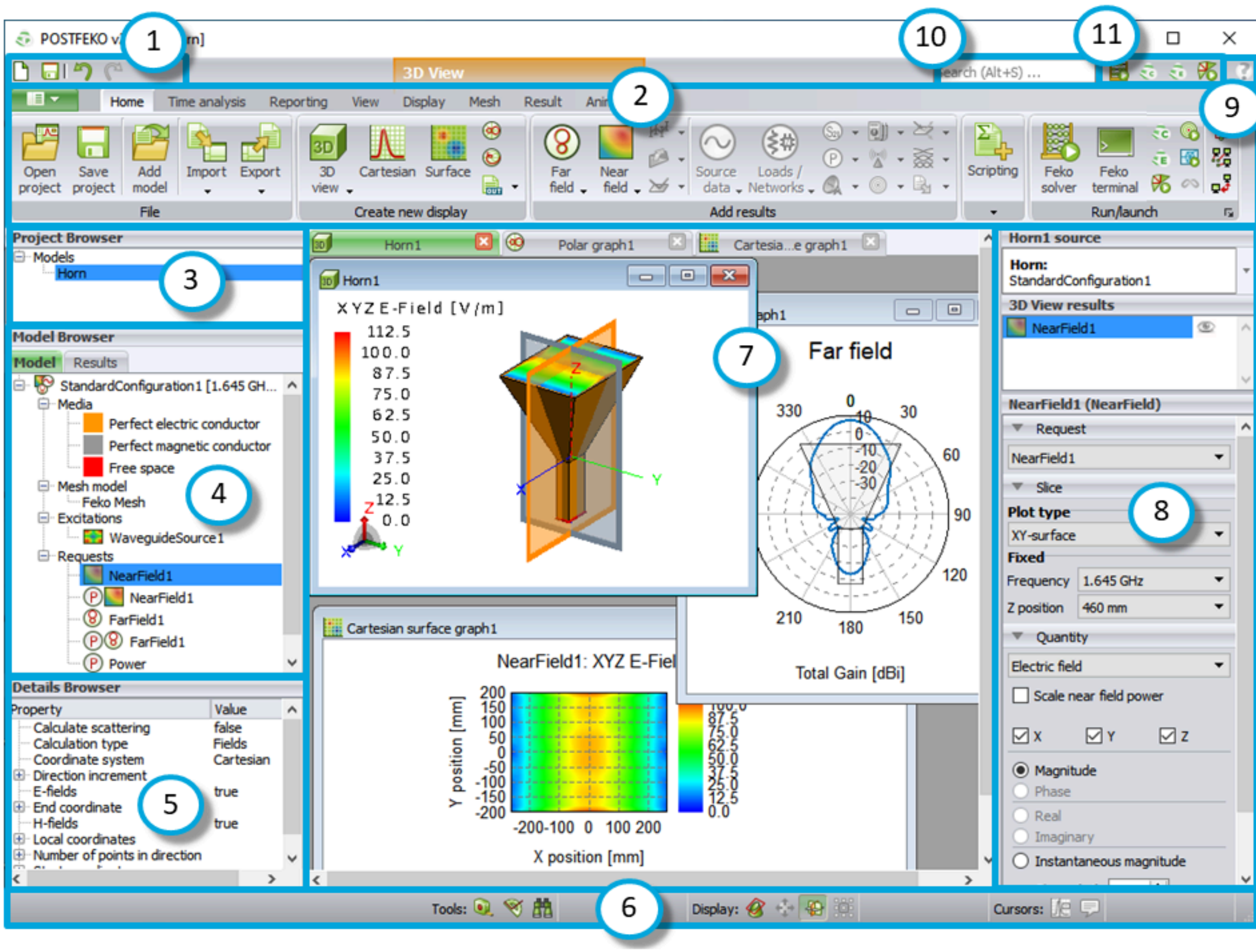
Use one of the following workflows to launch POSTFEKO:

- On the **Solve/Run** tab, in the **Run/Launch** group, click the **POSTFEKO** icon.
- On the application launcher toolbar, click the **POSTFEKO** icon in the group.
- Press Alt+3 to use the keyboard shortcut.

POSTFEKO opens by default with a single 3D view containing the model geometry.

1.6.2 User Interface Layout

View the main elements and terminology in the POSTFEKO graphical user interface (GUI).



1. *Quick access toolbar*

   The quick access toolbar is a small toolbar that gives quick access to actions that are performed often. The actions available on the quick access toolbar are also available via the ribbon. The quick access toolbar includes: **New project**, **Open project**, **Save project**, **Undo** and **Redo**.

2. *Ribbon*

   The ribbon is a command bar that groups similar actions in a series of tabs. The ribbon consists of the application menu, core tabs and contextual tab sets.

3. *Project browser*

   The project browser is a panel that lists the models loaded in the current project, imported data, stored data and scripted data.

> **Tip:** Collapse the project browser to expand the 3D view.
>
> On the **View** tab, in the **Show** group, click the **Project** icon.

4. *Model browser*

   The model browser is a panel that organises the model information of the selected model in the project browser (3), into two separate tabs.

   Model tab

   The **Model** tab lists the model information and results for the selected model.

   Results tab

   The **Results** tab lists the results and solution information.

5. *Details browser*

   The details browser is a panel that shows in-depth detail for the selected item in the model browser (4).

   > **Tip:** View the solution information for the selected model.
   >
   > On the model browser (**Results** tab), click **Solution information** to view:
   > - start and end time
   > - memory per process
   > - total CPU-time

6. *Status bar*

   The status bar is a small toolbar that gives quick access to general display settings, tools, and graph cursor settings.

7. *3D view/2D graphs*

   3D view

   The 3D view displays the geometry, mesh, solution settings as well as 3D results.

   2D graphs

   The 2D graphs display the 2D results on either a Cartesian graph, polar graph, Smith chart or Cartesian surface graph.

   > **Tip:**
   > - Re-order the window tabs by simply dragging the tab to the desired location.
   > - Rename the window tab by using the right-click context menu and selecting **Rename**.

8. *Result palette*

   The result palette is a panel that gives access to options that control the data in the 3D view or 2D graph for the relevant result type. For example, 3D far field data allows the phi cut plot type and gain in dB to be specified.

9. *Help*

   The **Help** icon gives quick access to the Feko manuals.

   > **Tip:** Press F1 to access context-sensitive help.

10. *Search bar*

    The search bar is a single-line textbox that allows you to enter a keyword and search for relevant information in the GUI. Entering a keyword in the search bar will populate a drop-down list of actions as well as the location of the particular action on the ribbon or context menu. Clicking on an item in the list will execute the action.

11. *Application launcher*

    The application launcher toolbar is a small toolbar that gives quick access to other Feko components.

1.6.3 Validating the Model in POSTFEKO

View the model using visualisation tools in POSTFEKO to confirm the model was created as intended.

Confirm the horn model is open in the 3D view.

1. Enable the mesh edges of the model.

a) On the **3D View** contextual tabs set, on the **Mesh** tab, in the **Visibility** group, click the **Metal** icon. From the drop-down list, select the **Edges** check box.

2. Zoom to extents of the 3D view using one of the following workflows:

- On the **View** tab, in the **Zoom** group, click the **Zoom to extents** icon.
- Press F5 to use the keyboard shortcut.

3. Enable tick marks on the axes.

a) On the **3D View** contextual tabs set, on the **Display** tab, in the **Axes** group, click the **Tick Marks** icon.

4. Use the distance measurement tool to validate the dimensions of the horn.

a) On the **3D View** contextual tabs set, on the **Mesh** tab, in the **Tools** group, click the **Measure Distance** icon.

b) On the **Measure Distance** dialog, ensure that the **Point1** field is active.

c) In the 3D view, press Ctrl+Shift+left click on the first point.

d) Repeat Step 4.c for the second point.

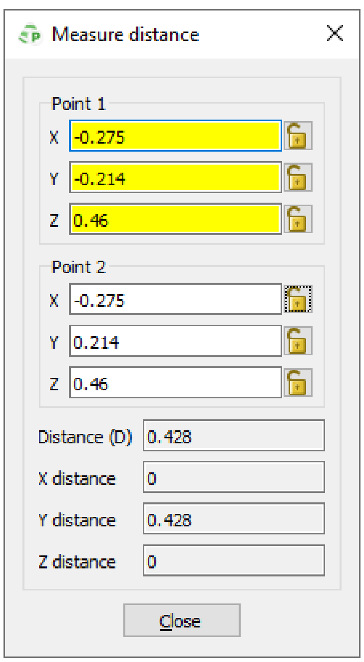


*Figure 14: The **Measure distance** dialog showing the distance between two points.*

e) Click **Close** the close the dialog.

1.6.4 Viewing the Near Field Results (3D)

View the near field results in the 3D view and add a legend and contours.

1. Add the near field data to the 3D view.
   a) On the **Home** tab, in the **Add results** group, click the **Near Fields** icon. From the drop-down list, select **NearField1**.

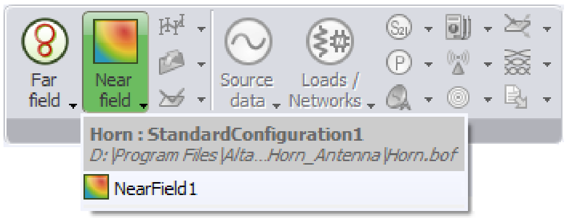


*Figure 15: Section of the ribbon showing the **Add results** group.*

2. View the magnitude of the E_y component of the field.
   a) On the result palette, in the **Quantity**, clear the **X** check box and the **Z** check box.

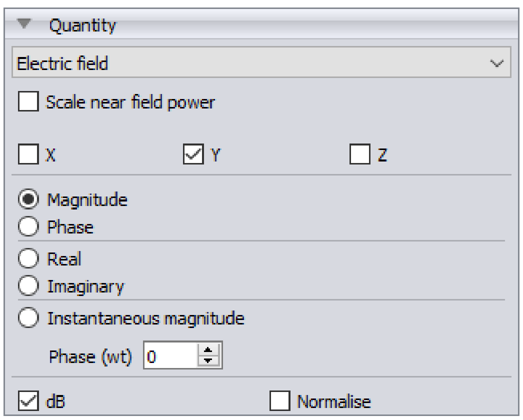


*Figure 16: The **Quantity** panel in the result palette.*

3. View the electric field in dB.
   a) On the result palette, in the **Quantity** panel, select the **dB** check box.
4. Add a legend to the 3D view (top left).
   a) On the **3D View** contextual tabs set, on the **Display** tab, in the **Legends** group, click the **Top left** icon. From the drop-down list select **NearFields**.
5. Add contours to the near field result.
   a) On the **3D View** contextual tabs set, on the **Result** tab, on the **Contours** group, click the **Show contours** icon.
6. Specify the number of contours for the near field.
   a) On the **3D View** contextual tabs set, on the **Result** tab, in the **Contours** group, click the **Position** icon. Click **Number of contours** and set its value to `11`.

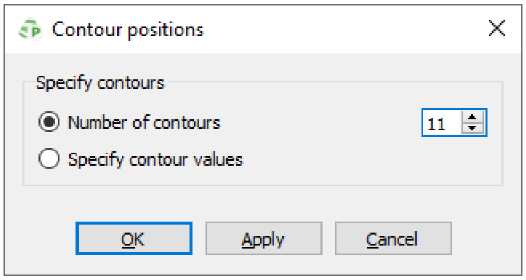


Figure 17: The ***Contour positions*** *dialog.*

b) Click **OK** to close the dialog.

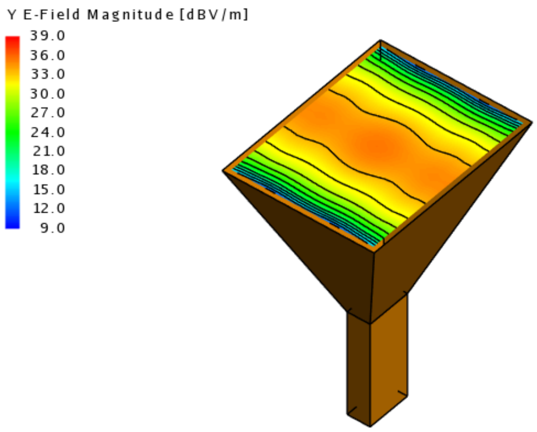


Figure 18: The near field results with contours.

1.6.5 Viewing the Near Field Results (2D)

Create a new Cartesian graph. Create two near field traces and compare the E_y and E_x components of the near field along the X direction.

1. Create a new Cartesian graph.
   a) On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. Add the near field result to the Cartesian graph.
   a) On the **Home** tab, in the **Add results** group, click the **Near Fields** icon. From the drop-down list, select **NearField1**.
3. View the near field along the X direction.
   a) On the result palette, in the **Slice** panel, make the following changes:
      - From the **Independent axis (Horizontal)** list, select **X position**.
      - From the **Frequency** list, select **1.645 GHz**.
      - From the **Y position** list, select **100 mm**.
      - From the **Z position** list, select **460 mm**.

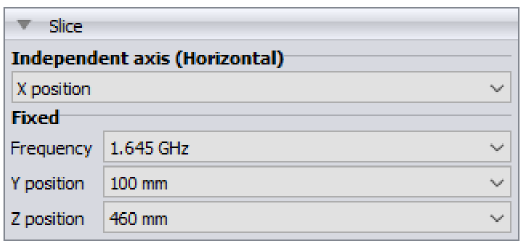


*Figure 19: The **Slice** panel in the result palette.*

4. View the magnitude of the E_y component of the field.
   a) On the result palette, in the **Quantity** panel, clear the **X** check box and the **Z** check box.

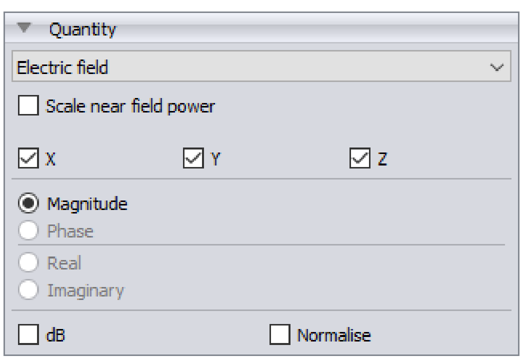


*Figure 20: The **Slice** and part of the **Quantity** panels in the result palette.*

5. Add a second trace to the Cartesian graph by duplicating the **NearField1** trace.
   a) On the **Trace** tab, in the **Manage** group, click the **Duplicate trace** icon.

   A second trace, **NearField1_1**, is created.

6. View the magnitude of the E_x component of the field.

a) On the result palette, select the **NearField1_1** trace.

b) On the result palette, in the **Quantity** panel, select the **X** check box and clear the **Y** check box.

7. Set the vertical axis to dB.

a) In the result palette, select both traces (**NearField1** and **NearField1_1**).

b) In the **Quantity** panel, select the **dB** check box.

8. Modify the minimum and maximum values for the vertical axis.

a) On the **Cartesian** context tab, on the **Display** tab, on the **Axes** group, click the **Axis settings** icon.

b) On the **Axis settings (Cartesian graph)** dialog, select the **Vertical** tab.

c) Clear the **Automatically determine the grid ranges** check box.

d) In the **Maximum value** field, enter a value of `40`.

e) In the **Minimum value** field, enter a value of `-20`.

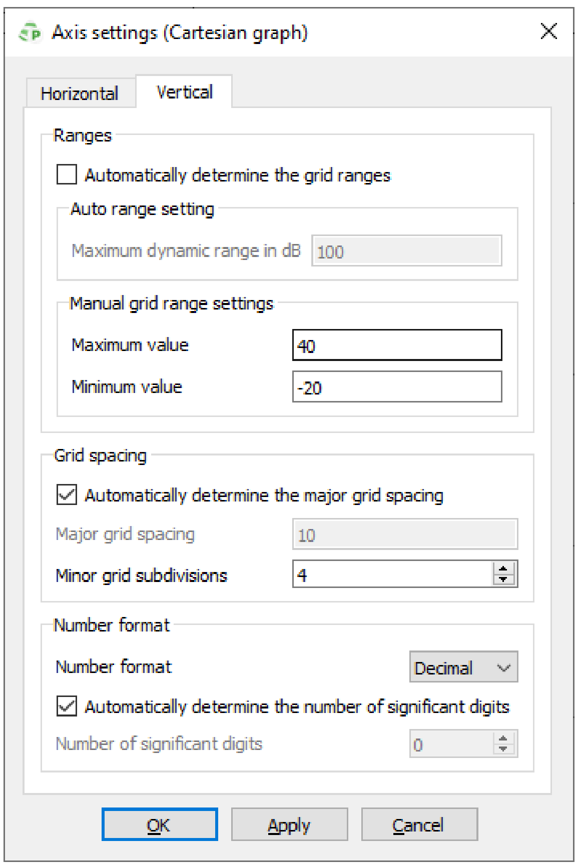


*Figure 21: The **Axis settings (Cartesian graph)** dialog.*

f) Click **OK** to close the dialog.

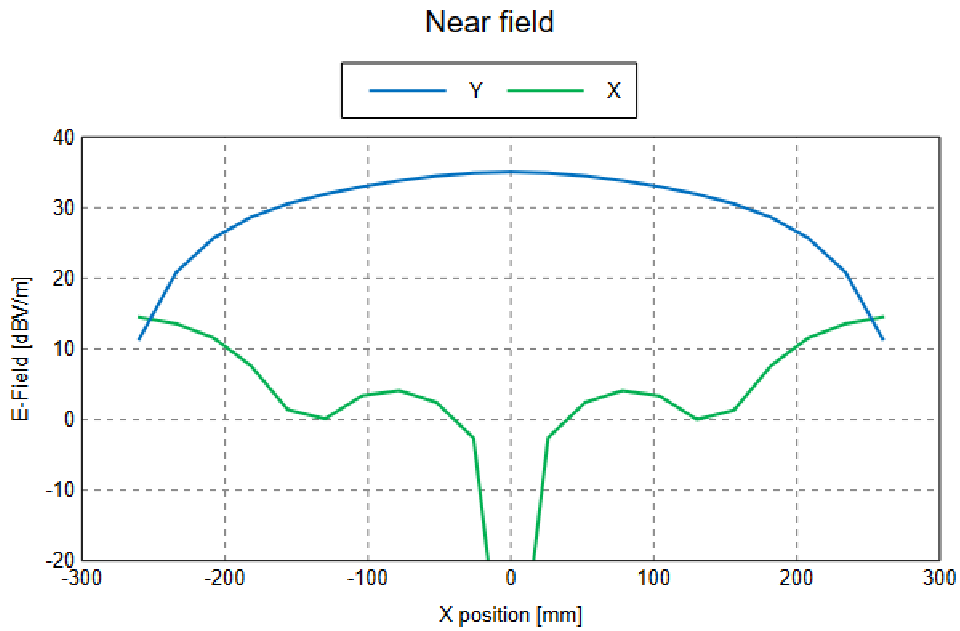


Figure 22: The E_y and E_x components of the near field along the X direction.

1.6.6 Viewing the Far Field Results (3D)

View the far field results in the 3D view.

1. Select the 3D view window.
2. Add the far field result to the 3D view.
   a) On the **Home** tab, in the **Add results** group, click the **Far field** icon. From the drop-down list, select **FarField1**.
3. Hide the near field result still displayed in the 3D view.
   a) In the result palette, on the **Traces** panel, click the "eye" icon next to **NearField1**.

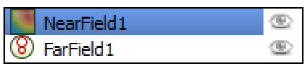


Figure 23: An open "eye" icon indicates that the trace is shown.

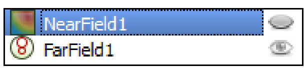


Figure 24: A closed "eye" icon indicates that the trace is hidden.

4. Add an annotation to the far field.
   a) Add an annotation to the desired location by pressing Ctrl+Shift+left click.
5. View the fields in dB.
   a) On the result palette, in the **Quantity** panel, select the **dB** check box.
6. Change the size of the far field compared to the geometry.
   a) On the **3D View** contextual tabs set, on the **Result** tab, in the **Rendering** group, click the **Size** icon. From the drop-down list, select **Custom**.
   b) On the **Specify** dialog, set the size as `70%`.

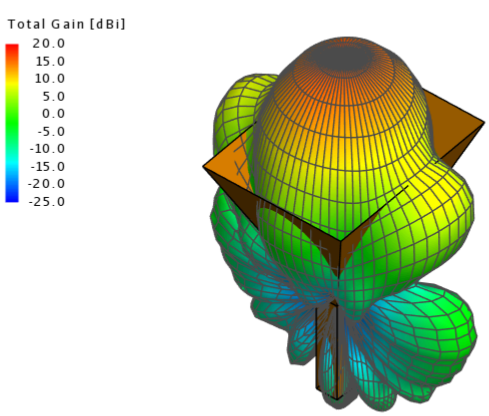


Figure 25: The 3D far field result.

   c) Click **OK** to specify the size of the far field and to close the dialog.

1.6.7 Viewing the Far Field Results (2D)

View the far field results on a polar graph.

> **Note:** Since a full 3D set of data was requested for this example, 2D cuts can be extracted.

1. Create a new polar graph.
   a) On the **Home** tab, in the **Create new display** group, click the **Polar** icon.
2. Add the far field result to the polar graph.
   a) On the **Home** tab, in the **Add results** group, click the **Far field** icon. From the drop-down list, select **FarField1**.
3. View the far field gain plotted in the YZ plane.
   a) On the result palette, in the **Slice** panel, make the following changes:
      - From the **Independent axis (Angular)** drop-down list, select **Theta (wrapped)**.
      - From the **Frequency** drop-down list, select **1.645 GHz**.
      - From the **Phi** drop-down list, select **90 deg (wrapped)**.
4. On the result palette, in quantity panel panel, select the **dB** check box.

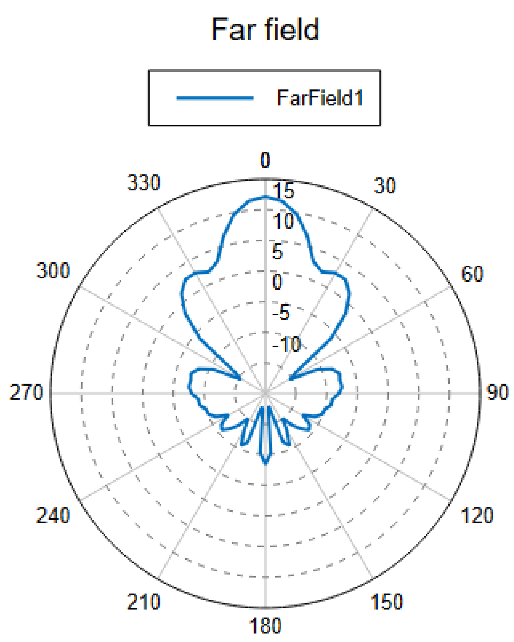


Figure 26: The far field results on a polar graph.

Creating CADFEKO Models 2

The example is intended for users with no or little experience with CADFEKO. This example is not an example intended for simulation, but rather to familiarise yourself with model creation in CADFEKO.

This chapter covers the following:

2.1 Example Overview

Create a simple model using basic geometry and transformations to familiarise yourself with model creation in CADFEKO.

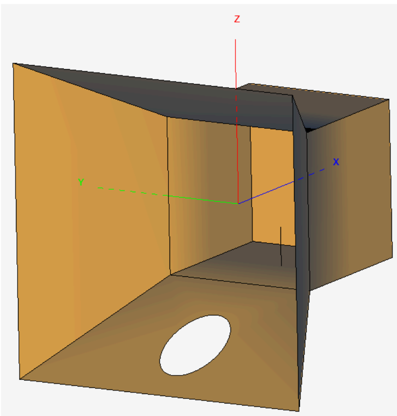


Figure 27: Illustration of the geometry created in this example.

> **Note:**
>
> - The example does not use the fastest or most effective way to create geometry, but instead it highlights a subset of tools available in CADFEKO to create complicated geometrical structures.

> **Note:**
>
> - This example is not intended for running the Solver. No electromagnetic solution is performed and no results are presented.

2.2 Topics Discussed in this Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- CADFEKO
    - Launch CADFEKO.
    - Define variables to create a parametric model.
    - Add a custom workplane.
    - Create a rectangle.
    - Create a line.
    - Create a cuboid by sweeping the rectangle along the line.
    - Create a flare.
    - Union the flare and cuboid to ensure the parts are electrically connected.
    - Remove a redundant face in the flare to create a horn.
    - Add a feed pin to the line.
    - Use automatic selection in the 3D view.
    - Create an ellipse and subtract the shape from the cuboid face to create a hole.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[7]:
>
> `GS 2: Model Construction`

7. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

2.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 40 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[8] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

8. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

2.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

2.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

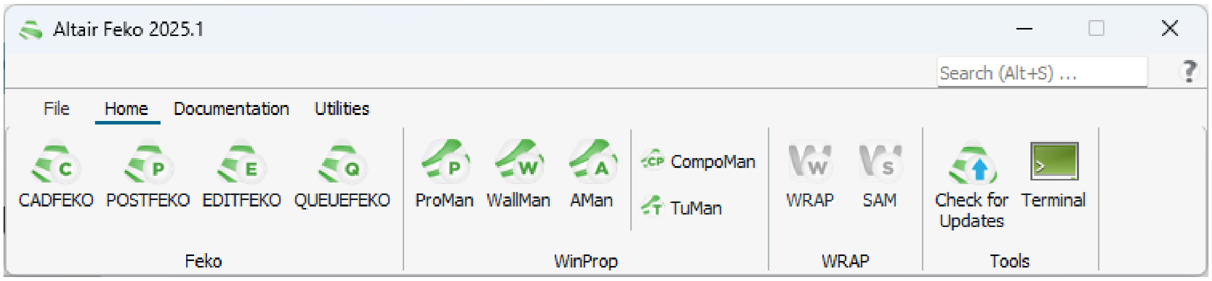


Figure 28: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[9] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

2.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the “initfeko” script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

9. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

2.4.3 Building a Horn

Learn to create variables, workplanes and primitive shapes. Continue by combining these basic entities and modifying their properties.

> **Note:** To demonstrate the path sweep tool, the cuboid in this example is constructed using a rectangle and the path sweep tool.

Adding Variables

Define variables to create a parametric model.

A model is parametric when it is created using variable expressions. When a variable expression is modified, any items dependent on that variable are re-evaluated and automatically updated. It is the recommended construction method when creating a model, but not compulsory.

Defined variables are stored as part of the model in the `.cfx` file.

1. Open the **Create Variable** dialog using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Add Variable** icon.
   - On the model tree, a right-click context menu is available on **Variables**. From the list, select **Add Variable**.

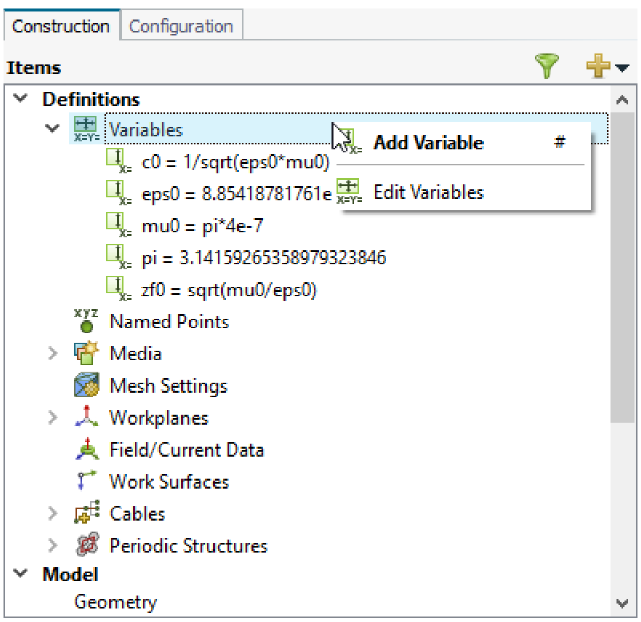


*Figure 29: The model tree (**Construction** tab).*

   - On the model tree, click the icon. From the drop-down list, select **Add Variable**.

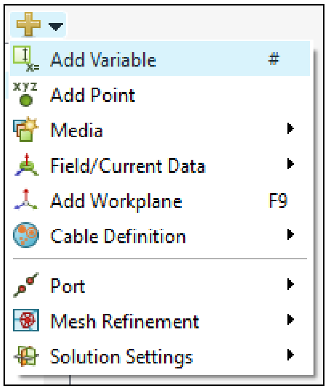


Figure 30: The drop-down list available in the model tree.

- Press # to use the keyboard shortcut.

2. Create the following variables:

Name	**Expression**	**Comment [Optional]**
Width	1	Width of rectangle.
Length	1	Length of rectangle.
BottomDepth	1	Bottom depth of flare.
BottomWidth	1	Bottom width of flare.
FlareLength	1	Length of flare.
TopWidth	2	Top width of flare.
TopDepth	2	Top depth of flare.

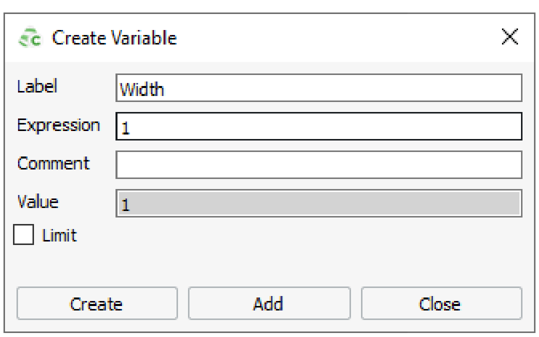


Figure 31: The ***Create Variable*** *dialog.*

Tip:

- Click **Add** to keep the **Create Variable** dialog open and add more variables.
- Click **Create** to add a variable and close the **Create Variable** dialog.

Defining a Workplane

Define a workplane to create an oblique plane. Workplanes simplify the process of creating geometry on oblique planes in comparison to using transforms.

The use of workplanes during construction is not compulsory, but is a more efficient method for creating geometry. For this example you will create a custom workplane and set as the default workplane.

Note: A workplane can be defined relative to another workplane.

1. Open the **Create Workplane** dialog using one of the following workflows:
    - On the **Construct** tab, in the **Define** group, click the **Add Workplane** icon.
    - On the model tree, a right-click context menu is available on the **Workplanes** group. Select **Add Workplane** from the drop-down list.

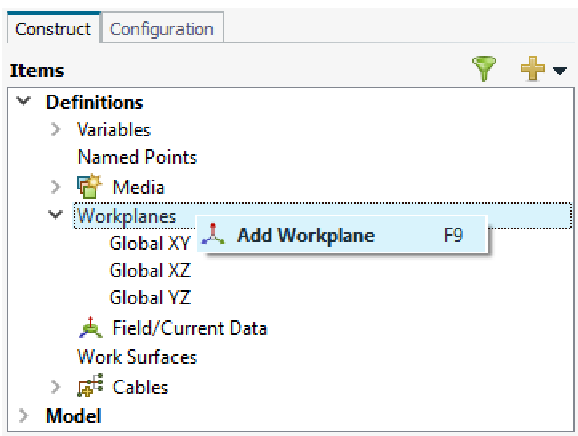


*Figure 32: The **Add Workplane** group is available on both the **Construct** and **Configuration** tabs in the model tree.*

    - On the model tree, click the icon. From the drop-down list, select **Add Workplane**.
    - Press F9 to use the keyboard shortcut.
2. On the **Create Workplane** dialog, from the drop-down list, select **Global YZ**.
3. Use the default workplane label, `Workplane1`.

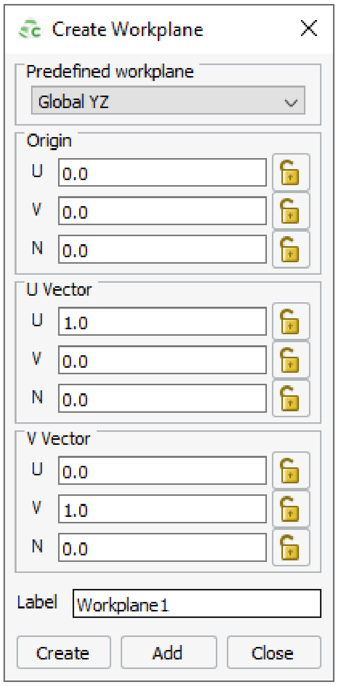


*Figure 33: The **Create Workplane** dialog.*

4. Click **Create** to create the workplane and to close the dialog.

The default workplane is used when creating new geometry primitives. For this example, set the new workplane as the default workplane.

5. In the model tree, select **Workplane1**.
   a) From the right-click context menu, select **Set as default**.

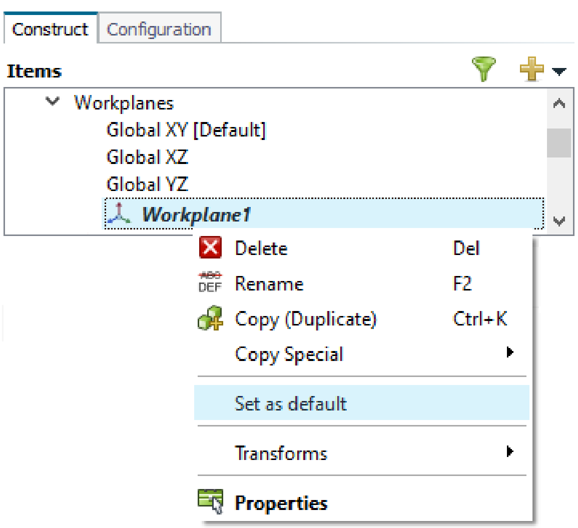
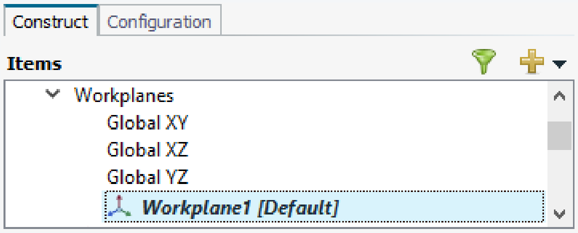


Figure 34: The right-click context menu options for workplanes.

Note: The current default workplane is indicated by the text, **[Default]**.



Creating a Rectangle

Create a rectangle to be used in the construction of the horn.

1. On the **Construct** tab, in the **Create Surface** group, click the **Rectangle** icon.

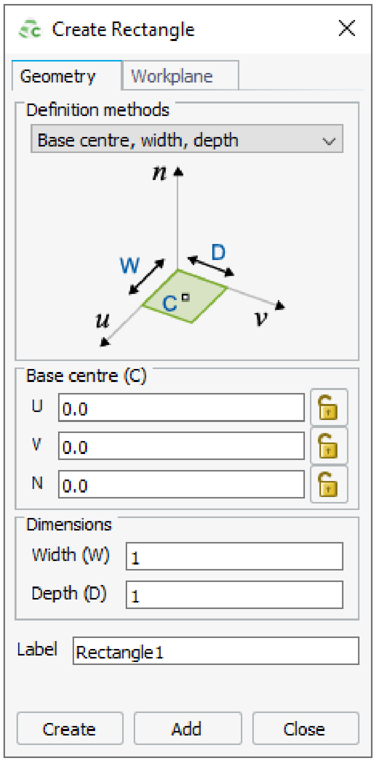


*Figure 35: The **Create Rectangle** dialog.*

> **Note:** Default values are used on geometry creation dialogs to allow a preview in the 3D view. You may change the values as required.

> **Tip:** An active field allowing point-entry is indicated by a yellow outline.
>
> Point-entry allows a variable or named points to be entered by pressing Ctrl+Shift+left click on a variable or named point in the model tree.

2. Create a rectangle using the **Base centre, width, depth** definition method.
   a) Use the following dimensions:
      - **Base centre (C)**: (0, 0, 0)
      - **Width (W)**: 1
      - **Depth (D)**: 1
3. Click **Create** to create the rectangle and to close the dialog.

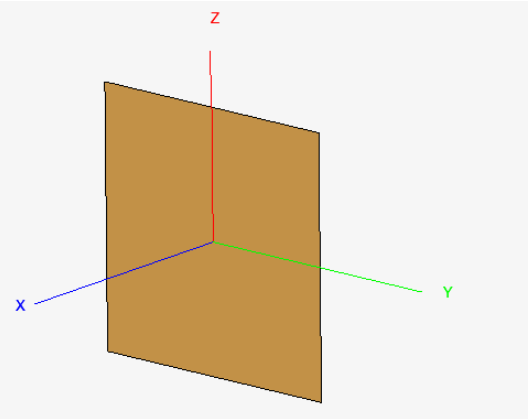


*Figure 36: The rectangle created using **Workplane1**.*

Creating a Line

Create a line to be used in the construction of the horn. This line will be used to create a cuboid by sweeping the rectangle along the line.

1. On the **Construct** tab, in the **Create Curve** group, click the **Line** icon.
2. On the **Create Line** dialog, enter the start point and end point for the line.
   - **Start point**: (0, 0, 0)
   - **End point**: (0, 0, 1)

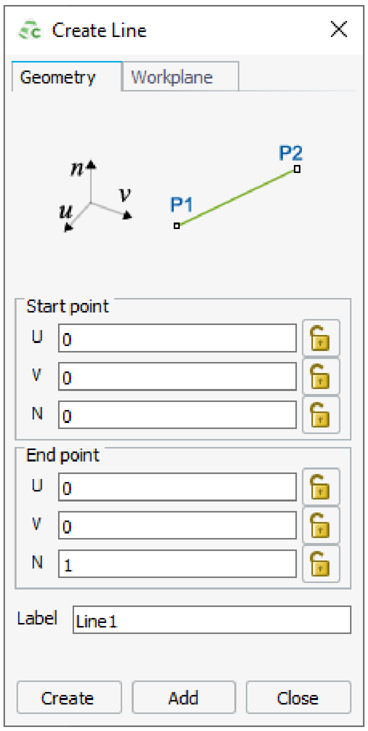


*Figure 37: The **Create Line** dialog.*

3. Click **Create** to create the line and to close the dialog.

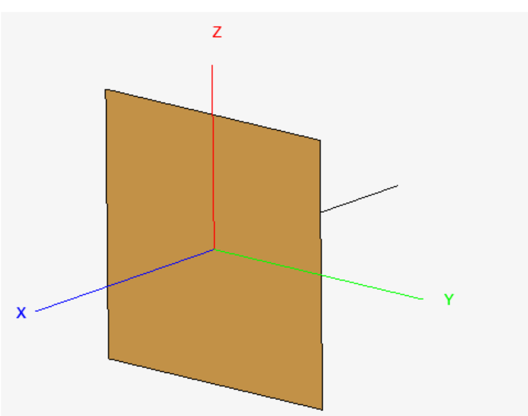


*Figure 38: The rectangle and line created using **Workplane1**.*

Creating a Cuboid by Sweeping a Rectangle along a Line

Create a cuboid by sweeping the rectangle along a line (path).

> **Tip:** For demonstrative purposes, a rectangle is swept along a path to create a cuboid. The preferred method to create a cuboid is to make use of the cuboid tool.

1. In the model tree, select **Rectangle1**.

   > **Note:** Selecting **Rectangle1** in the model tree enables **Path Sweep** on the ribbon.

2. On the **Construct** tab, in the **Extend** group, click the **Path Sweep** icon.

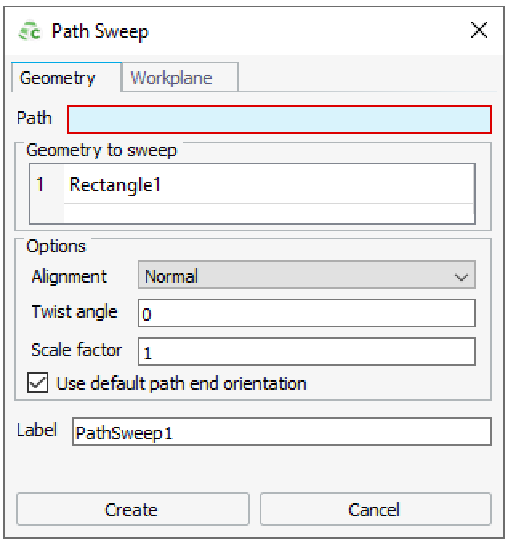


*Figure 39: The **Path Sweep** dialog.*

3. In the model tree, click **Line1** to use as path.
4. On the **Path Sweep** dialog, use the default values.
5. Click **Create** to create the path sweep and to close the dialog.

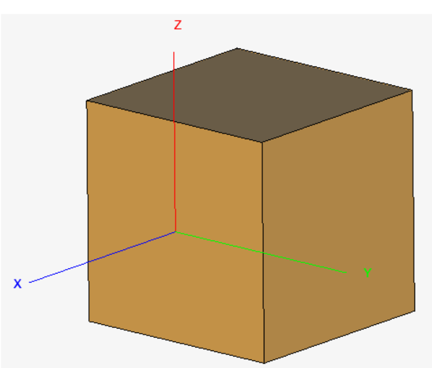


Figure 40: The rectangle swept along a path (line) to create a cuboid.

Creating a Flare

Create a flare primitive to be used in the construction of the horn.

1. On the **Construct** tab, in the **Create Solid** group, click the **Flare** icon.
2. Create the flare using the **Base centre, width, depth, height, top width, top depth** method.
3. Specify the flare dimensions using one of the following workflows:
   - Add the defined variables manually.
   - Select a field on the **Create Flare** dialog and use point-entry to enter the values.

   > **Note:** An active field allowing point-entry is indicated by a yellow outline.
   >
   > Point-entry allows a variable or named points to be entered by pressing Ctrl+Shift+left click on a variable or named point in the model tree.

   Use the following dimensions:

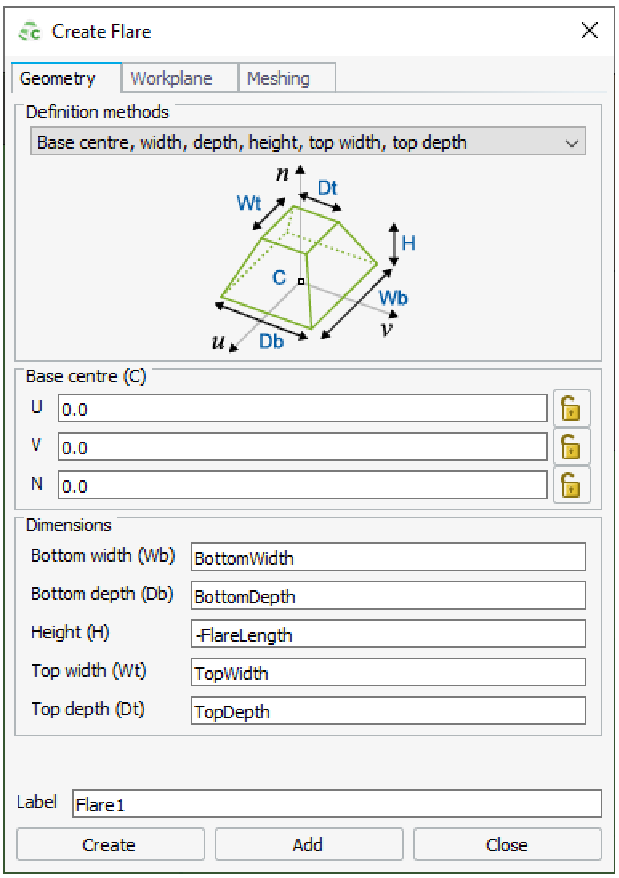


*Figure 41: The **Create Flare** dialog.*

   - **Base centre (C)**: (0, 0, 0)
   - **Bottom width (Wb)**: *BottomWidth*
   - **Bottom depth (Db)**: *BottomDepth*
   - **Height (H)**: *-FlareLength*

- **Top width (Wt)**: *TopWidth*
- **Top depth (Dt)**: *TopDepth*

> **Tip:** Parametric models are the preferred construction method. A parametric model updates automatically when updating a defined variable.
>
> Alternatively, use values instead of defined variables.

4. View the preview of the flare in the 3D view. Confirm that the model looks correct.

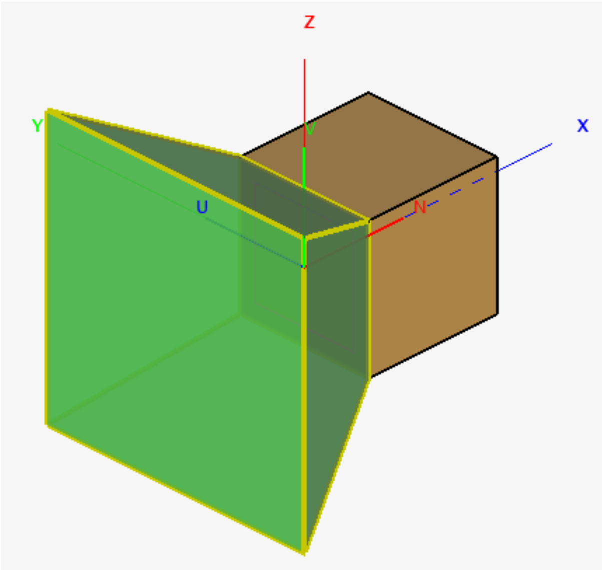


Figure 42: The preview of the flare is indicated in green.

5. Click **Create** to create the flare and to close the dialog.

Creating a Union of the Flare and Cuboid

Union the flare and cuboid to create the horn.

> **Note:** The union operation is used to define connectivity between parts.
>
> Parts that touch, but are not unioned, are not considered to be physically connected and will result in an incorrect mesh.

1. In the model tree, select the flare and the cuboid (**PathSweep1**).

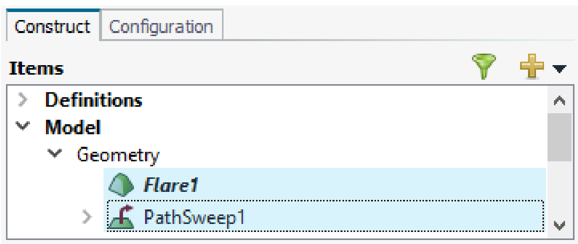


*Figure 43: The **Construction** tab in the model tree showing the selected **Flare1** and **PathSweep1**.*

2. Union using one of the following workflows:
   - On the **Construct** tab, in the **Modify** group, click the **Union** icon.
   - Press U to use the keyboard shortcut.

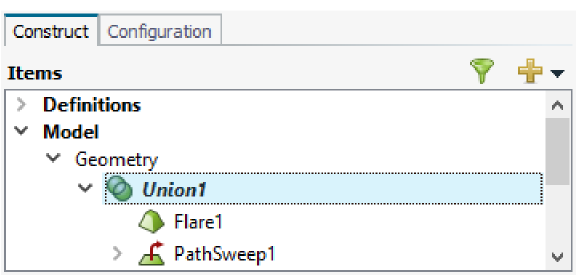


*Figure 44: The **Construction** tab in the model tree showing the unioned part, **Union1**.*

Removing Redundant Geometry Faces

Delete the redundant faces in the geometry to create a horn.

1. In the model tree, select **Union1**.
2. In the details tree, under **Faces**, go through the list of faces. For each face, click on ![icon] to hide the face until only the two redundant faces remain.

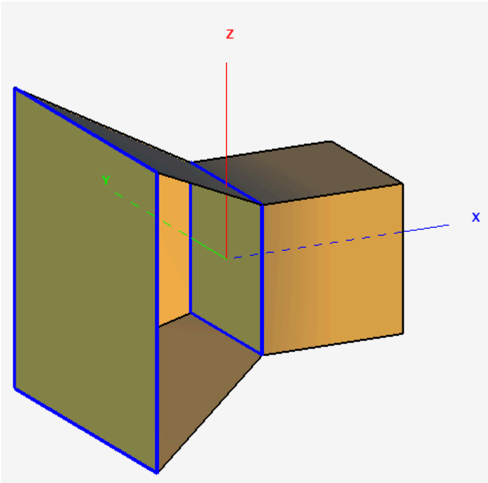


Figure 45: For illustration, a side face is hidden to show the two redundant faces highlighted in yellow with a blue outline.

3. Select the two faces that remain.
4. From the right-click context menu, click **Delete**.

> **Note:** Deleting one of a regions's enclosing faces, removes the PEC region.

5. Select any of the remaining faces and from the right-click context menu, click **Visibility** > **Show All in Model**.

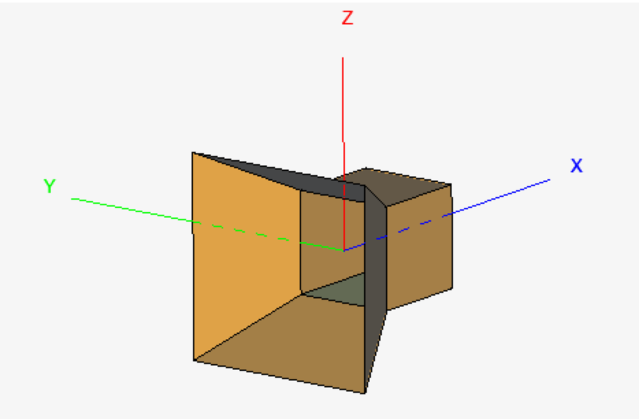


Figure 46: The redundant faces were deleted.

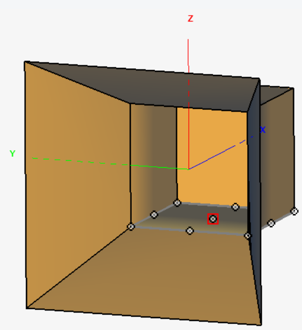
2.4.4 Adding a Feed Pin to the Horn

Add a wire feed to the model. As this example is only for demonstration purposes, this example does not cover the adding of a port or source to the wire feed.

Creating the Feed Wire

Create a wire feed for the model.

1. On the **Construct** tab, in the **Create Curve** group, click the **Line** icon.
2. On the **Create Line** dialog, click the **Workplane** tab.
   a) On the **Workplane** tab, select **Custom** workplane.
   b) Under **Origin**, click on **X** field to make point-entry active (indicated by a yellow outline).
3. Press Ctrl+Shift while moving the mouse cursor over the bottom face centre of the cuboid.

> **Note:** The circles with a black outline indicate special snapping points. The red outline indicates the position of the mouse cursor.
>
> Use snapping points to snap the workplane to an object. Although only special snapping points are indicated, you can snap to any point in the 3D view.
>
> 

4. Press Ctrl+Shift+left click to snap the workplane to the bottom face centre of the cuboid.
5. On the **Create Line** dialog, click the **Geometry** tab.
   a) Create a line.
      - **Start point**: (0, 0, 0)
      - **End point**: (0, 0, 0.25)
      - **Label**: `Feed`
6. Click **Create** to create the line and to close the dialog.

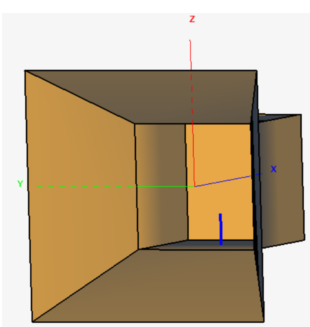


Figure 47: The feed wire is selected (highlighted in blue).

Unioning the Feed Wire and Horn

Union the feed wire and horn to ensure mesh connectivity and a correct mesh.

1. In the model tree, expand **Union1**.
2. In the model tree, select **Feed** and drag it to below **Union1**.
3. From the right-click context menu, select **Move in**.
4. View the model tree and confirm that it is correct.

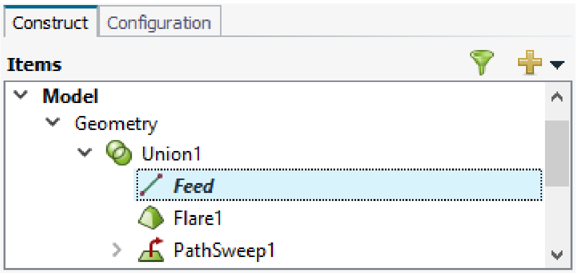


Figure 48: Union by dragging as item into a union in the model tree.

2.4.5 Using Selection in the 3D View

Use the selection type tool to highlight an element in the 3D view.

The following steps are not required for constructing the model, but it illustrates how selection works in CADFEKO.

1. Move the mouse cursor to one of the faces on the inside of the flare.
2. Click on a face.

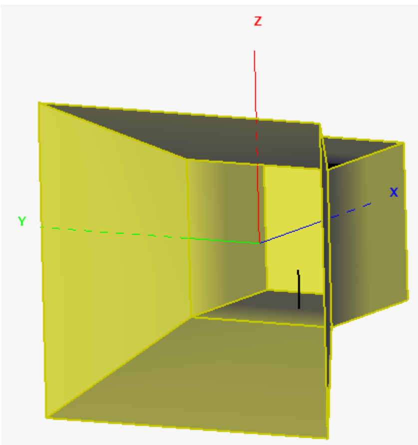


Figure 49: The part is selected and highlighted in yellow.

3. Click again on the face.

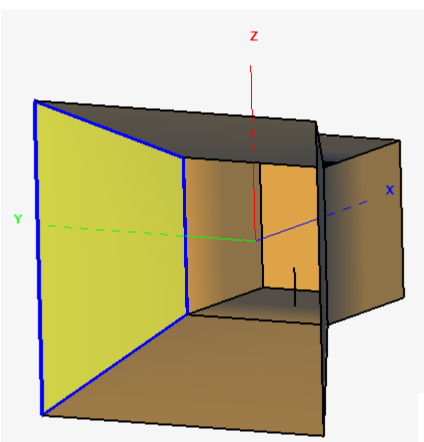


Figure 50: The face is selected and highlighted in yellow with a blue outline.

> **Note:**
> The default selection method (**Auto**) cycles through the applicable selection types when repeatedly clicking on the model.
>
> The first click selected the part. The second click selected the face.

4. Change the selection type using one of the following workflows:

- On the **Tools** tab, in the **Selection** group, click the **Selection Type** icon.
- On the **status bar**, click **Selection Type** icon. Select the required selection type from the list. Selection:

2.4.6 Creating an Aperture in a Face

Create an aperture (hole) in a face or region by using the subtract tool. Create the geometry to be removed and subtract it from the target part. The target is the part that is reduced by cutting away a section of the part.

Creating and Placing the Ellipse

Create an ellipse to subtract from the horn. The ellipse is placed on the face of the horn.

1. On the **Construct** tab, in the **Create Surface** group, click the **Ellipse** icon.
2. On the **Create Ellipse** dialog (**Geometry** tab), create an ellipse using the following dimensions:
   - **Centre point (C)**: (0, 0, 0)
   - **Radius (Ru)**: 0.3
   - **Radius (Rv)**: 0.2
3. On the **Create Ellipse** dialog, click the **Workplane** tab.
   a) On the **Workplane** tab, select **Custom** workplane.
   b) Under **Origin**, click on **X** field to make point-entry active (indicated by a yellow outline).
   c) Move the mouse cursor over the flare while holding down Ctrl+Shift until the local workplane is orientated as displayed in the image.

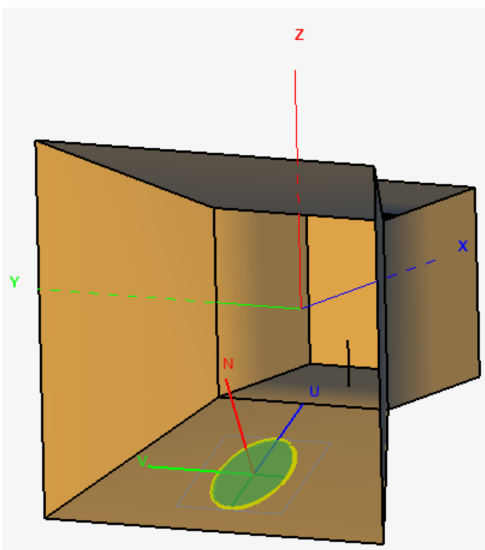


Figure 51: The placement of the ellipse on the horn.

> **Note:** The history of from where the mouse cursor was moved to the face centre, affects the orientation of the workplane.

4. Click **Create** to create the ellipse and to close the dialog.

Subtracting the Ellipse from the Horn

The ellipse is subtracted from the horn to create a hole.

1. In the model tree, select **Ellipse1**.
2. Subtract using one of the following workflows:
    - On the **Construct** tab, in the **Modify** group, click the **Subtract From** icon.
    - On the model tree, a right-click context menu is available on the primitive. From the list select **Apply** > **Subtract From**.

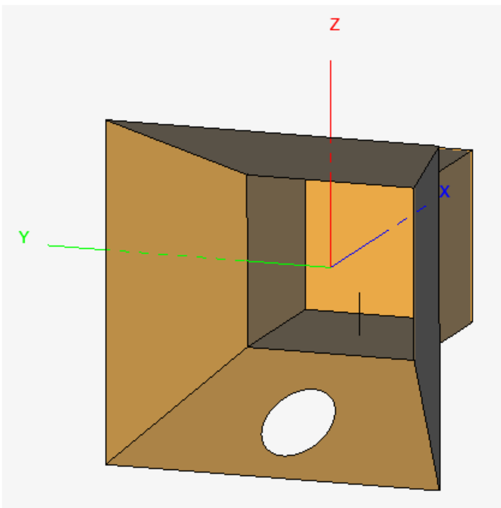
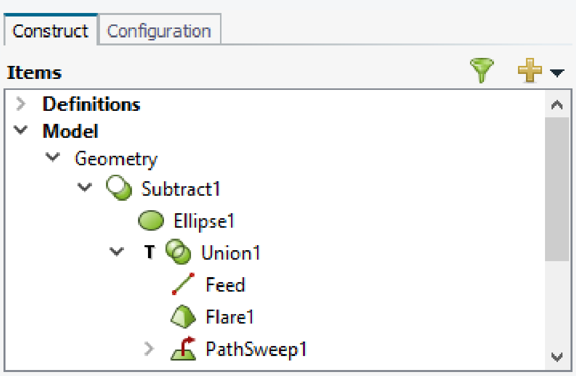


Figure 52: The ellipse was subtracted from the horn.

3. In the model tree, select **Union1**.

> **Note:** The **T** in the model tree indicates the target (object that was subtracted from).
>
> 
>

4. Click **Create** to subtract the ellipse and to close the dialog.

2.4.7 Setting the Simulation Frequency

Specify the frequency range of interest. For this example, a single frequency point is used.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.
2. In the **Frequency (Hz)** field, enter *1e9*.

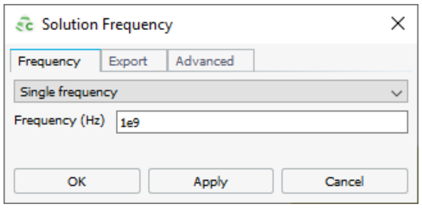


*Figure 53: The **Solution Frequency** dialog.*

3. Click **OK** to set the frequency and to close the dialog.

2.4.8 Saving the Model

Save the model to a CADFEKO .cfx file.

1. Save the model using one of the following workflows:
   - On the **Home** tab, in the **File** group, click the **Save** icon.
   - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Model_creation.cfx`.
3. Click **Save** to close the dialog.

2.5 Final Remarks

This example showed aspects of model creation in CADFEKO.

> **Important:** This example is not an example intended for simulation, but rather an introductory example that illustrates the power of CADFEKO when creating complex models.

GPS Patch Antenna 3

The example considers a left-handed circular polarised GPS patch antenna on a finite substrate.

This chapter covers the following:

3.1 Example Overview

Calculate the input reflection coefficient and circular components of a left-handed circular polarised GPS patch antenna on a finite substrate close to 1.57 GHz.

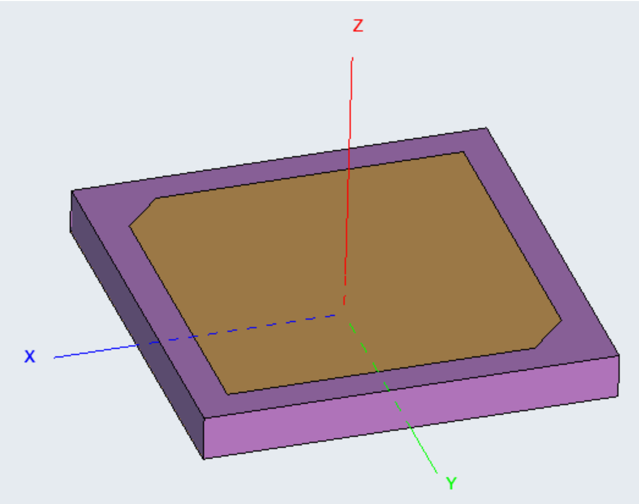


Figure 54: The chamfered GPS patch antenna on a finite substrate.

3.2 Topics Discussed in Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- CADFEKO
    - Activate macro recording of a model.
    - Changing the model unit.
    - Create and group variables.
    - Create a dielectric.
    - Create geometry (polygon, cuboid and line).
    - Set the region of a cuboid to dielectric.
    - Set the faces of a dielectric cuboid to PEC[10].
    - Add a voltage source to a wire segment.
    - Modify the auto-generated mesh.
    - Add a far field request.
    - Deactivate macro recording and run the resulting Feko Lua script.
    - Run the Solver.
- POSTFEKO
    - View the input reflection coefficient on a Cartesian graph.
    - View the left-hand and right-hand circular components of the far field on a Cartesian graph.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[11]:
>
> `GS 3: GPS Patch Antenna`

10. perfect electric conductor
11. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

3.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 40 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[12] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

12. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

3.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

3.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

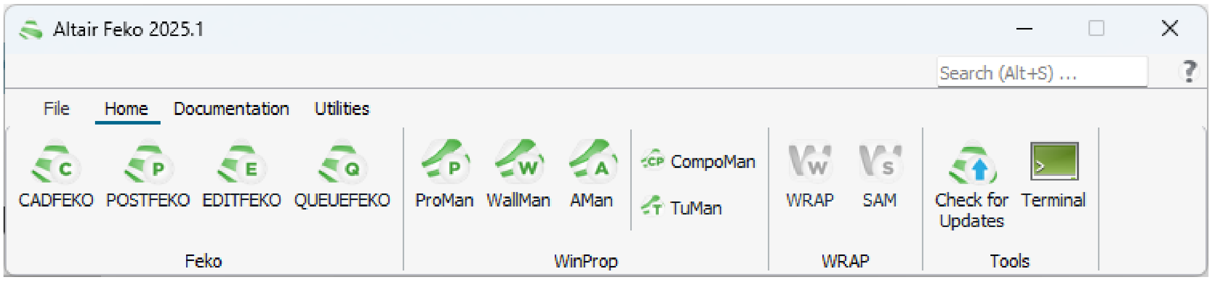


Figure 55: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[13] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

3.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the "initfeko" script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

13. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

3.4.3 Activating Macro Recording of Model

Use macro recording to record actions in a script. Play the script back to automate the process or view the script to learn the Lua-based scripting language by example. Macro recording allows you to perform repetitive actions faster and with less effort.

> **Note:** This step is optional when creating a model in CADFEKO but it is included in this example to highlight the functionality.

Activate macro recording using one of the following workflows:

- On the **Home** tab, in the **Scripting** group, click the **Record Macro** icon.
- Click the icon in the status bar.

3.4.4 Setting the Model Unit

Set the model unit to millimeters.

The default unit length in CADFEKO is metres. Since the structure that you will build is small, the model unit is set to millimetres. All dimensions entered will be in the new model unit.

1. Set the model unit to millimetres using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Model unit** icon.
   - On the status bar, click Unit: m.
2. On the **Model Unit** dialog, select **Millimetres (mm)**.
3. Click **OK** to change the model unit to millimetres and to close the dialog.

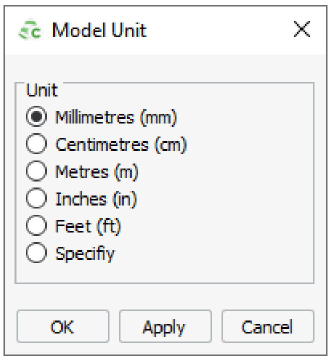


*Figure 56: The **Model Unit** dialog.*

3.4.5 Adding Variables

Define variables to create a parametric model.

A model is parametric when it is created using variable expressions. When a variable expression is modified, any items dependent on that variable are re-evaluated and automatically updated. It is the recommended construction method when creating a model, but not compulsory.

Defined variables are stored as part of the model in the `.cfx` file.

1. Open the **Create Variable** dialog using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Add Variable** icon.
   - On the model tree, a right-click context menu is available on **Variables**. From the list, select **Add Variable**.

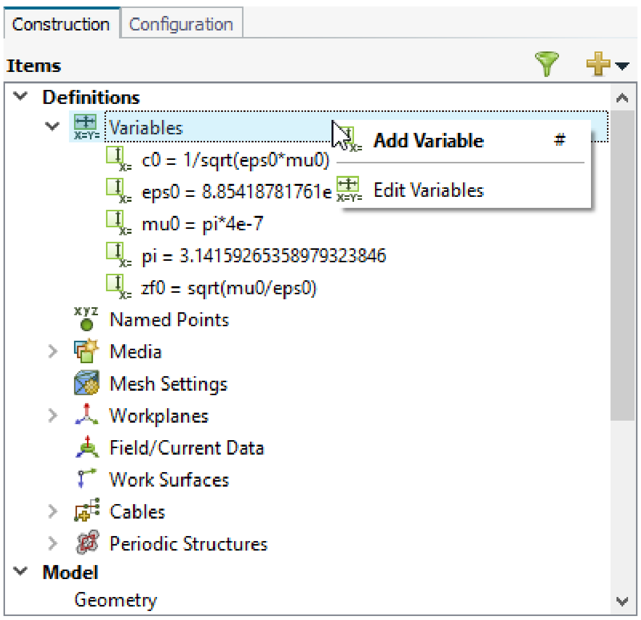


*Figure 57: The model tree (**Construction** tab).*

   - On the model tree, click the icon. From the drop-down list, select **Add Variable**.

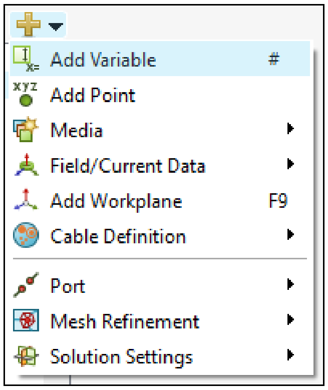


Figure 58: The drop-down list available in the model tree.

- Press # to use the keyboard shortcut.

2. Create the following variables:

Name	Expression	Unit
patch_size	18.8	mm
chamfer_d	4.3	mm
feed_pos	-6.4	mm
substrate_w	45	mm
substrate_d	45	mm
substrate_h	5	mm
ceramic_epsR	5.6	
ceramic_tanD	0.0041	

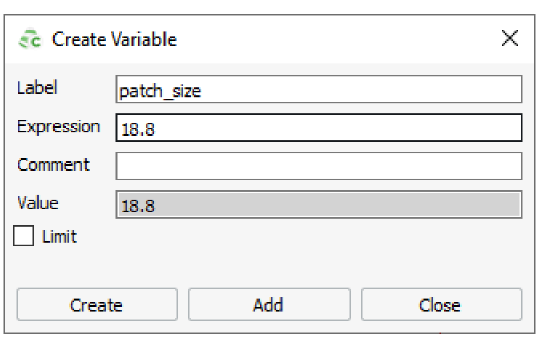


*Figure 59: The **Create Variable** dialog.*

> **Tip:**
> - Click **Add** to keep the **Create Variable** dialog open and add more variables.
> - Click **Create** to add a variable and close the **Create Variable** dialog.

3. [Optional] Group the variables related to the patch.
   a) In the model tree, under **Variables**, select *patch_size* and *chamfer_d*.

   > **Tip:** To select multiple objects, press and hold Ctrl while you click the items.

   b) From the right-click context menu, select **Group** > **Create**.
   c) Select **Group1** and from the right-click context menu, click **Rename**.
   d) Rename the group to `Patch`.

4. [Optional] Group the variables related to the substrate.
   a) In the model tree, select *substrate_w*, *substrate_d* and *substrate_h*.
   b) From the right-click context menu, select **Group** > **Create**.
   c) Select **Group2** and from the right-click context menu, click **Rename**.

   > **Tip:** Press F2 to use the keyboard shortcut to rename a selected item.

   d) Rename the group to `Substrate`.

3.4.6 Defining a Dielectric Medium

Define a lossy frequency-independent dielectric with a relative permittivity (ε_r) = 5.6 and a dielectric loss tangent ($\tan\delta$) = 0.0041 to be used as the patch substrate.

1. Open the **Create Dielectric Medium** dialog using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Media** icon. From the drop-down list, click the **Dielectric** icon.
   - On the model tree, a right-click context menu is available on **Media**. From the list, click **Dielectric**.

Two variables (*ceramic_epsR* and *ceramic_tanD*) were added to the model to define the dielectric.

2. Set the **Relative permittivity** (ε_r) to *ceramic_epsR*.
3. Set the **Dielectric loss tangent** ($\tan\delta$) to *ceramic_tanD*.

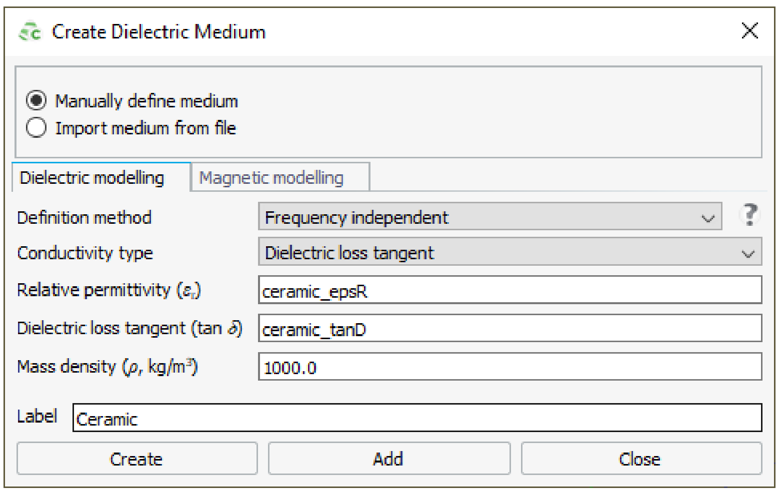
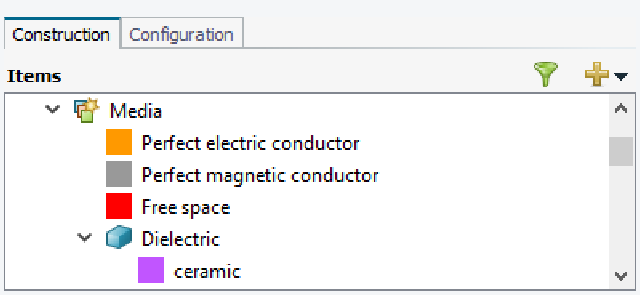


*Figure 60: The **Create Dielectric Medium** dialog.*

4. Set the **Label** to `Ceramic`.
5. Click **Create** to create the dielectric and to close the dialog.

> **Note:** In the model tree, the defined dielectric is displayed under **Dielectric**. CADFEKO assigns a colour to each medium randomly but the colour may be changed by using the **Change Display Colour** right-click context menu option.



3.4.7 Creating the Patch

Create the chamfered[14] patch using a polygon.

1. On the **Construct** tab, in the **Create Surface** group, click the **Polygon** icon.

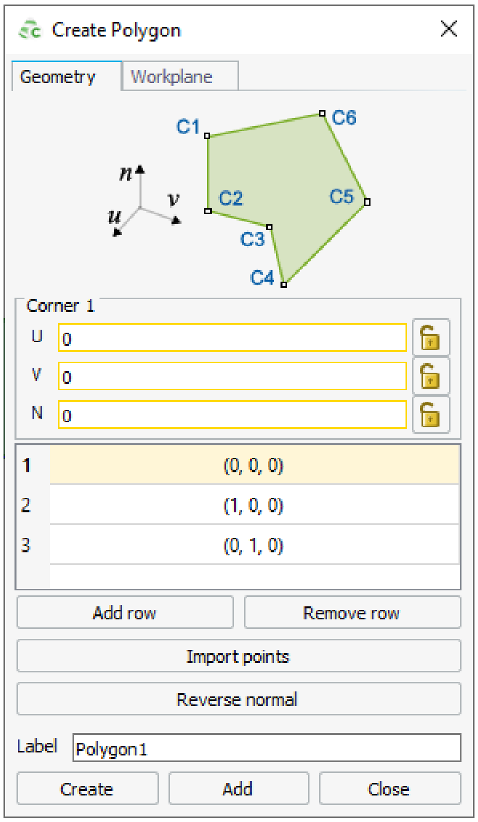


*Figure 61: The **Create Polygon** dialog showing the default values. Active fields are outlined in yellow.*

> **Note:** Default values are used on geometry creation dialogs to allow a preview in the 3D view. You may change the values as required.

> **Tip:** An active field allowing point-entry is indicated by a yellow outline.
>
> Point-entry allows a variable or named points to be entered by pressing Ctrl+Shift+left click on a variable or named point in the model tree.

2. Under **Corner 1**, add the following coordinates:
   - Corner 1:
     - **U**: *Patch.patch_size*
     - **V**: *Patch.patch_size*
     - **N**: *Substrate.substrate_h*

14. An edge created at 45° between two adjoining right-angled edges.

3. In the table, click on the second row to make **Corner 2** active. Add the following coordinates:
   - Corner 2:
     - **U**: *-Patch.patch_size + Patch.chamfer_d*
     - **V**: *Patch.patch_size*
     - **N**: *Substrate.substrate_h*
4. Click on the third row to make **Corner 3** active. Add the following coordinates:
   - Corner 3:
     - **U**: *-Patch.patch_size*
     - **V**: *Patch.patch_size - Patch.chamfer_d*
     - **N**: *Substrate.substrate_h*
5. Click **Add row** for **Corner 4**. Add the following coordinates:
   - Corner 4:
     - **U**: *-Patch.patch_size*
     - **V**: *-Patch.patch_size*
     - **N**: *Substrate.substrate_h*
6. Repeat Step 5 twice to add **Corner 5** and **Corner 6** using the following coordinates:
   - Corner 5:
     - **U**: *Patch.patch_size - Patch.chamfer_d*
     - **V**: *-Patch.patch_size*
     - **N**: *Substrate.substrate_h*
   - Corner 6:
     - **U**: *Patch.patch_size*
     - **V**: *-Patch.patch_size + chamfer_d*
     - **N**: *Substrate.substrate_h*
7. Set the **Label** to `patch`.

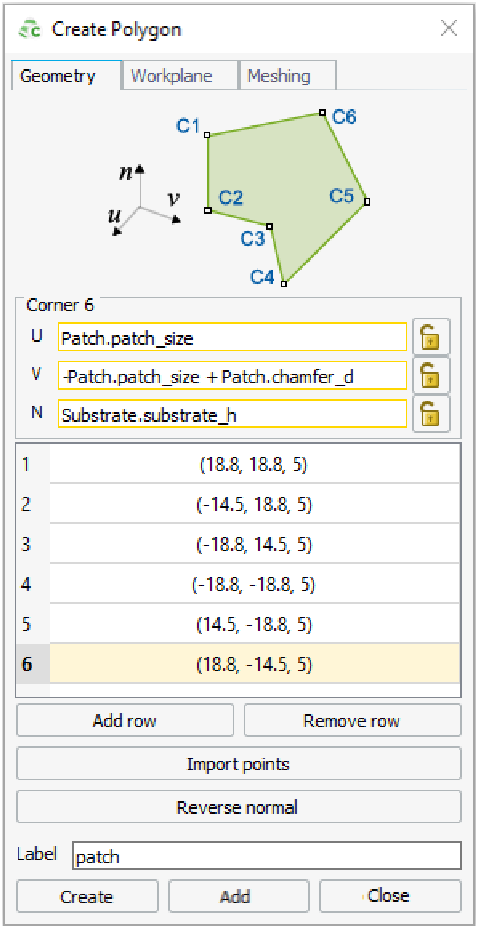


*Figure 62: The **Create Polygon** dialog.*

8. Click **Create** to create the polygon and to close the dialog.

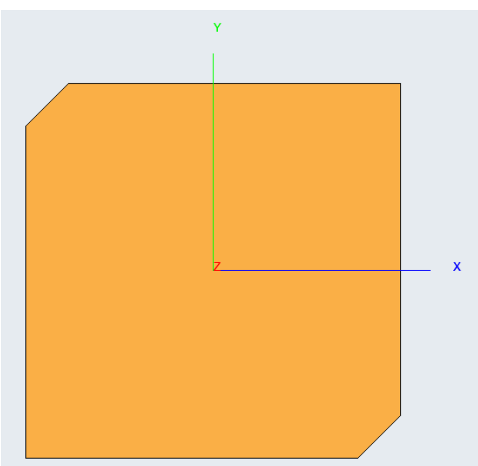


Figure 63: Top view of the chamfered patch. Note that the face is set to perfect electric conductor (PEC) by default (PEC is indicated by the colour orange).

3.4.8 Creating the Patch Substrate

Create a finite substrate[15] by creating a cuboid. Set the region of the cuboid to the medium, *ceramic*.

Create the cuboid.

a) On the **Construct** tab, in the **Create Solid** group, click the **Cuboid** icon.

b) Create the cuboid using the **Base corner, width, depth, height** definition method.

c) Use the following dimensions:

- **Base corner (C)**: (-22.5, -22.5, 0)
- **Width (W)**: *Substrate.substrate_w*
- **Depth (D)**: *Substrate.substrate_d*
- **Height (H)**: *Substrate.substrate_h*
- **Label**: `substrate`

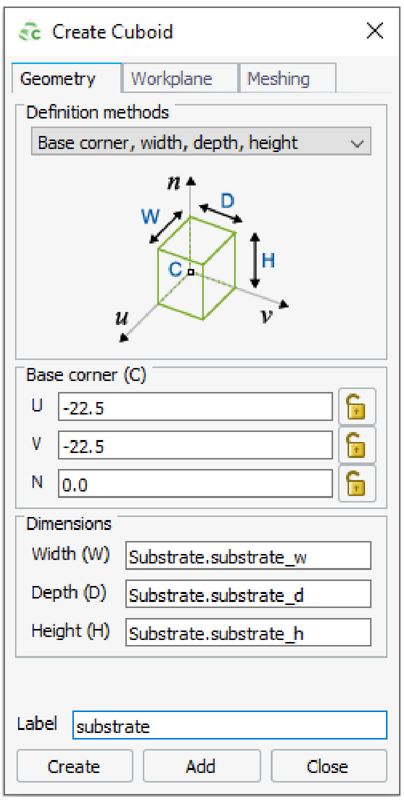


*Figure 64: The **Create Cuboid** dialog.*

d) Click **Create** to create the substrate and to close the dialog.

15. An alternative method is to model the substrate using an infinite planar multilayer substrate. See the Feko Example Guide for an example.

3.4.9 Setting a Region to a Dielectric

Change the region property of the substrate to dielectric.

1. In the model tree, select **substrate**.
2. In the details tree, under **Regions**, select **Region1**.
3. From the right-click context menu, select **Properties**.

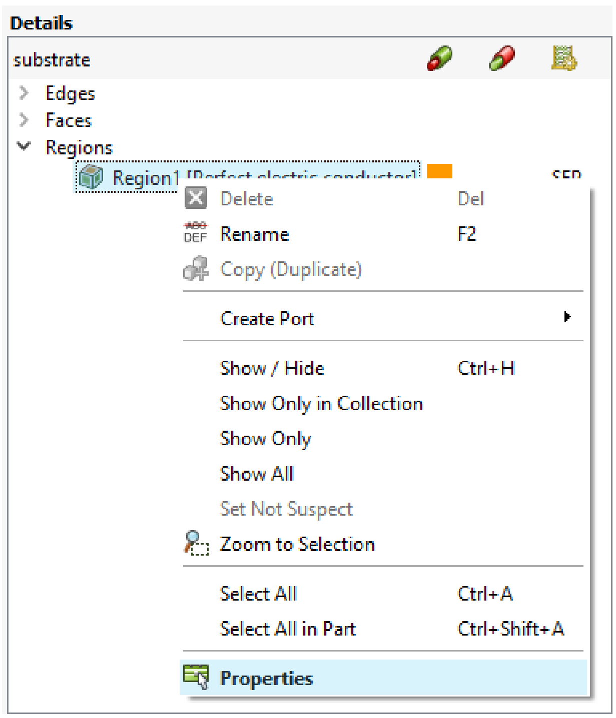


Figure 65: The right-click context menu options available for regions.

4. On the **Modify Region** dialog (**Properties** tab), set **Medium** to **Ceramic**.

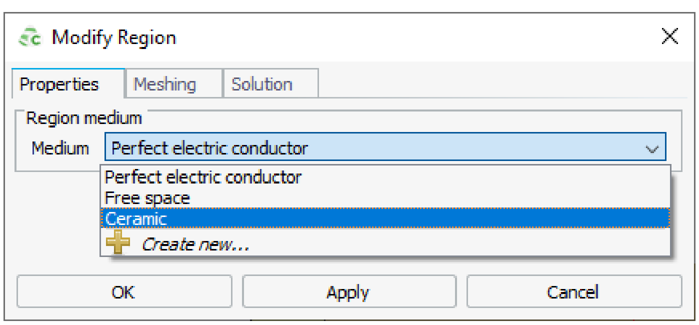
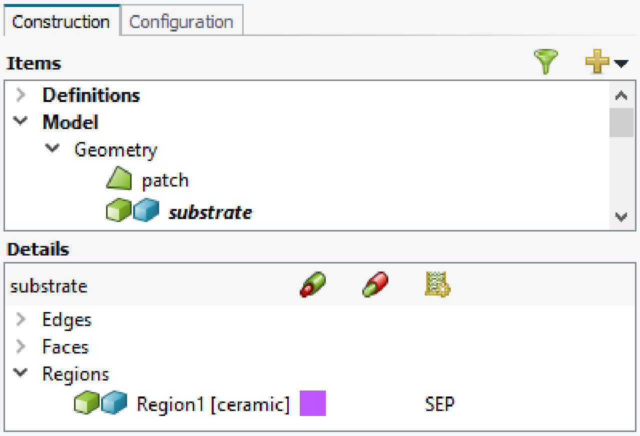


*Figure 66: The **Modify Region** dialog.*

> **Tip:** Simplify workflow by using **Create new** to define items when needed.

5. Click **OK** to modify the region property and to close the dialog.

Note: The icon in the model tree and details tree indicate items set to dielectric.



3.4.10 Creating the Feed Pin

Create the feed pin using a single line element.

1. On the **Construct** tab, in the **Create Curve** group, click the **Line** icon.
2. On the **Create Line** dialog, enter the start and end point for the line.
   - **Start point**: (0, *feed_pos*, 0)
   - **End point**: (0, *feed_pos*, *Substrate.substrate_h*)
   - **Label**: `feed_line`

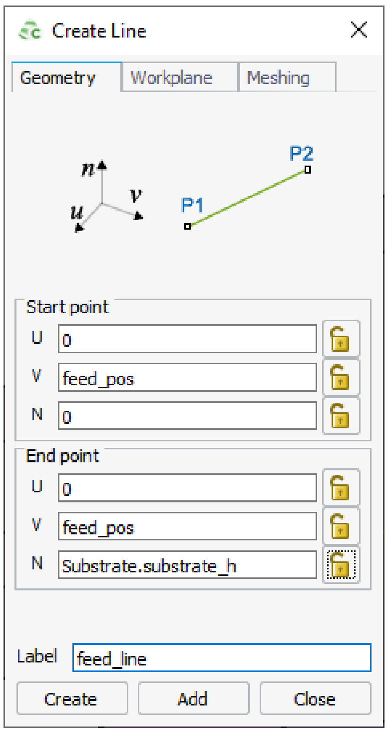


*Figure 67: The **Create Line** dialog.*

3. Click **Create** to create the line and to close the dialog.

3.4.11 Unioning the Geometry for Mesh Connectivity

Union the geometry (*feed_line*, *patch* and *substrate*) to create a single geometry part. A single geometry part will ensure mesh connectivity when the model is meshed.

1. In the model tree, select **feed_line**, **patch** and **substrate**[16].

   Tip: To select multiple objects, press and hold Ctrl while you click the items.

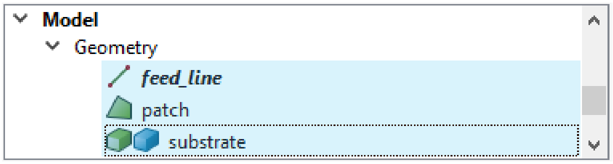


Figure 68: The three geometry parts in the model tree.

2. On the **Construct** tab, in the **Modify** group, click the **Union** icon.

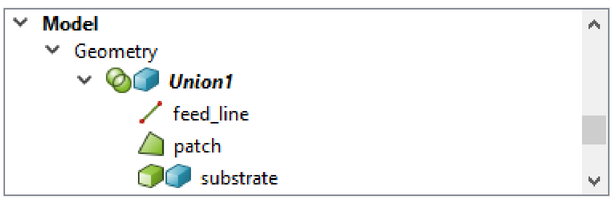


Figure 69: The model tree showing ***Union1*** *(the union between* ***feed_line****,* ***patch*** *and* ***substrate****).*

16. Alternative method is to select the items in the 3D view.

3.4.12 Setting Faces to PEC

Change the surface property of the patch to perfect electric conductor (PEC).

1. Change the face of the patch to PEC.

a) In the 3D view, left-click on the patch face repeatedly until the face is highlighted in yellow.

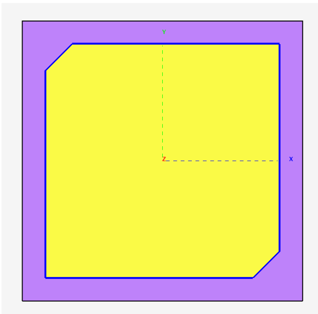


Figure 70: Top view of patch and substrate. The yellow highlighting indicates that the patch face is selected.

b) From the right-click context menu, select **Properties**.

c) On the **Modify Face** dialog **(Properties** tab), set the **Medium** to **Perfect electric conductor**.

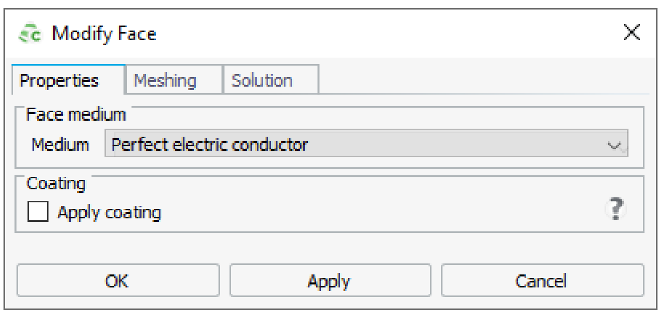


*Figure 71: The **Modify Face** dialog.*

d) Click **OK** to change the face property and to close the dialog.

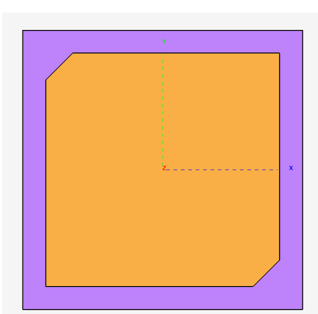


Figure 72: Top view showing the face of the patch set to PEC.

2. Change the face of the bottom substrate to PEC.

a) In the model tree, select **Union1**.
b) In the details tree, under **Faces**, go through the list of faces. For each face, click on to hide the face until only the bottom face[17] of the substrate remains.

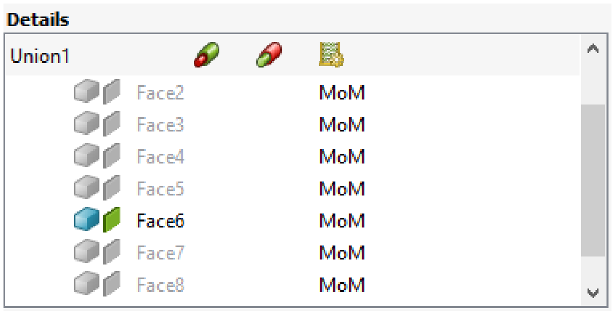


Figure 73: Hidden items are greyed out when hidden in the 3D view.

c) From the right-click context menu, select **Properties**.
d) On the **Modify Face** dialog (**Properties** tab), set **Medium** to **Perfect electric conductor**.
e) Click **OK** to modify the face property and to close the dialog.

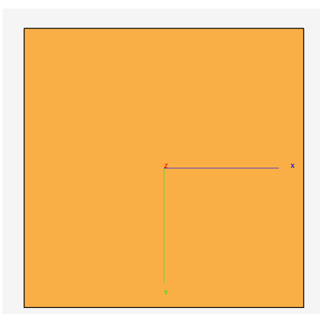
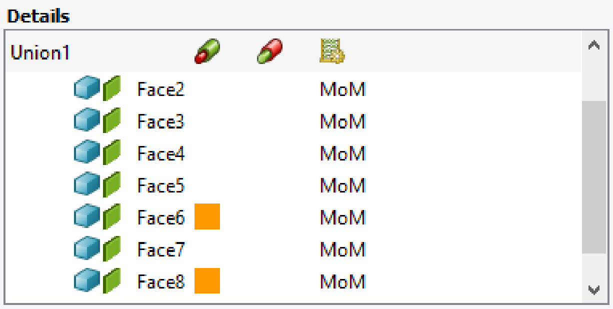


Figure 74: Bottom view showing the bottom substrate face set to PEC.

f) In the details tree, click on any of the faces. From the right-click context menu, click **Visibility**, **Show All in Model** to make all faces visible again.

17. numbering could differ

Note: The icon in the details tree indicate faces set to PEC.



3.4.13 Ports, Sources and Loads in CADFEKO

Voltage sources and discrete loads are applied to ports and not directly to the model geometry or mesh. A port must be defined before a source or load can be added.

Creating the Port

Define a wire port on the feed pin. A voltage source will be added to this port.

> **Note:** A port is a mathematical representation of where energy can enter (source) or leave a model (sink). Use a port to add sources and discrete loads to a model.

1. Open the **Create Wire Port** dialog using one of the following workflows:
   - On the **Source/Load** tab, in the **Ports** group, click the **Wire Port** icon.
   - In the details tree, a right-click context menu is available on the wire. From the list, click **Create port** > **Wire Port**.
2. Select the wire where the port is to be added.
   a) In the model tree, select **feed_line**.
   b) In the details tree, select the wire of **feed_line**.
3. Under **Location on wire**, select **Start**.

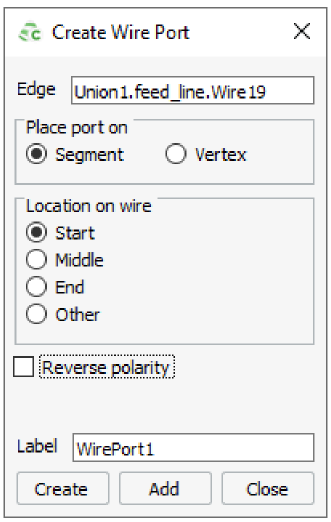


Figure 75: The ***Create Wire Port*** *dialog.*

4. Click **Create** to create the port and to close the dialog.
5. Change the label to `Port1`.

Adding a Voltage Source

Add a voltage source to the port of the pin.

1. On the **Source/Load** tab, in the **Sources on Ports** group, click the **Voltage Source** icon.
2. On the **Add Voltage Source** dialog, use the default settings.

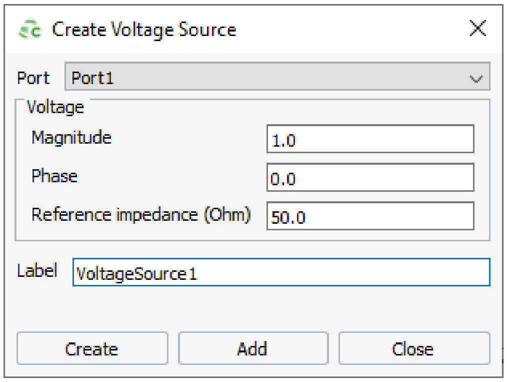


*Figure 76: The **Create Voltage Source** dialog.*

3. Click **Create** to define the voltage source and to close the dialog.

> **Note:** The **Configuration** tab was selected automatically when you defined the voltage source. You may also add sources, loads and set the frequency from here.

3.4.14 Setting the Simulation Frequency

Specify the frequency range of interest. For this example continuous frequency sampling is used where Feko automatically determines the frequency sampling for optimal interpolation.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.
2. On the **Solution Frequency** dialog, from the drop-down list, select **Continuous (interpolated) range**.
3. In the **Start frequency (Hz)** field, enter `1.27e9`.
4. In the **End frequency (Hz)**, enter `1.85e9`.

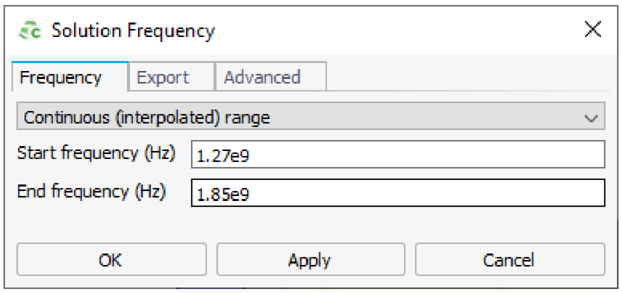


*Figure 77: The **Solution Frequency** dialog.*

5. Click **OK** to specify the frequency and to close the dialog.

3.4.15 Modifying the Auto-Generated Mesh

When the frequency is set or local mesh settings are applied to the geometry, the automatic mesh algorithm calculates and creates the mesh automatically while the GUI is active using default mesh settings. When required, these mesh settings may be modified.

The patch requires a finer mesh as the standard mesh size[18] and a wire segment radius needs to be specified.

1. Open the **Modify Mesh Settings** dialog using one of the following workflows:
   - On the **Mesh** tab, in the **Meshing** group, click the **Create Mesh** icon.
   - Press Ctrl+M to use the keyboard shortcut.
2. Set the **Mesh size** to **Fine**.
3. Set the **Wire segment radius** to `0.7`.

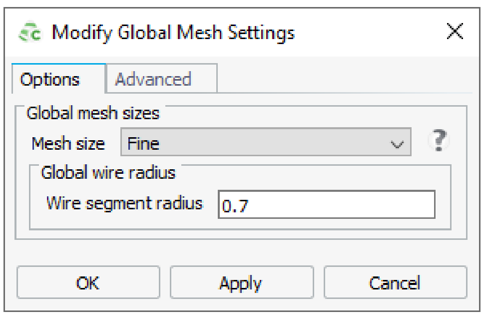


*Figure 78: The **Modify Mesh Settings** dialog.*

4. Click **OK** to create the mesh and to close the dialog.

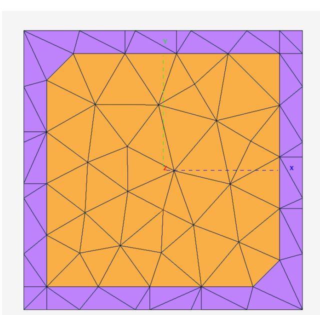


Figure 79: Top view of the patch and substrate showing the mesh.

5. View the effect in the 3D view of specifying a **Wire segment radius**.
   a) Press F5 to use the keyboard shortcut to zoom to extents the 3D view.
   b) Enable a default cutplane. In the model tree (**Construction** tab), under **Cutplanes**, click the icon next to **XZ-Cut**[19] and from the right-click context menu, select **Flip cutplane**.

18. See the Feko User Guide Appendix A-3 for more information regarding automatic mesh sizes.
19. To change the default cutplane settings, double-click on the cutplane text (for example, **XZ-Cut**).

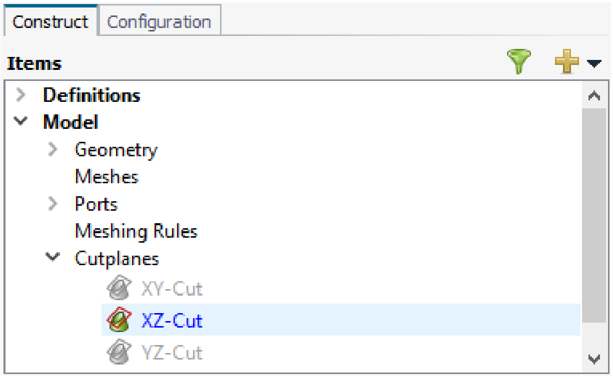


Figure 80: Note that a cutplane icon is greyed out when the cutplane is not active.

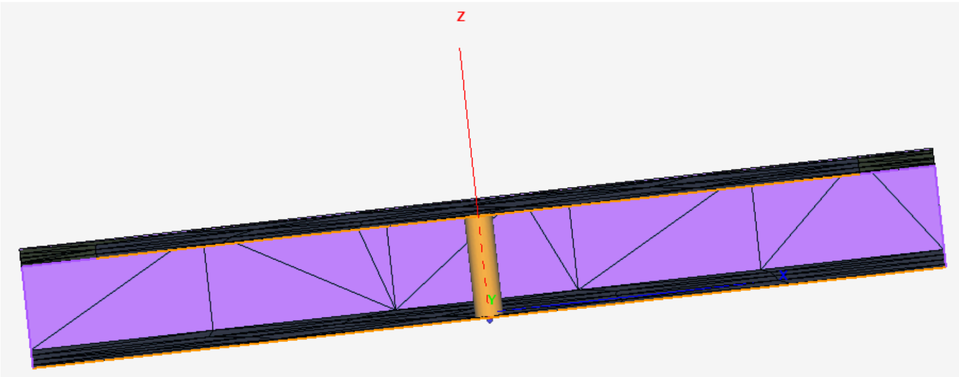


*Figure 81: The cutplane shows a cross-sectional view of the patch substrate. Note the thick feedpin as specified by the **Wire segment radius**.*

c) Disable the cutplane. Click the button next to **XZ-Cut** again.

3.4.16 Setting Local Mesh Sizes for Chamfered Edges

Refine the mesh locally at the chamfered edges.

The mesh can be refined globally but will result in an unnecessary large number of mesh elements. A more efficient approach is to only refine the mesh locally where a finer mesh is required.

> **Note:** Local mesh refinement takes precedence over global mesh settings.

1. In the 3D view, select a chamfered edge.

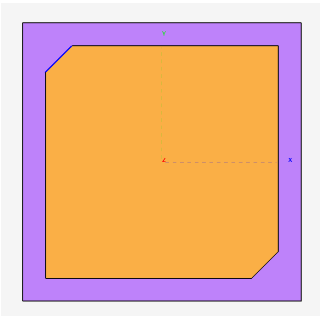


Figure 82: Top view of patch and substrate. The blue edge indicates that it is selected.

2. From the right-click context menu, select **Properties**.
3. On the **Modify Edge** dialog (**Meshing** tab), specify the following:
   a) Select the **Local mesh size** check box.
   b) Set the **Mesh size** to `2`.

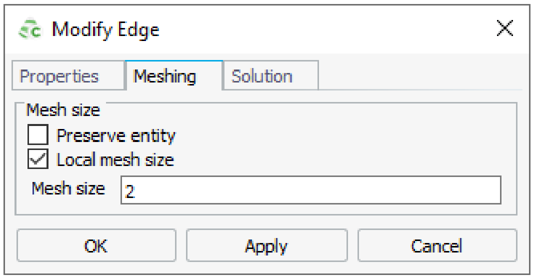


*Figure 83: The **Modify Edge** (**Meshing** tab) dialog.*

4. Repeat Step 1 to Step 3 for the second chamfered edge.

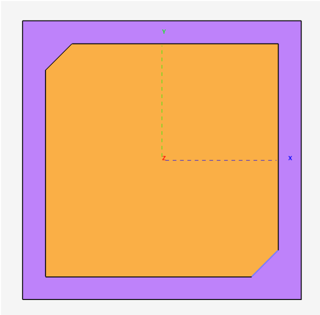


Figure 84: Top view of patch and substrate. The second chamfered edge is selected.

5. Click **OK** to apply the properties and to close the dialog.

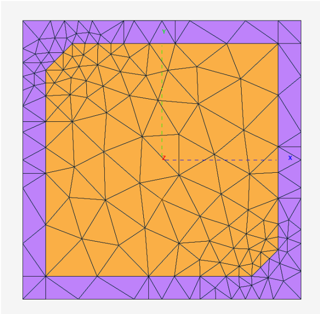


Figure 85: Top view of patch and substrate showing the localised mesh refinement at the chamfered edges.

> **Note:** The icon in the details tree indicate that a local mesh setting is applied.

3.4.17 Adding a Far Field Request

Add a far field request to the model.

1. On the **Request** tab, in the **Solution Requests** group, click the **Far Fields** icon.
2. On the **Request Far Fields** dialog, click **3D pattern**.

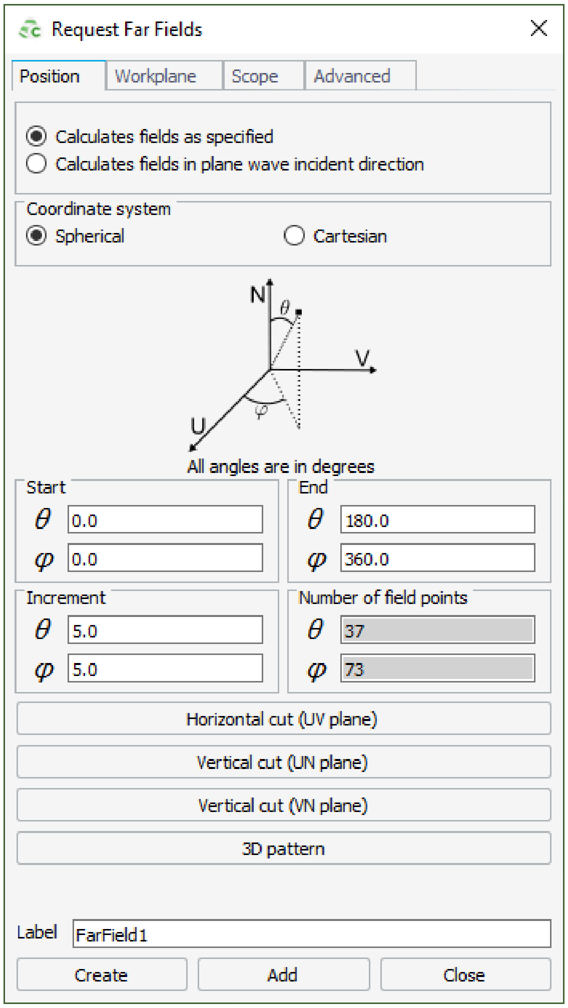


*Figure 86: The **Request Far Fields** dialog.*

3. Click **Create** to create a far field request and to close the dialog.

3.4.18 Deactivating Macro Recording

Deactivate the macro recording of the model, inspect the resulting Feko Lua script and use the script to recreate the model.

1. Deactivate macro recording using one of the following workflows:
    - On the **Home** tab, in the **Scripting** group, click the **Record Macro** icon.
    - Click the icon in the status bar.

Macro recording is deactivated. The **Script Editor** window is displayed containing the Feko Lua script.

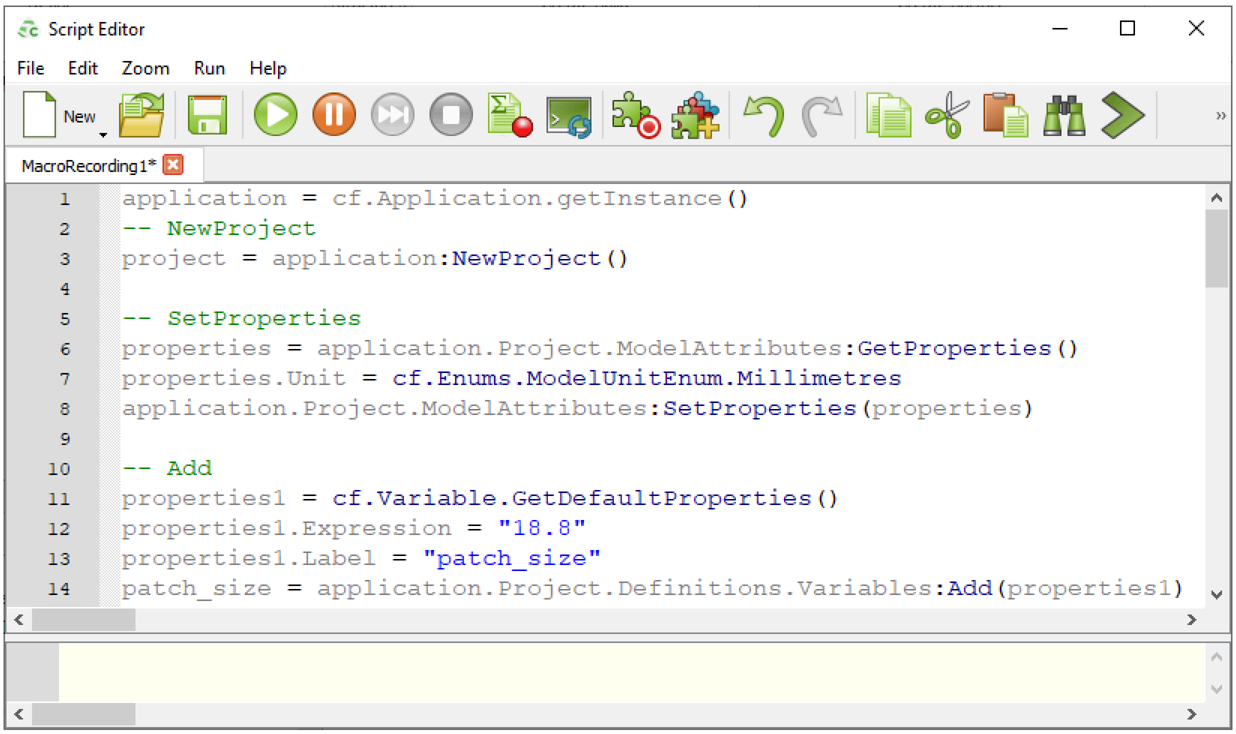


*Figure 87: The **Script Editor** window.*

2. Save the Feko Lua script.
    a) On the **Script Editor** window, save the Feko Lua script by clicking on the icon.
    b) On the **Save As** dialog, browse to a folder and specify a file name.
    c) Click **Save** to save the Feko Lua script and to close the **Save As** dialog.
3. Open a new project and recreate the model using the Feko Lua script.
    a) On the **Home** tab, in the **File** group, click the **New Project** icon.
    b) On the **Script Editor** window, click the icon to run the Feko Lua script.

The model is recreated using the Feko Lua script (see Macro Recording of Example 3).

> **Note:** For more information regarding scripts and the Feko application programming interface (API), see the Feko Scripting and API Reference Guide.

3.4.19 Macro Recording of Example 3

The CADFEKO macro recorded Lua script for Example 3 is given below.

```
application = cf.Application.getInstance()

-- NewProject
project = application:NewProject()

-- SetProperties
properties = application.Project.ModelAttributes:GetProperties()
properties.Unit = cf.Enums.ModelUnitEnum.Millimetres
application.Project.ModelAttributes:SetProperties(properties)

-- Add
properties1 = cf.Variable.GetDefaultProperties()
properties1.Expression = "18.8"
properties1.Label = "patch_size"
patch_size = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "4.3"
properties1.Label = "chamfer_d"
chamfer_d = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "-6.4"
properties1.Label = "feed_pos"
feed_pos = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "45"
properties1.Label = "substrate_w"
substrate_w = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "45"
properties1.Label = "substrate_d"
substrate_d = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "5"
properties1.Label = "substrate_h"
substrate_h = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "5.6"
properties1.Label = "ceramic_epsR"
ceramic_epsR = application.Project.Definitions.Variables:Add(properties1)

-- Add
properties1.Expression = "0.0041"
properties1.Label = "ceramic_tanD"
ceramic_tanD = application.Project.Definitions.Variables:Add(properties1)

-- CreateGroup
group1 = application.Project.Definitions.Variables:CreateGroup()

-- MoveIn
group1:MoveIn({patch_size, chamfer_d})

-- Setting Label
group1.Label = "Patch"

-- CreateGroup
group11 = application.Project.Definitions.Variables:CreateGroup()

-- MoveIn
group11:MoveIn({substrate_w, substrate_d, substrate_h})

-- Setting Label
group11.Label = "Substrate"

-- AddDielectric
properties2 = cf.Dielectric.GetDefaultProperties()
properties2.DielectricModelling.RelativePermittivity = "ceramic_epsR"
```

```
properties2.DielectricModelling.LossTangent = "ceramic_tanD"
properties2.Label = "Ceramic"
ceramic = application.Project.Definitions.Media.Dielectric:AddDielectric(properties2)

-- Setting Colour
ceramic.Colour = "#aa55ff"

-- AddPolygon
properties3 = cf.Polygon.GetDefaultProperties()
properties3.Corners[1].U = "patch_size"
properties3.Corners[1].V = "patch_size"
properties3.Corners[1].N = "substrate_h"
properties3.Corners[2].U = "-patch_size + chamfer_d"
properties3.Corners[2].V = "patch_size"
properties3.Corners[2].N = "substrate_h"
properties3.Corners[3].U = "-patch_size"
properties3.Corners[3].V = "patch_size - chamfer_d"
properties3.Corners[3].N = "substrate_h"
properties3.Corners[4] = {}
properties3.Corners[4].U = "-patch_size"
properties3.Corners[4].V = "-patch_size"
properties3.Corners[4].N = "substrate_h"
properties3.Corners[5] = {}
properties3.Corners[5].U = "patch_size - chamfer_d"
properties3.Corners[5].V = "-patch_size"
properties3.Corners[5].N = "substrate_h"
properties3.Corners[6] = {}
properties3.Corners[6].U = "patch_size"
properties3.Corners[6].V = " -patch_size + chamfer_d"
properties3.Corners[6].N = "substrate_h"
properties3.LocalWorkplane.WorkplaneDefinitionOption
 = cf.Enums.LocalWorkplaneDefinitionEnum.UsePredefinedWorkplane
globalXY = application.Project.Definitions.Workplanes:Item("Global XY")
properties3.LocalWorkplane.ReferencedWorkplane = globalXY
properties3.Label = "patch"
patch = application.Project.Contents.Geometry:AddPolygon(properties3)

-- AddCuboid
properties4 = cf.Cuboid.GetDefaultProperties()
properties4.Origin.U = "-22.5"
properties4.Origin.V = "-22.5"
properties4.Width = "substrate_w"
properties4.Depth = "substrate_d"
properties4.Height = "substrate_h"
properties4.LocalWorkplane.WorkplaneDefinitionOption
 = cf.Enums.LocalWorkplaneDefinitionEnum.UsePredefinedWorkplane
properties4.LocalWorkplane.ReferencedWorkplane = globalXY
properties4.Label = "substrate"
substrate = application.Project.Contents.Geometry:AddCuboid(properties4)

-- SetProperties
properties5 = substrate.Regions:Item("Region1"):GetProperties()
properties5.Medium = ceramic
substrate.Regions:Item("Region1"):SetProperties(properties5)

-- AddLine
properties6 = cf.Line.GetDefaultProperties()
properties6.StartPoint.V = "feed_pos"
properties6.EndPoint.U = "0"
properties6.EndPoint.V = "feed_pos"
properties6.EndPoint.N = "substrate_h"
properties6.LocalWorkplane.WorkplaneDefinitionOption
 = cf.Enums.LocalWorkplaneDefinitionEnum.UsePredefinedWorkplane
properties6.LocalWorkplane.ReferencedWorkplane = globalXY
properties6.Label = "feed_line"
feed_line = application.Project.Contents.Geometry:AddLine(properties6)

-- AddUnion
union1 = application.Project.Contents.Geometry:Union({patch, substrate, feed_line})

-- SetProperties
properties7 = union1.Faces:Item("Face8"):GetProperties()
perfectElectricConductor = application.Project.Definitions.Media.PerfectElectricConductor
properties7.Medium = perfectElectricConductor
union1.Faces:Item("Face8"):SetProperties(properties7)

-- ToggleVisibility
```

```
substrate.Faces:Item("Face2"):ToggleVisibility()

-- ToggleVisibility
substrate.Faces:Item("Face2"):ToggleVisibility()

-- SetProperties
properties8 = union1.Faces:Item("Face6"):GetProperties()
properties8.Medium = perfectElectricConductor
union1.Faces:Item("Face6"):SetProperties(properties8)

-- AddWirePort
properties9 = cf.WirePort.GetDefaultProperties()
edge19 = feed_line.Edges:Item("Edge19")
properties9.Wire = edge19
properties9.Label = "Port1"
port1 = application.Project.Contents.Ports:AddWirePort(properties9)

-- AddVoltageSource
properties10 = cf.VoltageSource.GetDefaultProperties()
properties10.Terminal = port1
properties10.Label = "VoltageSource1"
voltageSource1 =
 application.Project.Contents.SolutionConfigurations.GlobalSources:AddVoltageSource(properties10)

-- SetProperties
properties11 = application.Project.Contents.SolutionConfigurations.GlobalFrequency:GetProperties()
properties11.Start = "1.27e9"
properties11.End = "1.85e9"
properties11.RangeType = cf.Enums.FrequencyRangeTypeEnum.Continuous
application.Project.Contents.SolutionConfigurations.GlobalFrequency:SetProperties(properties11)

-- SetProperties
properties12 = application.Project.Mesher.Settings:GetProperties()
properties12.MeshSizeOption = cf.Enums.MeshSizeOptionEnum.Fine
properties12.WireRadius = "07"
properties12.Advanced.GrowthRate = 30
properties12.Advanced.RefinementFactor = 80
properties12.Advanced.MinElementSize = 80
application.Project.Mesher.Settings:SetProperties(properties12)

-- ToggleVisibility
application.Project.Contents.Cutplanes:Item("XZ-Cut"):ToggleVisibility()

-- ToggleVisibility
application.Project.Contents.Cutplanes:Item("XZ-Cut"):ToggleVisibility()

-- SetProperties
properties13 = union1.Edges:Item("Edge6"):GetProperties()
properties13.LocalMeshSizeEnabled = true
properties13.LocalMeshSize = "2"
union1.Edges:Item("Edge6"):SetProperties(properties13)

-- SetProperties
properties14 = union1.Edges:Item("Edge3"):GetProperties()
properties14.LocalMeshSizeEnabled = true
properties14.LocalMeshSize = "2"
union1.Edges:Item("Edge3"):SetProperties(properties14)

-- Add
properties15 = cf.FarField.GetDefaultProperties()
properties15.Theta.End = "180.0"
properties15.Theta.Increment = "5.0"
properties15.Phi.End = "360.0"
properties15.Phi.Increment = "5.0"
properties15.Label = "FarField1"
properties15.LocalWorkplane.WorkplaneDefinitionOption
 = cf.Enums.LocalWorkplaneDefinitionEnum.UsePredefinedWorkplane
properties15.LocalWorkplane.ReferencedWorkplane = globalXY
farField1 =
 application.Project.Contents.SolutionConfigurations:Item("StandardConfiguration1").FarFields:Add(properties15)

-- SaveAs
application:SaveAs("C:/Users/eh/Desktop/Example2.CADFEKO")
```

3.4.20 Saving the Model

Save the model to a CADFEKO `.cfx` file.

1. Save the model using one of the following workflows:
    - On the **Home** tab, in the **File** group, click the **Save** icon.
    - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Patch.cfx`.
3. Click **Save** to close the dialog.

3.5 Launching the Solver

Launch the Solver to calculate the results. No requests were added to this model since impedance and current information are calculated automatically for all voltage and current sources in the model.

1. Launch the Solver using one of the following workflows:
   - On the **Solve/Run** tab, in the **Run/Launch** group, click the **Feko Solver** icon.
   - On the application launcher toolbar, click the **Feko Solver** icon in the group.
   - Press Alt+4 to use the keyboard shortcut.

If the model contains unsaved changes, the **Save Model** dialog is displayed.

2. Click **Yes** to save the model and to close the **Save Model** dialog.

The Feko Solver is launched and the **Executing runfeko** dialog is displayed. The dialog gives step-by-step feedback as the simulation progresses.

3. Click **Details** to expand the **Executing runfeko** to view the step-by-step feedback.

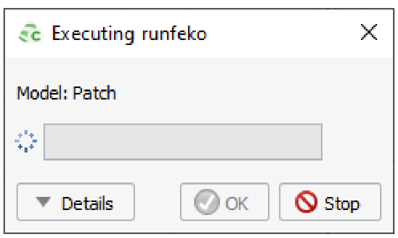


*Figure 88: The **Executing runfeko** dialog.*

3.6 Viewing the Results in POSTFEKO

Display the model as well as the results using the post-processor component, POSTFEKO.

3.6.1 Reviewing POSTFEKO and Launching OPTFEKO

Open POSTFEKO from within CADFEKO.

Use one of the following workflows to launch POSTFEKO:

- On the **Solve/Run** tab, in the **Run/Launch** group, click the **POSTFEKO** icon.
- On the application launcher toolbar, click the **POSTFEKO** icon in the group.
- Press Alt+3 to use the keyboard shortcut.

POSTFEKO opens by default with a single 3D view containing the model geometry.

3.6.2 Viewing the Input Reflection Coefficient

View the input reflection coefficient on a Cartesian graph in dB.

1. On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. On the **Home** tab, in the **Add results** group, click the **Source data** icon. From the drop-down list, select **VoltageSource1**.
3. View the input reflection coefficient in dB versus frequency.
   a) On the result palette, in the **Traces** panel, select **VoltageSource1**.
   b) On the **Quantity** panel, confirm that **Reflection coefficient** is selected (default option).
   c) On the **Quantity** panel, select the **dB** check box.

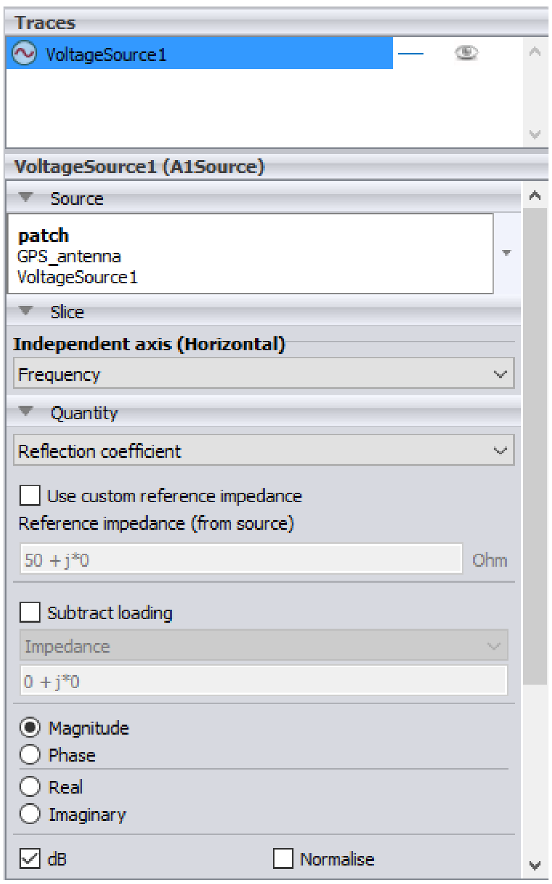


*Figure 89: The result palette containing the **Traces**, **Source**, **Slice** and **Quantity** panels (listed from top to bottom).*

4. Change the legend position to bottom-right.
   a) On the **Display** tab, in the **Legend** group, click the **Position** icon. From the drop-down list select **Overlay bottom right**.
5. Remove the graph footer.
   a) On the **Display** tab, in the **Display** group, click the **Chart text** icon.

b) In the **Graph footer** field, clear the **Auto** check box and delete the text.

c) Click **OK** to apply the text changes and to close the dialog.

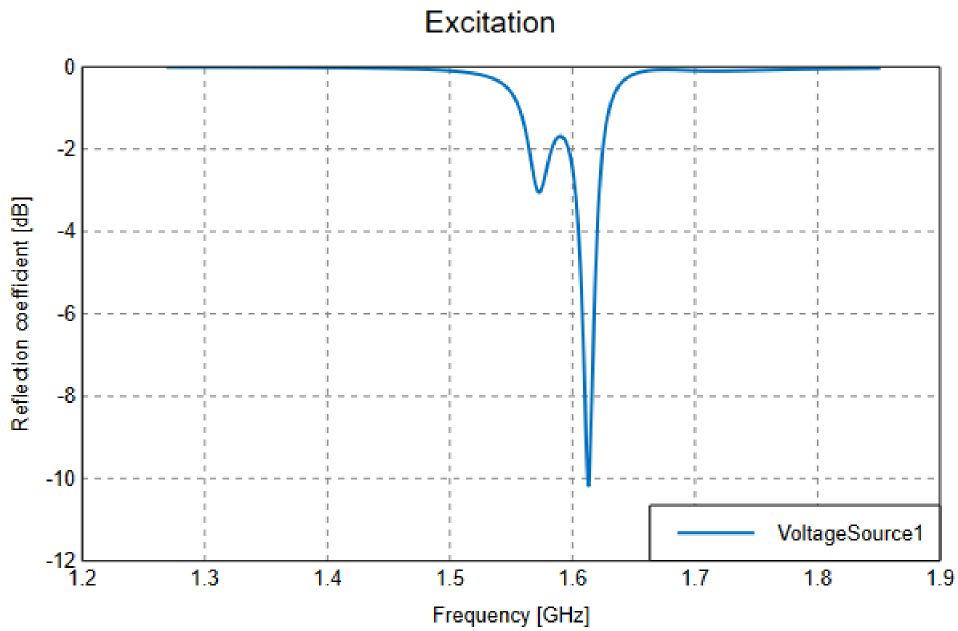


Figure 90: The input reflection coefficient in dB versus frequency.

3.6.3 Viewing the Circular Components of the Far Field

1. On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. On the **Home** tab, in the **Add results** group, click the **Far field** icon. From the drop-down list, select **FarField1**.
3. Make a copy of the trace, **FarField1**.
   a) On the result palette, in the **Traces** panel, select **FarField1**.
   b) Duplicate the trace, **FarField1**, using one of the following workflows:
      - On the **Cartesian** context tab, on the **Trace** tab, in the **Manage** group, click the **Duplicate trace** icon.
      - Press Ctrl+K to use the keyboard shortcut.

   A trace with label **FarField1_1** is created.
4. Rename the trace, **FarField1_1**.
   a) On the result palette, in the **Traces** panel, select **FarField1_1**.
   b) Press F2 to use the keyboard shortcut and rename the trace to **FarField2**.
5. View the left-hand circular component of the far field in dB versus frequency.
   a) In the **Traces** panel, select **FarField1**.
   b) On the result palette, in the **Quantity** panel, click **LHC**.
   c) On the result palette, in the **Quantity** panel, select the **dB** check box.
6. View the right-hand circular component of the far field in dB versus frequency.
   a) In the **Traces** panel, select the duplicate trace, **FarField2**.
   b) On the result palette, in the **Quantity** panel, click **RHC**.
   c) On the result palette, in the **Quantity** panel, select the **dB** check box.
7. [Optional] Repeat Step 4 and Step 5 of Viewing the Input Reflection Coefficient to change the legend position and remove the graph footer.

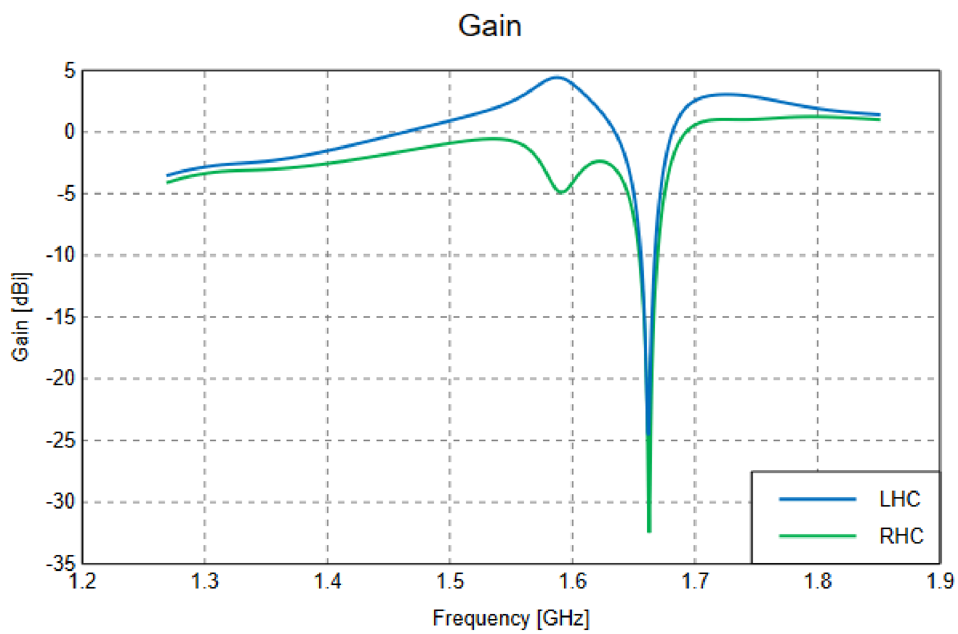


Figure 91: The left-hand circular and right-hand circular components of the far field.

3.7 Final Remarks

This example showed the construction, configuration and solution of a left-handed circular polarised GPS patch antenna on a finite substrate.

The input reflection coefficient and circular components of the far field were calculated and displayed.

GPS Patch on Quadcopter

4

The example considers the antenna placement of a GPS patch antenna on a quadcopter.

This chapter covers the following:

4.1 Example Overview

Calculate the input reflection coefficient and circular components of a left-handed circular polarised GPS patch antenna on a finite substrate close to 1.57 GHz placed on a quadcopter. Compare the results with that of Example 3.

4.2 Topics Discussed in Example

The topics discussed in this example are:

- CADFEKO
    - Specify the model unit.
    - Add a component from the component library.
    - Import a model from a `.cfx` file.
    - Create a workplane and perform transformations on the workplane (rotate).
    - Use the Align tool for antenna placement.
    - Run the .
    - Show/hide the simulation mesh in the 3D view.
    - Show/hide a part in the 3D view.
- POSTFEKO
    - View the Lua script to set up the graphs (similar to Example 2) for the following:
        - View the input reflection coefficient on a Cartesian graph.
        - View the left-hand and right-hand circular components of the far field on a Cartesian graph.
        - View an example of a Lua script to configure graphs.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[20]:
>
> `GS 4: GPS Patch on a Drone`

20. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

4.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 40 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[21] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

21. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

4.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

4.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

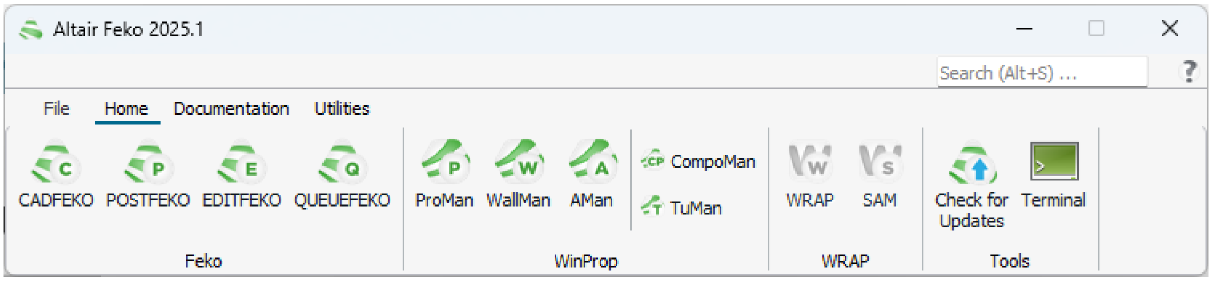


Figure 92: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[22] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

4.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the "initfeko" script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

22. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

4.4.3 Setting the Model Unit

Set the model unit to millimeters.

The default unit length in CADFEKO is metres. Since the structure that you will build is small, the model unit is set to millimetres. All dimensions entered will be in the new model unit.

1. Set the model unit to millimetres using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Model unit** icon.
   - On the status bar, click Unit: m.
2. On the **Model Unit** dialog, select **Millimetres (mm)**.
3. Click **OK** to change the model unit to millimetres and to close the dialog.

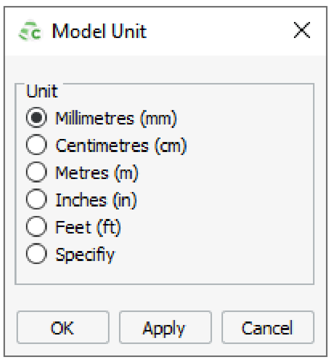


*Figure 93: The **Model Unit** dialog.*

4.4.4 Adding a Quadcopter from the Component Library

Select a simplified quadcopter model from the Component library and add it to the project.

1. On the **Home** tab, in the **File** group, click the **Component Library** icon.
2. On the **Component Library** dialog, in the **Filter** field, enter the text `quadcopter`.

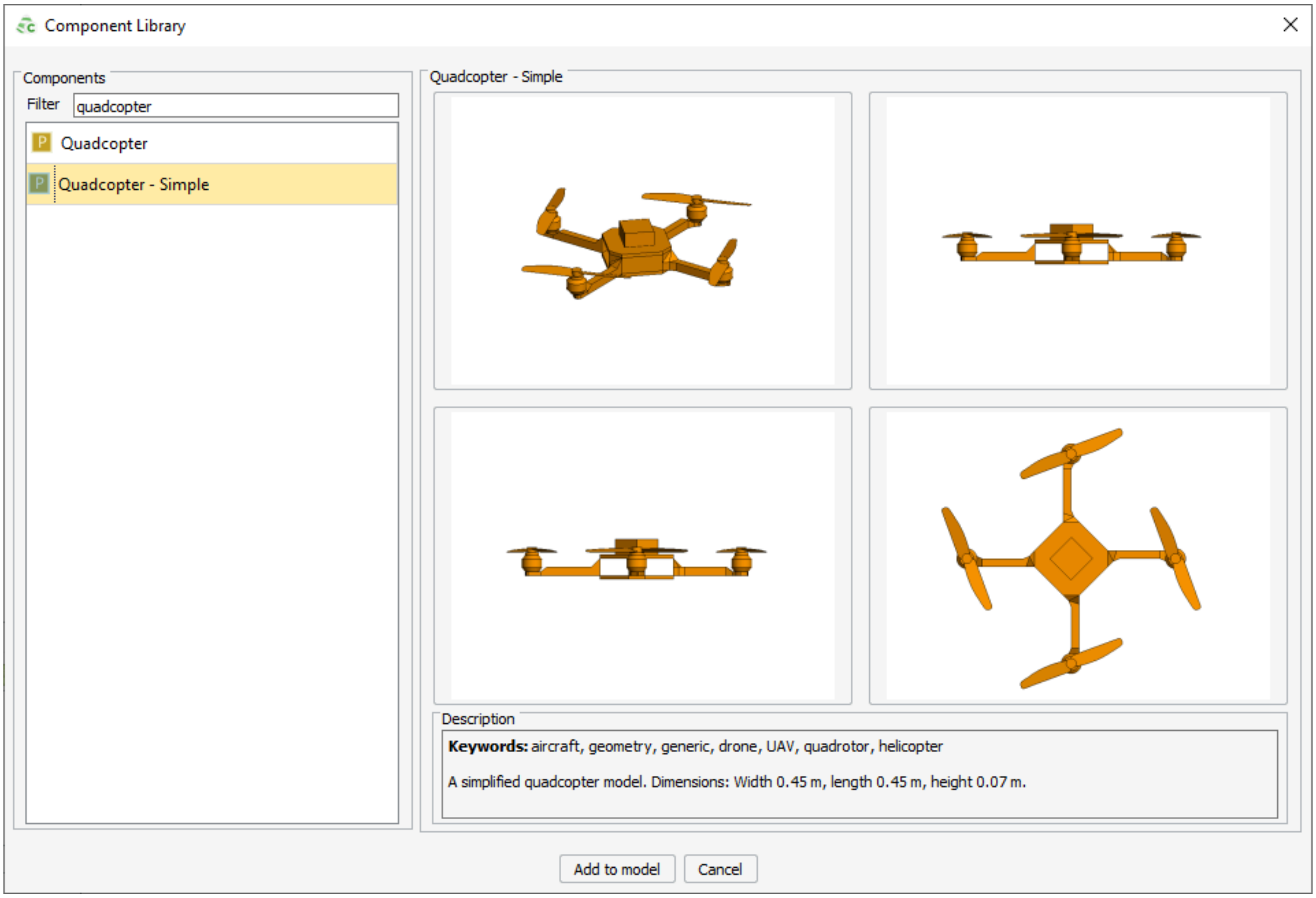


*Figure 94: The **Component Library** dialog.*

3. From the filtered results, click **Quadcopter - Simple**.
4. Click **Add to model** to add the quadcopter and to close the dialog.
5. On the **Align** dialog, under **Destination workplane**, in the **Origin** field, **Z** field, enter `-100`.

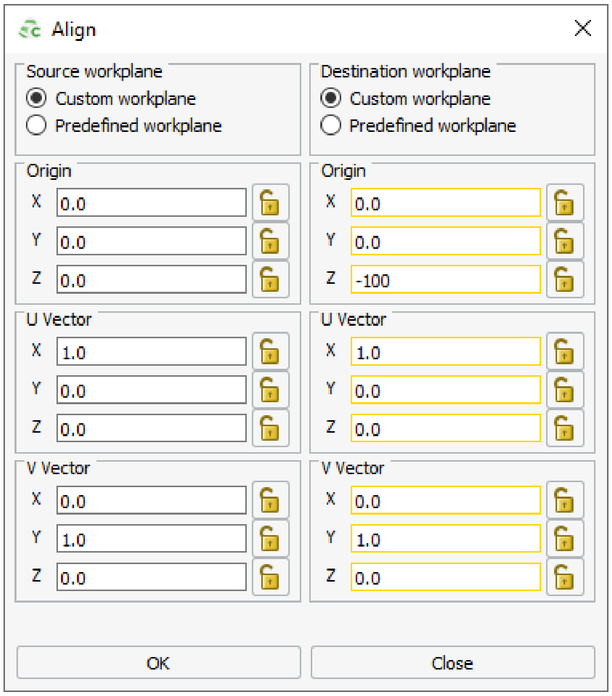


*Figure 95: The **Align** dialog.*

> **Note:** The offset separates the patch from the quadcopter on import.

6. Click **OK** to place the quadcopter and to close the dialog.

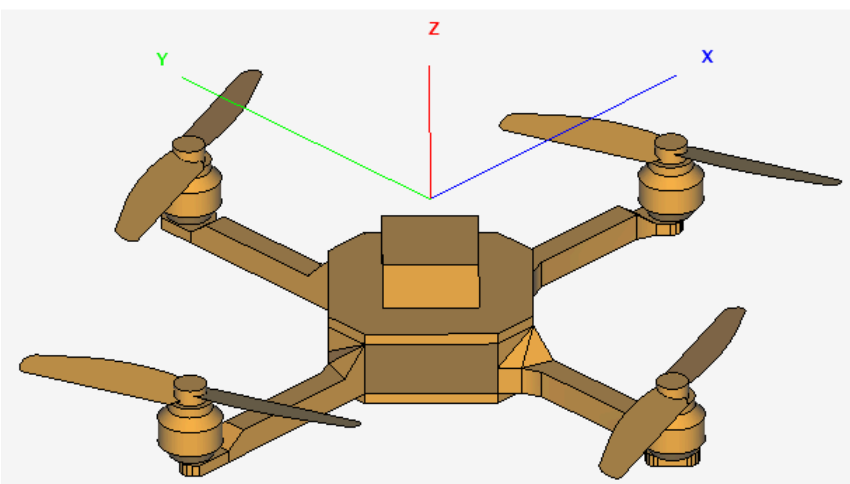


Figure 96: The simple quadcopter model from the component library.

4.4.5 Importing the GPS Patch

Import the GPS patch antenna (`.cfx` file) created in Example 3.

1. On the **Home** tab, in the **File** group, click the **Import** icon. From the drop-down list select the **CADFEKO Model (*.cfx)** icon.
2. On the **Import CADFEKO Model** dialog, browse to the location of where you saved Example 3[23] and click **OK**.

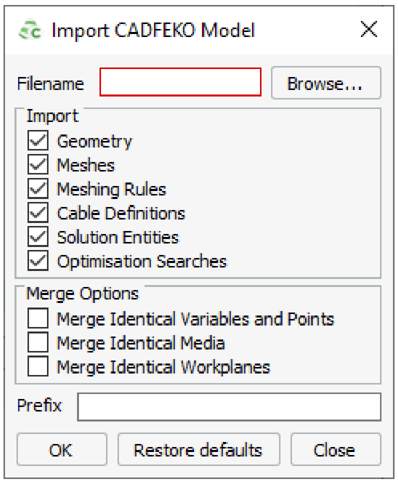


*Figure 97: The **Import CADFEKO Model** dialog.*

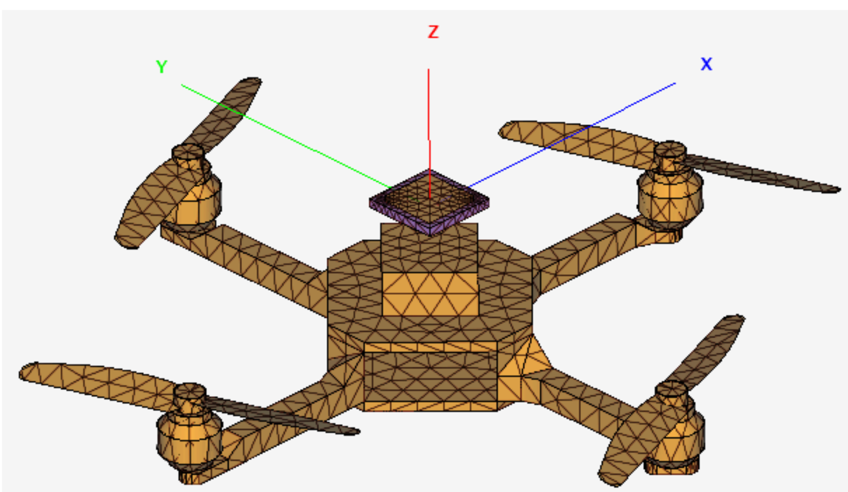


Figure 98: The imported GPS patch antenna is located above the quadcopter.

3. In the model tree, under **Geometry**, select **Union1**, press F2 and rename the part to *Patch*.

23. Alternatively, open *GS 3: GPS Patch Antenna* in the application macro library and save the model.

4.4.6 Setting the Simulation Frequency

Specify the frequency range of interest. For this example continuous frequency sampling is used where Feko automatically determines the frequency sampling for optimal interpolation.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.
2. On the **Solution Frequency** dialog, from the drop-down list, select **Continuous (interpolated) range**.
3. In the **Start frequency (Hz)** field, enter `1.27e9`.
4. In the **End frequency (Hz)**, enter `1.85e9`.

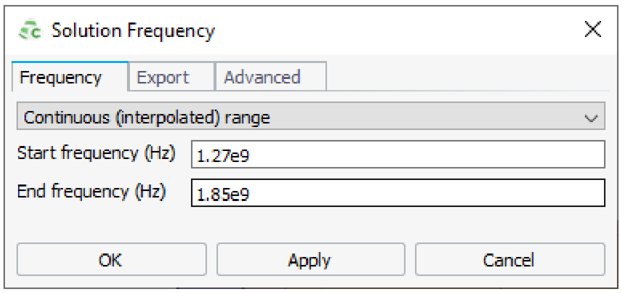


*Figure 99: The **Solution Frequency** dialog.*

5. Click **OK** to specify the frequency and to close the dialog.

4.4.7 Hiding Parts of the Model

Entities in the model tree and details tree can be hidden by clicking on the entity's icon.

1. In the model tree, click the icon next to **Patch**.
   Note that **Patch** is greyed out in the model tree and hidden in the 3D view.

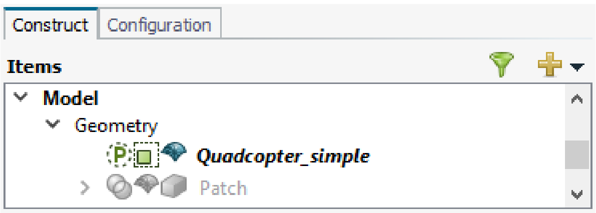


Figure 100: Patch is greyed out to indicate that the part is hidden in the 3D view.

2. Repeat Step 1 again to make the part visible again in the 3D view.

> **Tip:** Hide a wire/edge, face or region in the 3D view by clicking its icon in the details tree.

4.4.8 Hiding the Simulation Mesh

Make the simulation mesh hidden in the 3D view focus on the geometry.

Hide the simulation mesh[24] using one of the following workflows:

- On the status bar, click the **Overlay** icon.
- On the **3D View** context tab, on the **Display Options** tab, in the **Display Mode** group, click the **Overlay** icon.

24. The simulation mesh refers to the final mesh used by the Solver. CAD always has to be meshed.

4.4.9 Placing the Patch on the Quadcopter

To assist in placing the patch antenna, two workplanes are defined and used in the **Align** tool.

Rotating the Quadcopter

Rotate the quadcopter by 45°.

1. On the model tree, click *Quadcopter_simple1* and from the right-click context menu, click **Transforms** > **Rotate**.
2. On the **Rotate** dialog, under **Rotation angle**, in the **Angle [degrees]** field, enter `45°`.

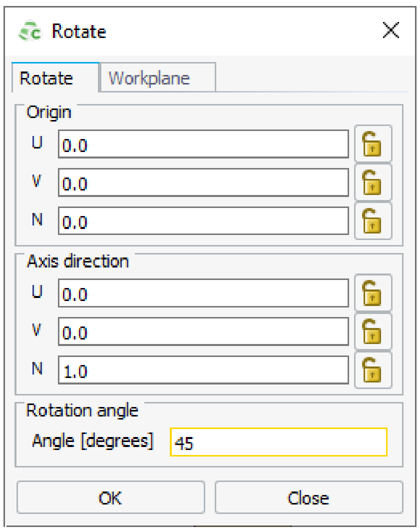


*Figure 101: The **Rotate** dialog.*

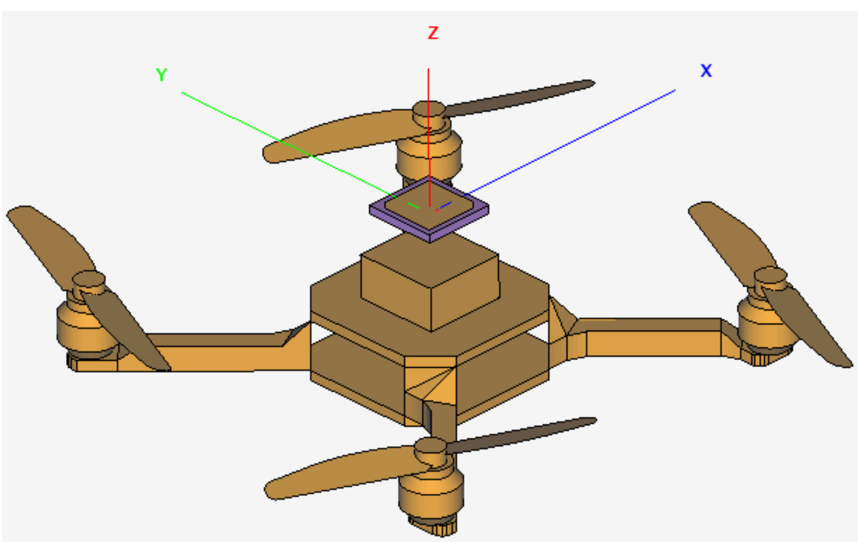


Figure 102: The quadcopter was rotated by 45°.

Defining a Workplane on the Quadcopter

Define a workplane on the top face of the quadcopter.

1. On the **Construct** tab, in the **Define** group, click the **Add Workplane** icon.

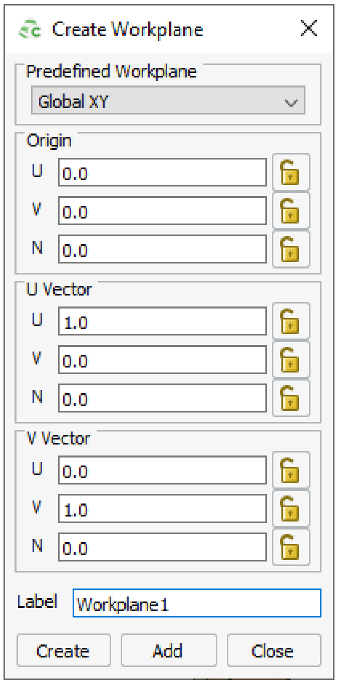
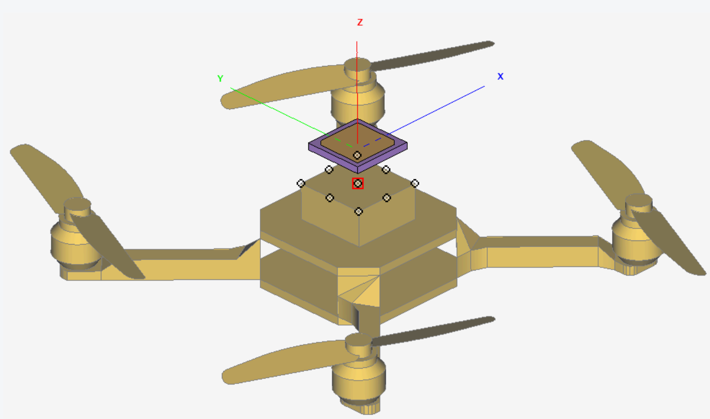


Figure 103: The ***Create Workplane*** *dialog.*

2. Press Ctrl+Shift while moving the mouse cursor over the top face centre of the quadcopter.

> **Note:** The circles with a black outline indicate special snapping points. The red outline indicates the position of the mouse cursor.
>
> Use snapping points to snap the workplane to an object. Although only special snapping points are indicated, you can snap to any point in the 3D view.
>
> 
>
> *Figure 104: Special snapping points are indicated by circles with a black outline.*

3. Press Ctrl+Shift+left click to snap the workplane to the top face centre of the quadcopter.
4. In the **Label** field, change the label to `Workplane_quadcopter`.
5. Click **Create** to create the workplane and to close the dialog.

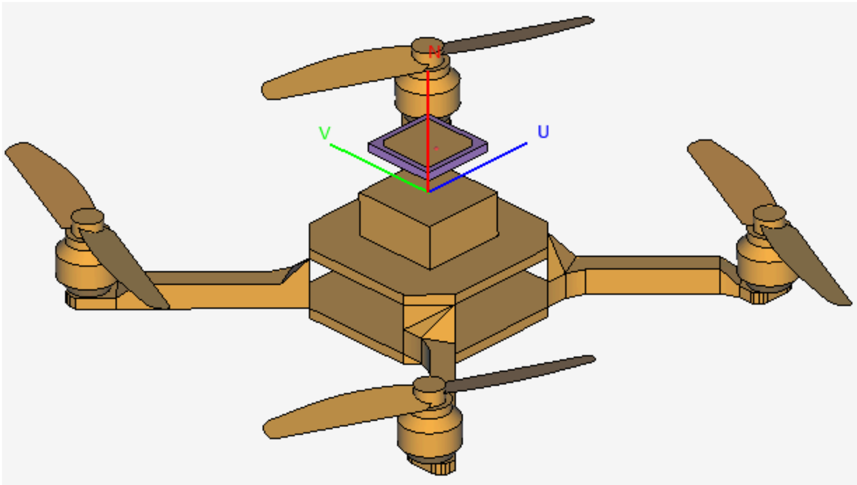


Figure 105: Workplane has snapped to the top centre of the quadcopter.

Defining a Workplane on the Patch

Define a workplane on the top face of the patch.

1. On the **Construct** tab, in the **Define** group, click the **Add Workplane** icon.
2. Press Ctrl+Shift+left click to snap the workplane to the top face centre of the patch.
3. Click **Create** to create *Workplane1* and to close the dialog.

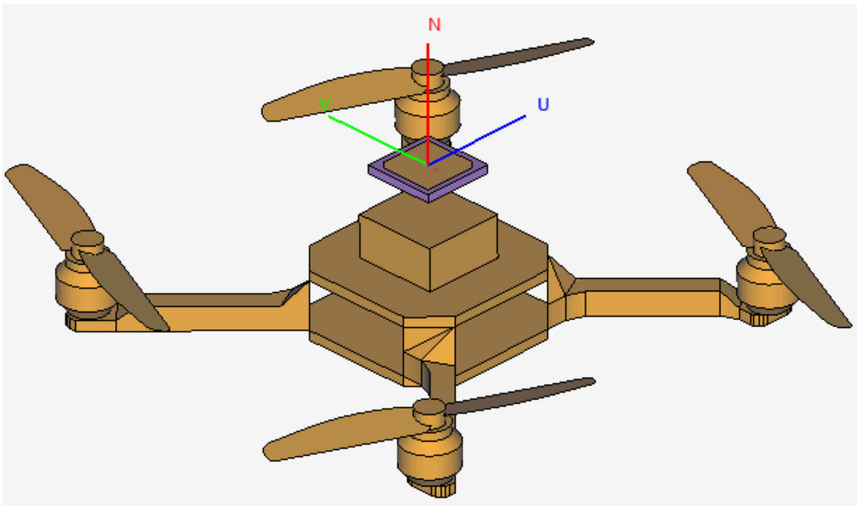


Figure 106: Workplane has snapped to the top centre of the quadcopter.

4. In the model tree, under **Workplanes**, select **Workplane1**, press F2 and rename the workplane to `Workplane_patch`.

Tip: Transforms can be applied to workplanes. In the model tree, select the workplane and from the right-click context menu, click **Transforms**.

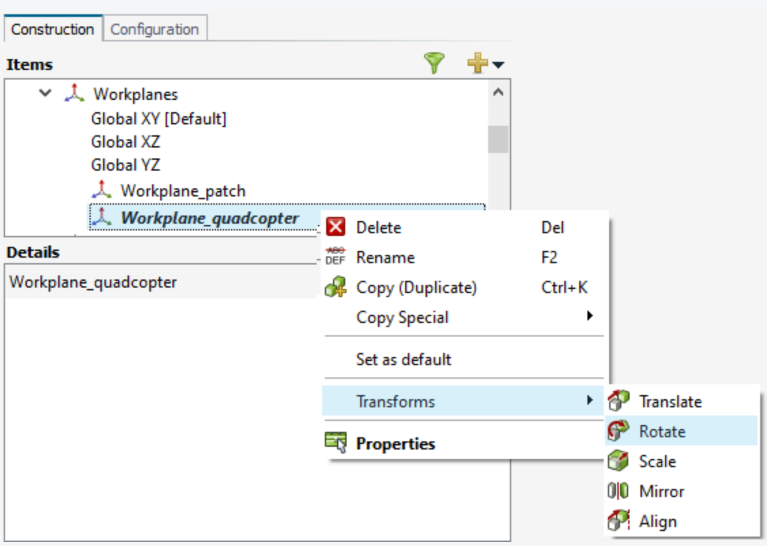


Figure 107: A workplane can be rotated by using the right-click context menu option.

Aligning the Patch and Quadcopter

Align the patch to the quadcopter.

1. In the model tree, select **Quadcopter_simple1**.
2. On the **Transform** tab, in the **Transform** group, click the **Align** icon.
3. On the **Align** dialog, under **Source workplane,** select **Predefined workplane**. From the drop-down list, select **Workplane_quadcopter**.

   > **Tip:** If *Workplane_quadcopter* is unavailable in the drop-down list, repeat Step 1.

4. On the **Align** dialog, under **Destination workplane** select **Predefined workplane**. From the drop-down list, select **Workplane_patch**.

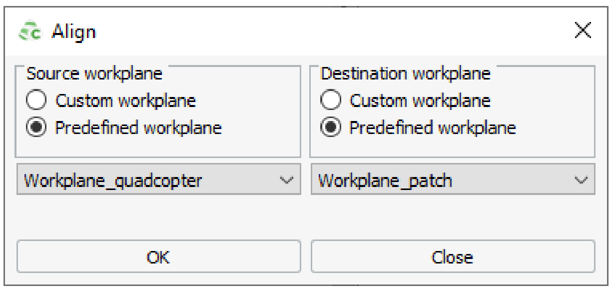
   

   *Figure 108: The **Align** dialog.*

5. Click **OK** to align the quadcopter to the patch and to close the dialog.

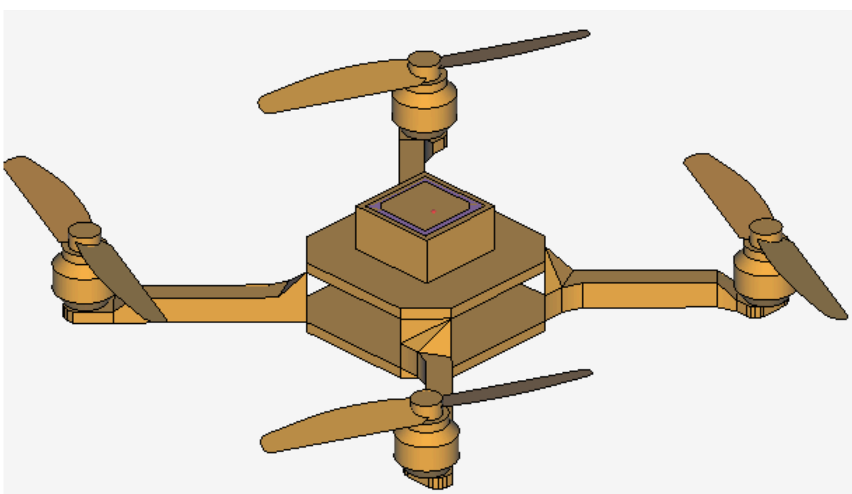
   

   Figure 109: The GPS patch antenna placed on the quadcopter.

Unioning the Model for Mesh Connectivity

Create a single geometry part from the patch antenna and quadcopter to ensure mesh connectivity when the model is meshed.

1. In the model tree, select **Patch** and **Quadcopter_simple1**.
2. Union the GPS patch antenna and the quadcopter using one of the following workflows:
   - On the **Construct** tab, in the **Modify** group, click the **Union** icon.
   - Press U to use the keyboard shortcut.

> **Note:** The union operation is used to define connectivity between parts.
>
> Parts that touch, but are not unioned, are not considered to be physically connected and will result in an incorrect mesh.

3. Show the simulation mesh again.
   a) On the status bar, click the **Overlay** icon.

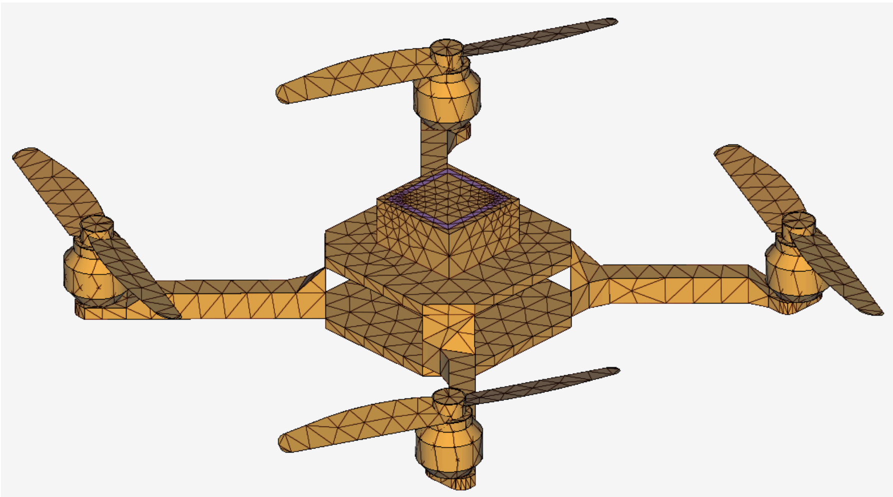


Figure 110: The patch antenna unioned with the quadcopter.

4.4.10 Saving the Model

Save the model to a CADFEKO `.cfx` file.

1. Save the model using one of the following workflows:
   - On the **Home** tab, in the **File** group, click the **Save** icon.
   - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Patch_on_Drone.cfx`.
3. Click **Save** to close the dialog.

4.5 Launching the Solver

Launch the Solver to calculate the results. No requests were added to this model since impedance and current information are calculated automatically for all voltage and current sources in the model.

1. Launch the Solver using one of the following workflows:
   - On the **Solve/Run** tab, in the **Run/Launch** group, click the **Feko Solver** icon.
   - On the application launcher toolbar, click the **Feko Solver** icon in the group.
   - Press Alt+4 to use the keyboard shortcut.

If the model contains unsaved changes, the **Save Model** dialog is displayed.

2. Click **Yes** to save the model and to close the **Save Model** dialog.

The Feko Solver is launched and the **Executing runfeko** dialog is displayed. The dialog gives step-by-step feedback as the simulation progresses.

3. Click **Details** to expand the **Executing runfeko** to view the step-by-step feedback.

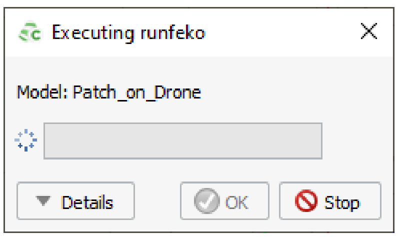


*Figure 111: The **Executing runfeko** dialog.*

4.6 Viewing the Results in POSTFEKO

Display the model as well as the results using the post-processor component, POSTFEKO.

4.6.1 Using a Lua Script to Configure Graphs

View the Lua script to set up the graphs for the input reflection coefficient and circular components of the far field.

The step-by-step instructions to create this script is beyond the scope of the Feko Getting Started Guide, but it is recommended to view the script and to compare it with setting up Graph 1 and Graph 2 from Example 3.

```
app = pf.GetApplication()
-- Graph 1
graph1 = app.CartesianGraphs:Add()
excitationTrace = graph1.Traces:Add(app.Models[1].Configurations[1].Excitations[1])
excitationTrace.Quantity.ValuesScaledToDB = true
graph1.Footer.Text = ""
graph1.Legend.Position = pf.Enums.GraphLegendPositionEnum.OverlayBottomRight

-- Graph 2
graph2 = app.CartesianGraphs:Add()
farFieldTraceLHC = graph2.Traces:Add(app.Models[1].Configurations[1].FarFields[1])
farFieldTraceLHC.Quantity.Type = pf.Enums.FarFieldQuantityTypeEnum.Gain
farFieldTraceLHC.Quantity.Component = pf.Enums.FarFieldQuantityComponentEnum.LHC
farFieldTraceLHC.Quantity.ValuesScaledToDB = true

farFieldTraceRHC = farFieldTraceLHC:Duplicate()
farFieldTraceRHC.Quantity.Type = pf.Enums.FarFieldQuantityTypeEnum.Gain
farFieldTraceRHC.Quantity.Component = pf.Enums.FarFieldQuantityComponentEnum.RHC
farFieldTraceRHC.Label = "FarField2"

farFieldTraceRHC.Quantity.ValuesScaledToDB = true
graph2.Footer.Text = ""
graph2.Legend.Position = pf.Enums.GraphLegendPositionEnum.OverlayBottomRight
```

Note: For more information regarding scripts and the Feko application programming interface (API), see the Feko Scripting and API Reference Guide.

4.7 Final Remarks

This examples showed how to place a GPS patch antenna on a quadcopter that was obtained from the component library.

The input reflection coefficient and circular components of the far field were calculated and displayed.

EMC Coupling 5

The example considers the coupling between a typical monopole antenna and a loaded transmission line above a ground plane.

This chapter covers the following:

5.1 Example Overview

Consider the coupling between a monopole antenna and a loaded transmission line above a ground plane.

Create the monopole antenna using a line and the loaded transmission line using a polyline. The ground plane is modelled using an infinite ground plane.

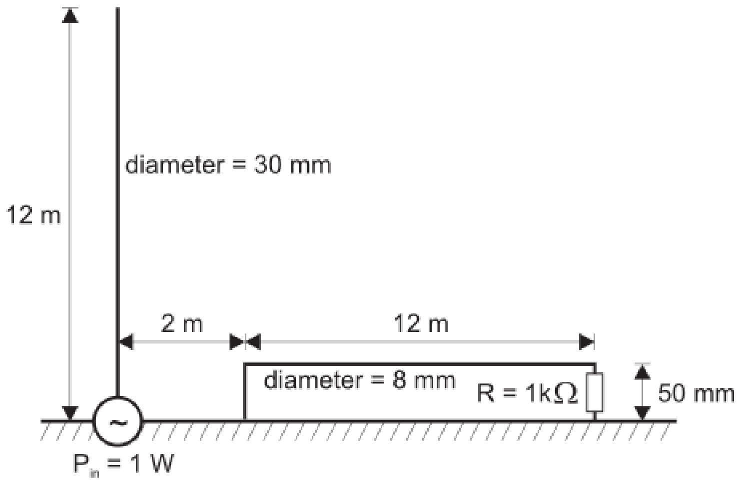


Figure 112: The monopole antenna and the loaded transmission line above an infinite ground plane.

5.2 Topics Discussed in Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- CADFEKO
    - Create a monopole using a line. Specify the local wire radius for the monopole.
    - Create the transmission line using a polyline.
    - Define a ground plane using an infinite ground plane.
    - Add a port and voltage source to the monopole.
    - Specify the radiated power of the model.
    - Add a port and complex load to the transmission line.
    - Set the solution frequency. Use adaptive frequency sampling to obtain continuous data.
    - Mesh the model.
    - Run CEM validate to ensure the model is electromagnetically validated.
    - Run the Solver.
- POSTFEKO
    - View the simulated input impedance and currents on a graph.
    - Change the line colour, marker style, marker colour of a trace on the graph.
    - Add shapes (line, arrow, rectangle or circle) to highlight certain areas on the graph.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[25]:
>
> `GS 5: EMC Coupling`

25. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

5.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 30 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[26] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

26. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

5.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

5.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

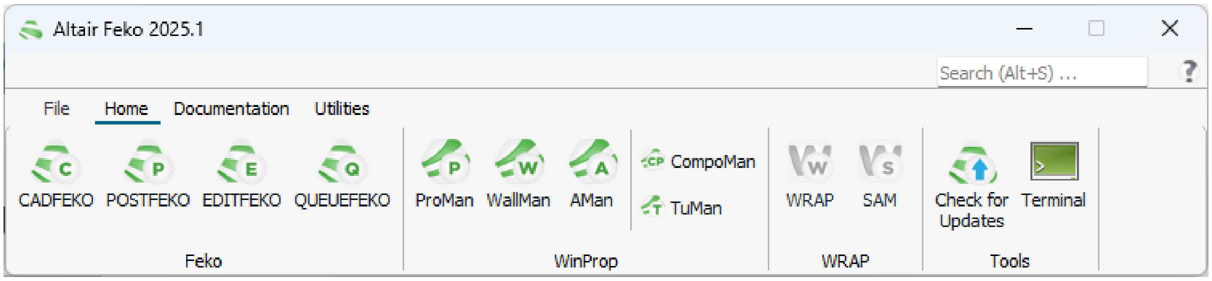


Figure 113: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[27] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

5.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the "initfeko" script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

27. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

5.4.3 Creating a Monopole

Create a monopole antenna as a single line element with a local wire radius. Zoom to extents and hide the main axes to view the full-length monopole in the 3D view.

Create the monopole antenna. The length of the monopole is 12 m along the Z axis.

1. Create a line.
   a) On the **Construct** tab, in the **Create Curve** group, click the **Line** icon.
   b) On the **Create Line** dialog, enter the start point and end point for the line.
      - **Start point**: (0, 0, 0)
      - **End point**: (0, 0, 12)

   > **Note:** Default values are used on geometry creation dialogs to allow a preview in the 3D view. You may change the values as required.

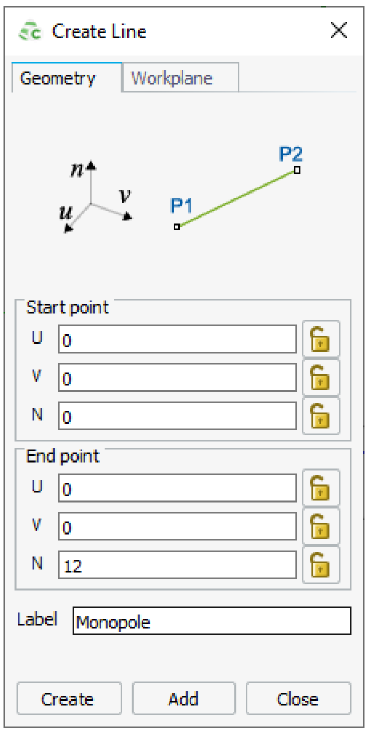


*Figure 114: The **Create Line** dialog.*

2. Set the label to `Monopole`.
3. Click **Create** to create the line and close the dialog.

To view the full-length monopole in the 3D window, zoom the monopole to the window extent.

4. Zoom to extents of the 3D view using one of the following workflows:
   - On the **View** tab, in the **Zoom** group, click the **Zoom to extents** icon.
   - Press F5 to use the keyboard shortcut.

Disable the main axes to view the monopole without the Z axis obstructing it.

5. On the **3D View** context tab, on the **Display Options** tab, in the **Axes** group, click the **Main Axes** icon.
6. Repeat Step 5 to enable the main axes display.

5.4.4 Creating a Transmission Line

Create a transmission line using a polyline curve with four corners. The length of the polyline is 12 m along the Y axis, placed 50 mm above ground.

1. On the **Construct** tab, in the **Create Curve** group, click the **Polyline** icon.

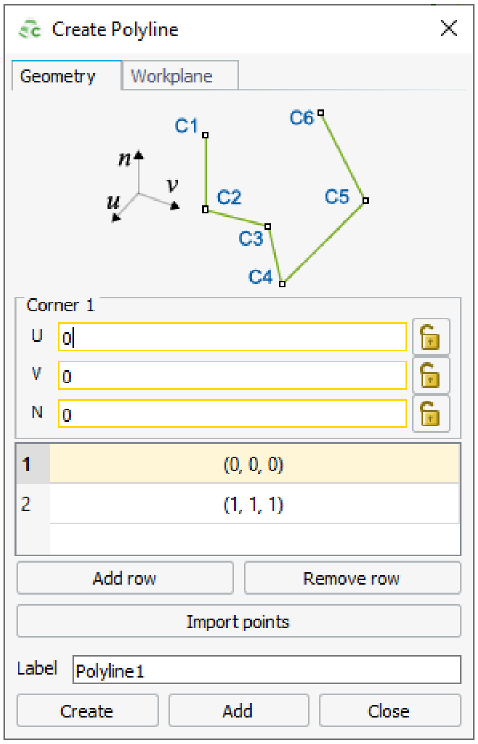


Figure 115: The ***Create Polyline*** *dialog.*

2. Under **Corner 1**, add the following coordinates:
   - Corner 1: (0, 2 0)
3. In the table, click on the second row to make **Corner 2** active. Under **Corner 2**, add the following coordinates:
   - Corner 2: (0, 2, 0.05)
4. Click on **Add row**. Under **Corner 3**, add the following coordinates:
   - Corner 3: (0, 14, 0.05)
5. Click on **Add row**. Under **Corner 4**, add the following coordinates:
   - Corner 4: (0, 14, 0)
6. Set the label to `Transmission_line`.
7. Click **Create** to create the polyline and to close the dialog.
8. Zoom to extents of the 3D view using one of the following workflows:
   - On the **View** tab, in the **Zoom** group, click the **Zoom to extents** icon.
   - Press F5 to use the keyboard shortcut.

5.4.5 Defining an Infinite Ground Plane

Define a perfectly conducting (PEC) infinite ground plane. An infinite ground plane is an efficient method to model a large ground plane compared to a discretised, finite sized ground plane.

Define the infinite ground plane.

a) On the **Construct** tab, in the **Structures** group, click the **Planes/Arrays** icon. From the drop-down list, select **Plane / Ground**.

b) On the **Plane / Ground** dialog, from the **Definition method** drop-down list, select **Perfect electric (PEC) ground plane at Z=0**.

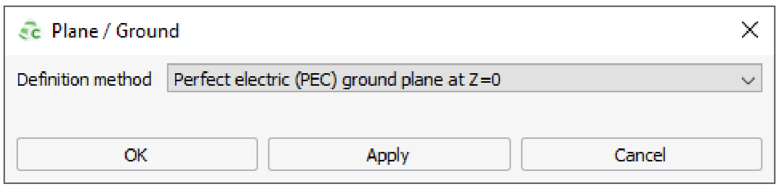


*Figure 116: The **Plane / Ground** dialog.*

c) Click **OK** to create the infinite plane and to close the dialog.

5.4.6 Ports, Sources and Loads in CADFEKO

Voltage sources and discrete loads are applied to ports and not directly to the model geometry or mesh. A port must be defined before a source or load can be added.

Adding a Wire Port to the Monopole

Define a wire port on the monopole. A voltage source will be added to this port.

1. Select the monopole using one of the following workflows:
    - Click on the monopole in the 3D view.
    - In the model tree, select **Monopole**. In the details tree, select **Wire1**.

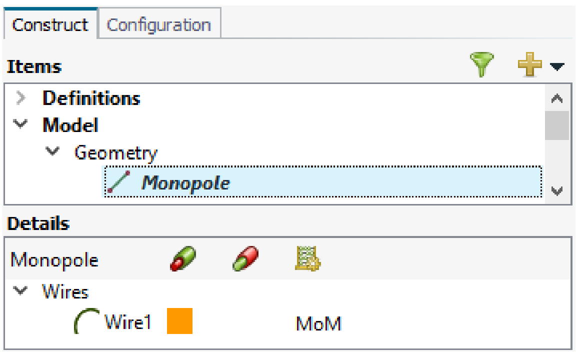


*Figure 117: **Wire1** in the details tree is the wire element associated with **Line1** in the model tree.*

2. Define the wire port on the selected wire (monopole) using one of the following workflows:
    - On the **Source/Load** tab, in the **Ports** group, click the **Wire Port** icon.
    - On the details tree, a right-click context menu is available on the edge. Click **Create Port** > **Wire Port**.

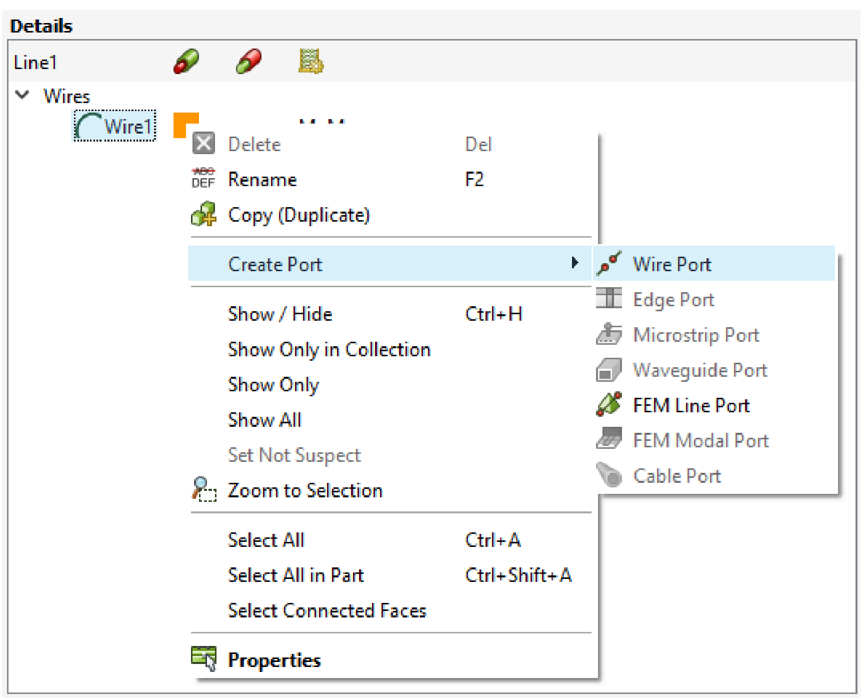


Figure 118: The right-click context menu options for a wire in the details tree.

3. Use the default port settings.

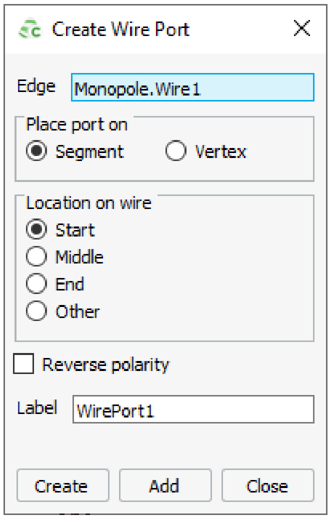


*Figure 119: The **Create Wire Port** dialog.*

4. Click **Create** to create the port and close the dialog.

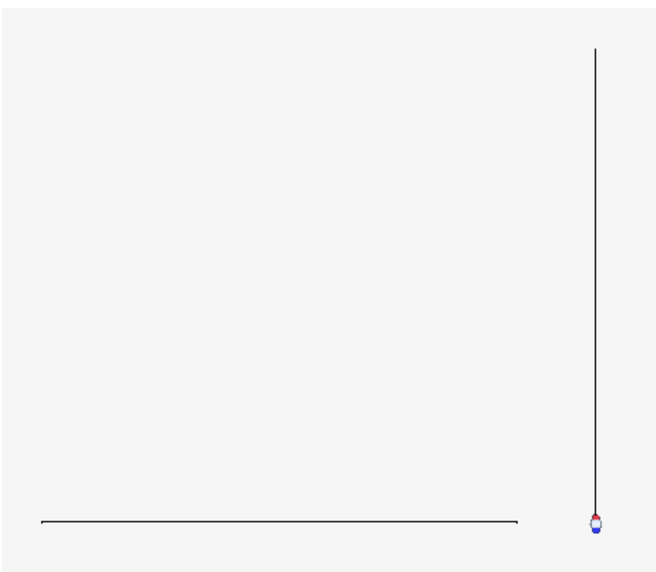


Figure 120: Front view of the monopole and its wire port and the transmission line. The port is indicated by a silver sphere.

Adding a Wire Port to the Transmission Line

Define a wire port for the transmission line.

1. Select the short, vertical wire in the 3D view located farthest away from the monopole.
   The wire element associated with the selected wire is highlighted in the details tree.
2. On the details tree, a right-click context menu is available on the edge. Click **Create Port** > **Wire Port** (see Figure 2).
3. View the port preview in the 3D view to ensure the correct edge is selected.
4. Use the default settings for the port.

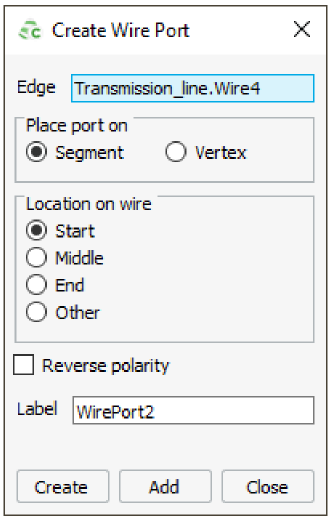


*Figure 121: The **Create Wire Port** dialog.*

5. Create **Create** to create the port and close the dialog.

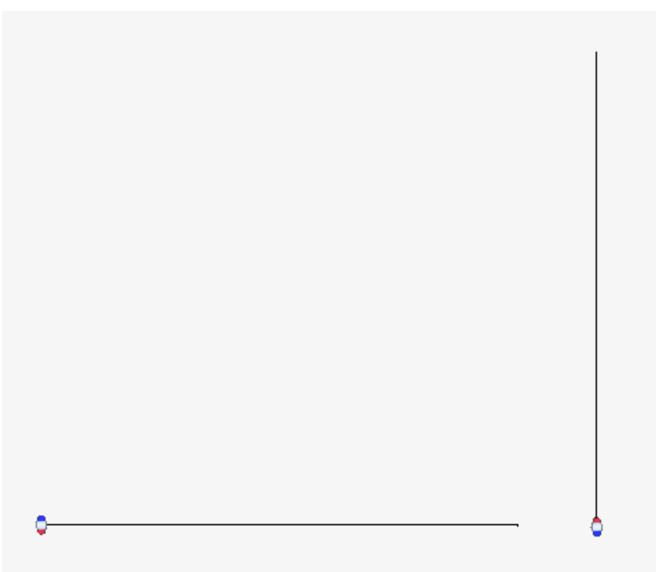


Figure 122: Front view of the monopole and transmission line together with their wire ports. The ports are indicated by silver spheres.

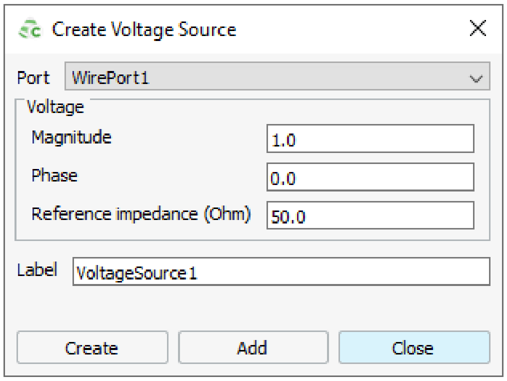
Adding a Voltage Source

Add a voltage source to the port of the monopole.

1. On the **Source/Load** tab, in the **Sources on Ports** group, click the **Voltage Source** icon.

The radiated power must be 1 Watt for this example. Since the input impedance for the monopole is not known, the voltage can not be changed to scale the radiated power.

2. Ensure **WirePort1** is selected in the drop-down list and use the default voltage settings.



3. Click **Create** to define the voltage source and to close the dialog.

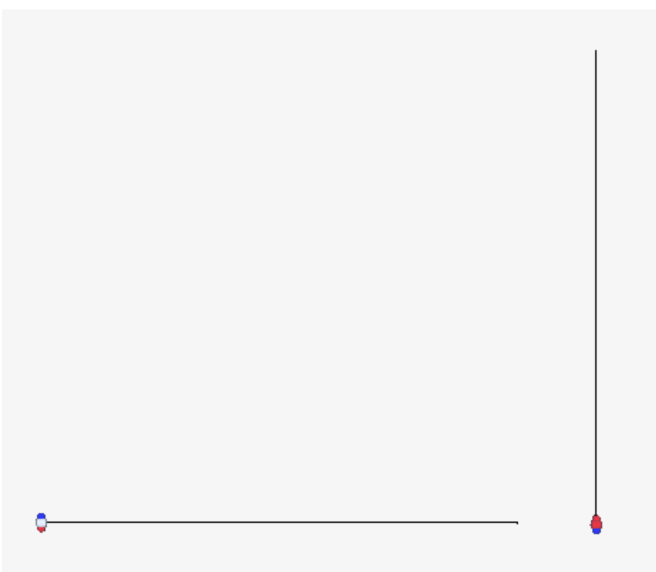
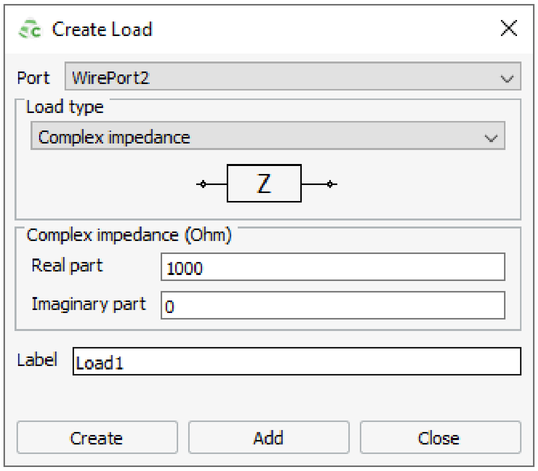


Figure 123: Front view of the monopole and transmission line together with their wire ports. The wire port with a voltage source applied to it, is indicated by a red sphere.

Adding a Complex Load to a Port

Add a resistive load to the port of the transmission line.

1. On the **Source/Load** tab, in the **Loads/Networks** group, click the **Add Load** icon.
2. Specify the port for the load as **WirePort2**.
3. Set the **Real part** of the complex impedance to `1000`.



4. Click **Create** to create the load and close the dialog.

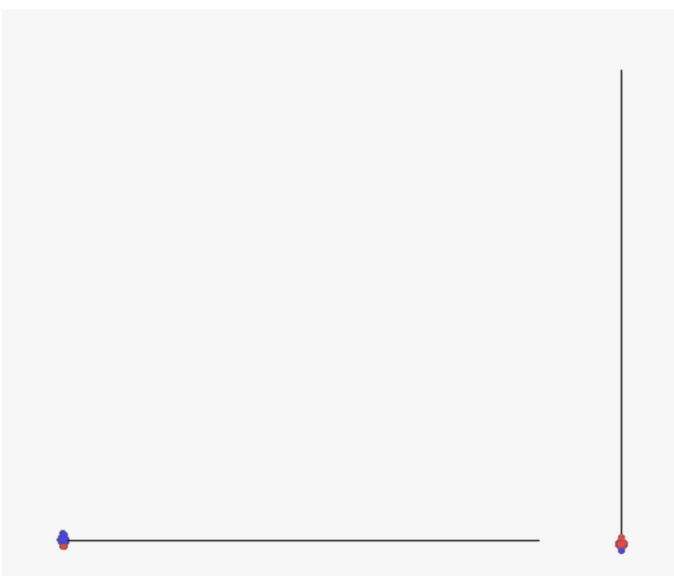


Figure 124: Front view of the monopole and transmission line together with their wire ports. The load is indicated by a blue sphere.

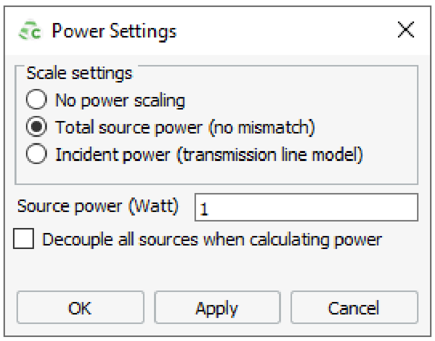
5.4.7 Setting the Radiated Power Level

Specify the power settings to scale the radiated power.

1. On the **Source/Load** tab, in the **Settings** group, click the (P) **Power** icon.

The radiated power should be 1 Watt. Power losses as a result of source mismatch are deducted before the 1 Watt is calculated.

2. Click **Total source power (no mismatch)**.
3. Enter a source power of `1` Watt.



4. Click **OK** to specify the source power and to close the dialog.

5.4.8 Setting the Simulation Frequency

Specify the frequency range of interest. For this example continuous frequency sampling is used where Feko determines the frequency sampling for optimal interpolation automatically.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.
2. On the **Solution Frequency** dialog, select **Continuous (interpolated) range** from the drop-down list.

Specify the frequency range between 1 MHz and 30 MHz.

3. Enter the start frequency and end frequency.
   - **Start frequency (Hz)**: `1e6`
   - **End frequency (Hz)**: `30e6`

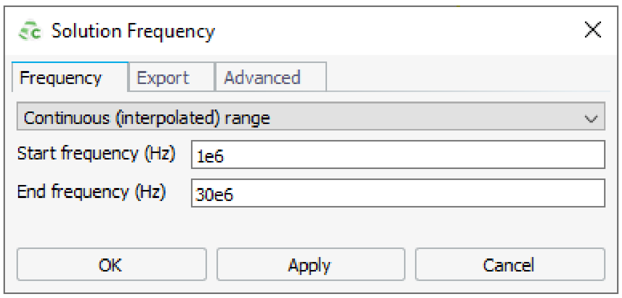


*Figure 125: The **Solution Frequency** dialog.*

4. Click **OK** to specify the frequency and to close the dialog.

5.4.9 Modifying the Auto-Generated Mesh

When the frequency is set or local mesh settings are applied to the geometry, the automatic mesh algorithm calculates and creates the mesh automatically while the GUI is active using default mesh settings. When required, these mesh settings may be modified.

Specify the global wire segment radius to be used in the model.

1. Open the **Modify Mesh Settings** dialog using one of the following workflows:
   - On the **Mesh** tab, in the **Meshing** group, click the **Create Mesh** icon.
   - Press Ctrl+M to use the keyboard shortcut.
2. On the **Modify Mesh Settings** dialog, set the **Wire segment radius** to `0.004`.

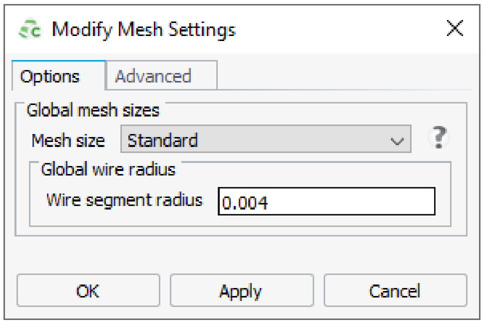


*Figure 126: The **Modify Mesh Settings** dialog.*

3. Click **OK** to create the mesh and to close the dialog.

5.4.10 Setting a Local Wire Radius for the Monopole

Specify a local wire radius for the monopole.

The monopole and transmission line are each assigned a different wire radius. Due to the differences in the wire radii, we specify a local wire radius for the monopole.

> **Note:** Local mesh refinement takes precedence over global mesh settings.

1. In the model tree (**Construction** tab), select **Monopole**. In the details tree, select **Wire1**.
2. From the right-click context menu, select **Properties**.
3. On the **Modify Edge** dialog (**Properties** tab), specify the following:
    a) Select the **Local wire radius** check box.
    b) Set the **Radius** to `0.015`.

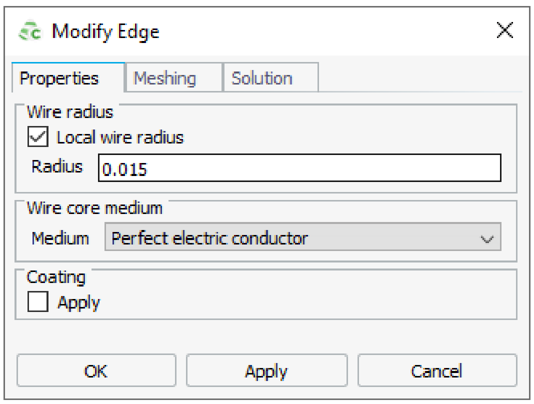
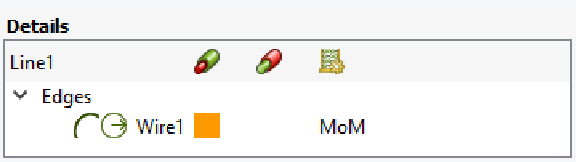


*Figure 127: The **Modify Edge** dialog.*

4. Click **OK** to apply the properties and to close the dialog.

> **Note:** The icon in the details tree indicate that a local wire radius is applied.



5.4.11 Saving the Model

Save the model to a CADFEKO `.cfx` file.

1. Save the model using one of the following workflows:
    - On the **Home** tab, in the **File** group, click the **Save** icon.
    - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Coupling.cfx`.
3. Click **Save** to close the dialog.

5.5 Launching the Solver

Launch the Solver to calculate the results. No requests were added to this model since impedance and current information are calculated automatically for all voltage and current sources in the model.

1. Launch the Solver using one of the following workflows:
    - On the **Solve/Run** tab, in the **Run/Launch** group, click the  **Feko Solver** icon.
    - On the application launcher toolbar, click the **Feko Solver** icon in the  group.
    - Press Alt+4 to use the keyboard shortcut.

If the model contains unsaved changes, the **Save Model** dialog is displayed.

2. Click **Yes** to save the model and to close the **Save Model** dialog.

The Feko Solver is launched and the **Executing runfeko** dialog is displayed. The dialog gives step-by-step feedback as the simulation progresses.

3. Click **Details** to expand the **Executing runfeko** to view the step-by-step feedback.

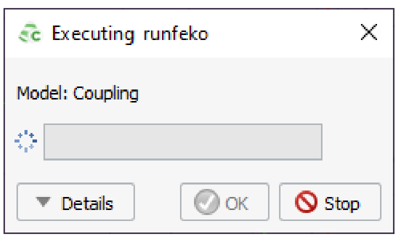


*Figure 128: The **Executing runfeko** dialog.*

5.6 Viewing the Results in POSTFEKO

Display the model as well as the results using the post-processor component, POSTFEKO.

5.6.1 Reviewing POSTFEKO and Launching OPTFEKO

Open POSTFEKO from within CADFEKO.

Use one of the following workflows to launch POSTFEKO:

- On the **Solve/Run** tab, in the **Run/Launch** group, click the **POSTFEKO** icon.
- On the application launcher toolbar, click the **POSTFEKO** icon in the group.
- Press Alt+3 to use the keyboard shortcut.

POSTFEKO opens by default with a single 3D view containing the model geometry.

Viewing the Load Current

View the load current of the transmission line on a Cartesian graph.

1. Create a new Cartesian graph.
   a) On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. Add the load current to the Cartesian graph.
   a) On the **Home** tab, in the **Add results** group, click the **Loads/Networks** icon. From the drop-down list, select **Load1**.
3. View the load current (in dB) versus frequency.
   a) On the result palette, on the **Traces** panel, select **Load1**.
   b) On the **Quantity** panel, from the drop-down list select **Current**.
   c) On the **Quantity** panel, select the **dB** check box.
4. Change the legend position to top-right.
   a) On the **Display** tab, in the **Legend** group, click the **Position** icon. From the drop-down list select **Overlay top right**.
5. Remove the graph footer.
   a) On the **Display** tab, in the **Display** group, click the **Chart text** icon.
   b) In the **Graph footer** field, clear the **Auto** check box and delete the text.
   c) Click **OK** to apply the text changes and to close the dialog.

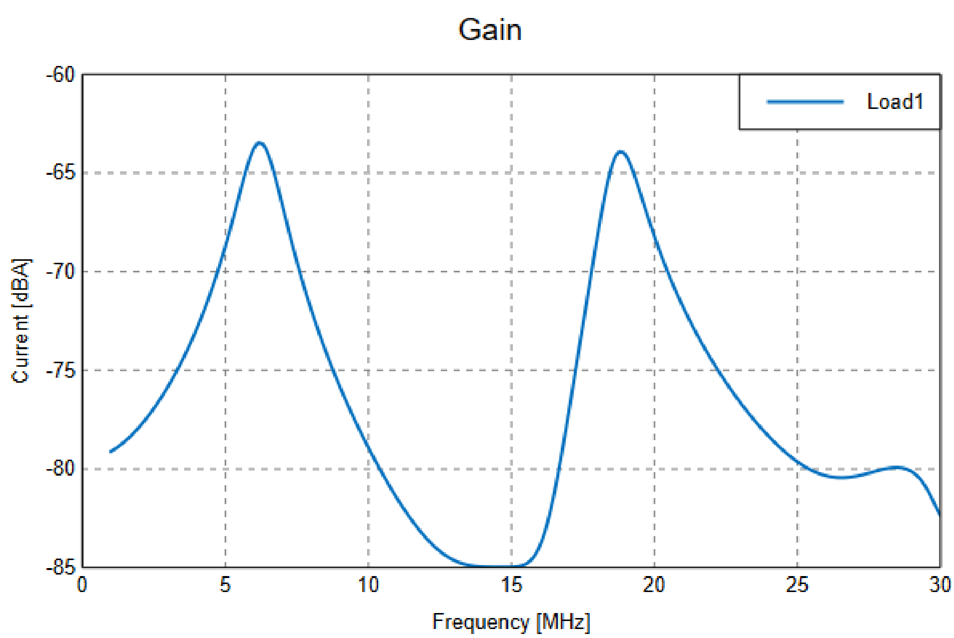


Figure 129: The load current in dB versus frequency.

Viewing the Input Impedance

View the source input impedance (real and imaginary) of the transmission line on a Cartesian graph.

1. Create a new Cartesian graph.
   a) On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. Add the source input impedance to the Cartesian graph.
   a) On the **Home** tab, in the **Add results** group, click the **Source data** icon. From the drop-down list, select **VoltageSource1**.
3. View the real part of the impedance and rename the legend text.
   a) On the result palette, in the **Traces** panel, select **VoltageSource1**.
   b) Press F2 and rename the trace to **Real**.
   c) On the **Quantity** panel, from the drop-down list select **Impedance**.
   d) On the **Quantity** panel, click **Real**.
4. Duplicate the **VoltageSource1** trace using one of the following workflows:
   - On the **Cartesian** context tab, on the **Trace** tab, in the **Manage** group, click the **Duplicate trace** icon.
   - Press Ctrl+K to use the keyboard shortcut.
5. View the imaginary part of the impedance and rename the legend text.
   a) On the result palette, in the **Traces** panel, select **VoltageSource1_1**.
   b) Press F2 and rename the trace to **Imaginary**.
   c) On the **Quantity** panel, click **Imaginary**.
6. [Optional] Repeat Step 4 and Step 5 of Viewing the Load Current to change the legend position and remove the graph footer.

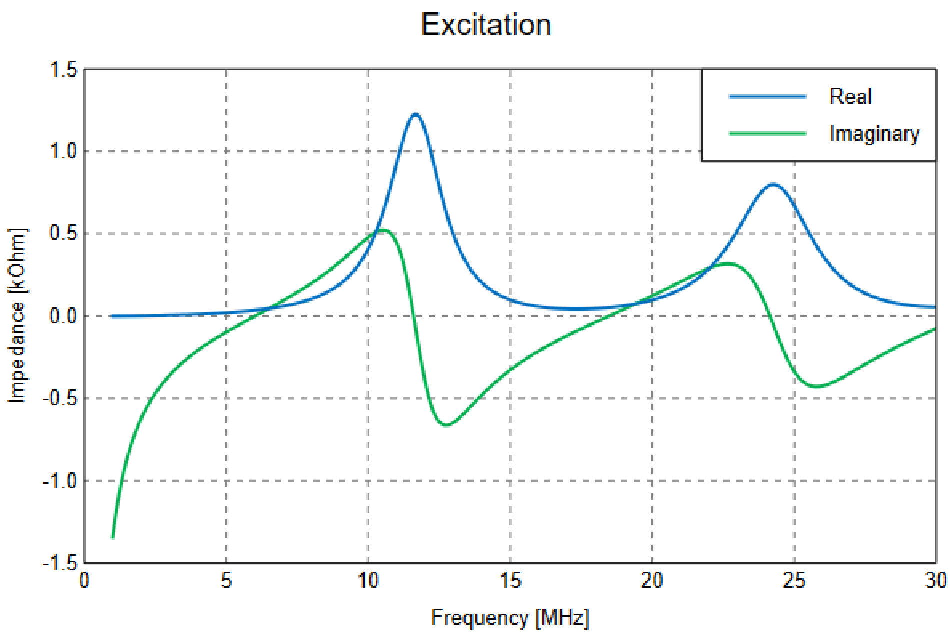


Figure 130: The input impedance (real and imaginary) versus frequency.

Formatting the Graph

Format the line colour, marker colour and marker style of a trace. Add a line, arrow (single or double), rectangle or circle to the graph to highlight certain aspects of the graph.

1. Select a trace to format.
   a) On the result palette, in the traces panel, select **VoltageSource1_1**.
2. Change the line colour to red.
   a) On the **Format** tab, in the **Line** group, click the **Line colour** icon. From the drop-down list, select the colour red.
3. Change the marker style to a triangle.
   a) On the **Format** tab, in the **Marker** group, click the **Marker style** icon. From the drop-down list select the triangle.
4. Change the marker colour to match the colour of the trace.
   a) On the **Format** tab, in the **Marker** group, click the **Marker colour** icon. From the drop-down list select the colour red.

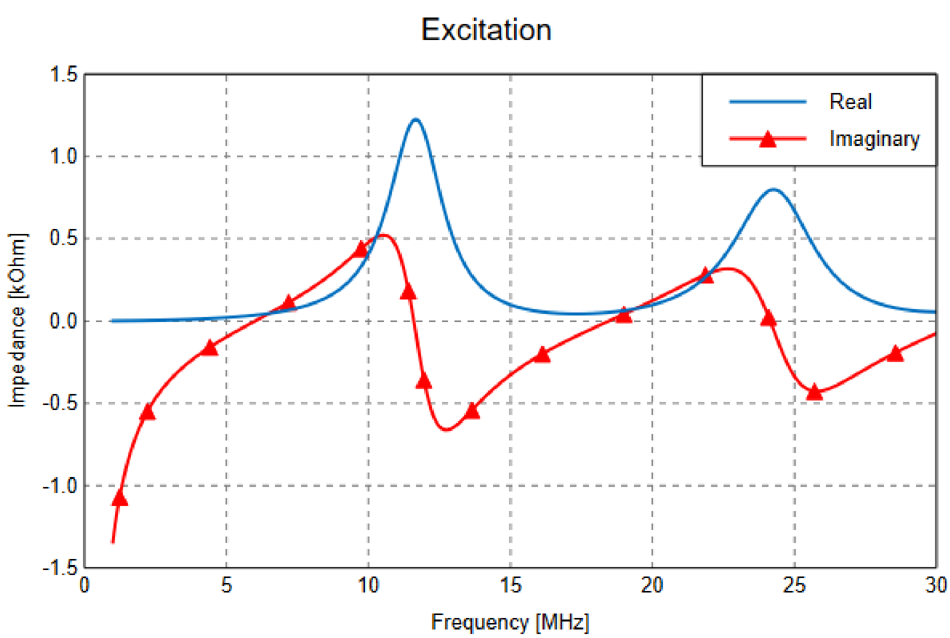


Figure 131: An example of a formatted graph.

5. [Optional] Add a line, arrow, double arrow, rectangle or circle to the graph to highlight an aspect on the graph.
   a) On the **Format** tab, in the **Drawing** group, click the **Shapes** icon. Select the required item from the drop-down list.

5.7 Final Remarks

This example showed the construction, configuration and solution of an EMC coupling scenario. The model consists of a monopole antenna and transmission line on an infinite PEC ground plane. Coupling of current into the transmission line is viewed from 1 MHz to 30 MHz.

Waveguide Power Divider

6

The example considers the transmission and reflection coefficients of a waveguide power divider.

This chapter covers the following:

6.1 Example Overview

Calculate the transmission and reflection coefficients of a waveguide power divider.

Create the power divider to split equally the power between the two output ports while minimising any power reflected back to the source port. The power is split by placing a metal pin at the junction between the three waveguide sections. The model utilises symmetry to reduce memory requirements and calculation speed.

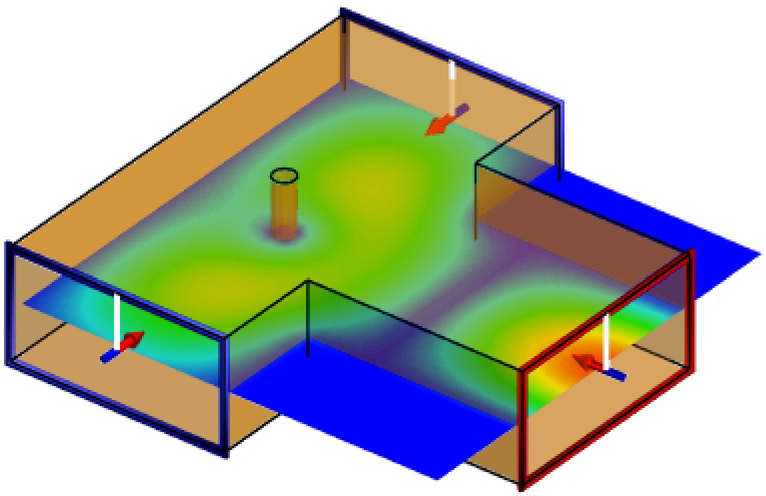


Figure 132: The waveguide power divider and instantaneous near field.

6.2 Topics Discussed in Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- CADFEKO
    - Create the pin using a cylinder.
    - Create the waveguide sections using cuboids.
    - Define waveguide ports on each end of the waveguide.
    - Add a waveguide source.
    - Set the solution frequency.
    - Specify local (fine) meshing for the waveguide ports.
    - Specify symmetry to save computational resources.
    - Specify near fields on a plane inside the waveguide.
    - Mesh the model.
    - Run CEM validate to ensure the model is electromagnetically validated.
    - Run the Solver.
- POSTFEKO
    - View the simulated input reflection coefficient on a graph.
    - View the instantaneous near field inside the waveguide.
    - View an animation of the near fields.

Note: Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

Tip: Find the completed model in the application macro library[28]:

`GS 6: Waveguide Power Divider`

28. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

6.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 30 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[29] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

29. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

6.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

6.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

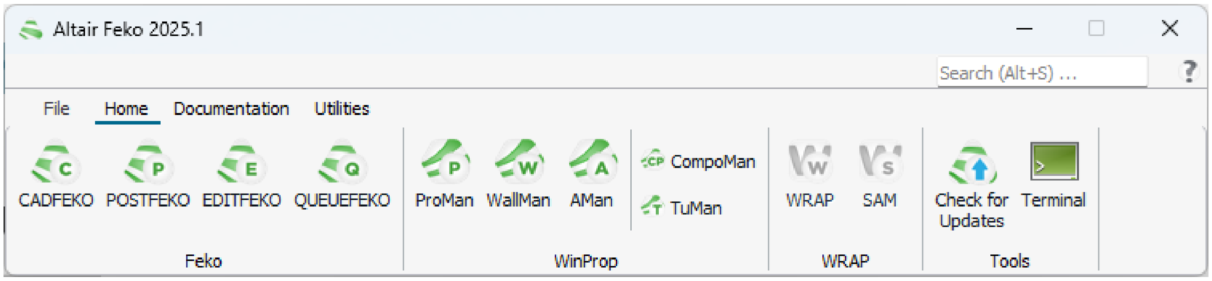


Figure 133: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[30] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

6.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the "initfeko" script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

30. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

Note: Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

6.4.3 Setting the Model Unit

Set the model unit to millimeters.

The default unit length in CADFEKO is metres. Since the structure that you will build is small, the model unit is set to millimetres. All dimensions entered will be in the new model unit.

1. Set the model unit to millimetres using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Model unit** icon.
   - On the status bar, click Unit: m.
2. On the **Model Unit** dialog, select **Millimetres (mm)**.
3. Click **OK** to change the model unit to millimetres and to close the dialog.

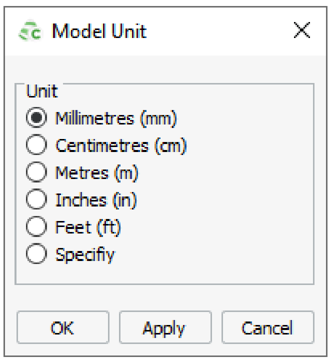


*Figure 134: The **Model Unit** dialog.*

6.4.4 Adding Variables

Define variables to create a parametric model.

A model is parametric when it is created using variable expressions. When a variable expression is modified, any items dependent on that variable are re-evaluated and automatically updated. It is the recommended construction method when creating a model, but not compulsory.

Defined variables are stored as part of the model in the `.cfx` file.

1. Open the **Create Variable** dialog using one of the following workflows:
   - On the **Construct** tab, in the **Define** group, click the **Add Variable** icon.
   - On the model tree, a right-click context menu is available on **Variables**. From the list, select **Add Variable**.

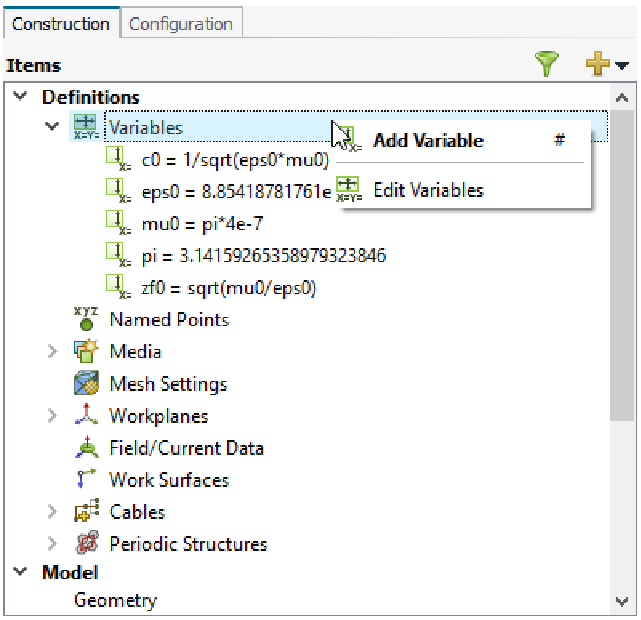


*Figure 135: The model tree (**Construction** tab).*

   - On the model tree, click the icon. From the drop-down list, select **Add Variable**.

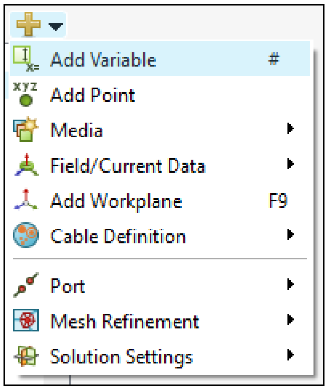


Figure 136: The  *drop-down list available in the model tree.*

- Press # to use the keyboard shortcut.

2. Create the following variables:

a) Optional: Add the variable comments.

Name	**Expression**	**Comment**
freq	9e9	The operating frequency.
lambda	*c0/freq**1000	Free space wavelength.
pin_r	1	Pin radius.
wg_h	10	Waveguide height.
wg_w	20	Waveguide width.

Tip:

- Click **Add** to keep the **Create Variable** dialog open and add more variables.
- Click **Create** to add a variable and close the **Create Variable** dialog.

6.4.5 Creating the Power Dividing Pin

Create the power dividing pin using a cylinder.

1. Create a cylinder located at the origin along the Z axis.
   a) On the **Construct** tab, in the **Create Solid** group, click the **Cylinder** icon.
   b) Create a cylinder using the **Base centre, radius, height** definition method.
   c) Use the following dimensions:
      - **Radius (R)**: *pin_r*
      - **Height (H)**: *wg_h*

> **Note:** Default values are used on geometry creation dialogs to allow a preview in the 3D view. You may change the values as required.

   a) Click **Create** to create the cylinder and to close the dialog.

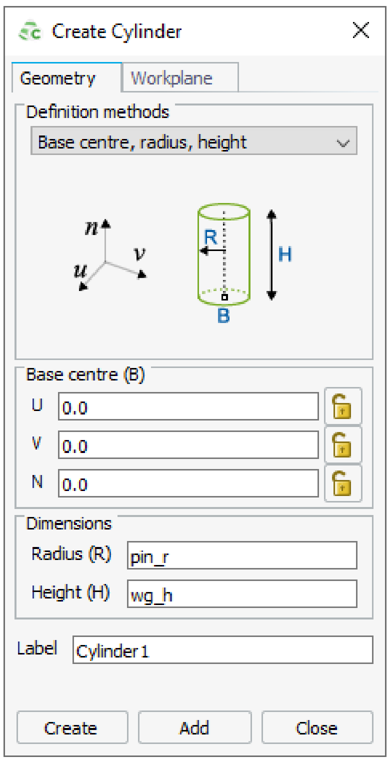


*Figure 137: The **Create Cylinder** dialog.*

2. Zoom to extents of the 3D view using one of the following workflows:
   - On the **View** tab, in the **Zoom** group, click the **Zoom to extents** icon.
   - Press F5 to use the keyboard shortcut.

6.4.6 Creating the Waveguide Sections

Create the waveguide sections using two cuboids.

1. Create the first waveguide section.
   a) On the **Construct** tab, in the **Create Solid** group, click the **Cuboid** icon.
   b) Create the cuboid using the **Base centre, width, depth, height** definition method.
   c) Use the following dimensions:
      - **Base centre (C)**: (0, 0, 0)
      - **Width (W)**: *wg_w*
      - **Depth (D)**: 2**wg_w*
      - **Height (H)**: *wg_h*
      - **Label**: `Cuboid1`

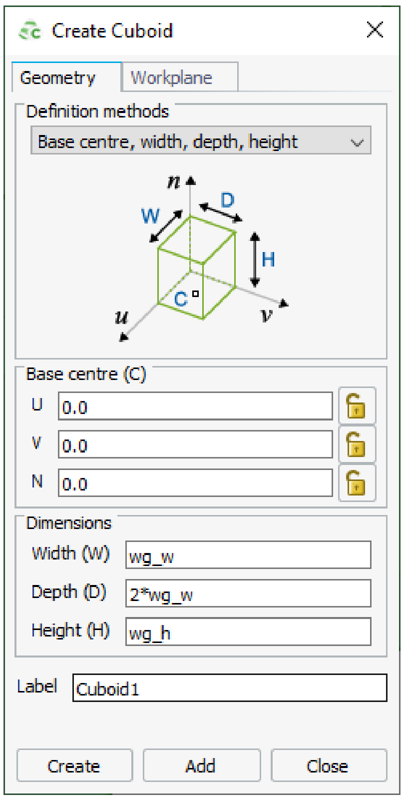


*Figure 138: The **Create Cuboid** dialog.*

   d) Click **OK** to create the waveguide section and to close the dialog.
2. Create the second waveguide section.
   a) On the **Construct** tab, in the **Create Solid** group, click the **Cuboid** icon.
   b) Create the cuboid using the **Base corner, width, depth, height** definition method.
   c) Use the following dimensions:
      - **Base corner (C)**: (*wg_w*/2, -*wg_w*/2, 0)

- **Width (W)**: *wg_w*
- **Depth (D)**: *wg_w*
- **Height (H)**: *wg_h*
- **Label**: `Cuboid2`

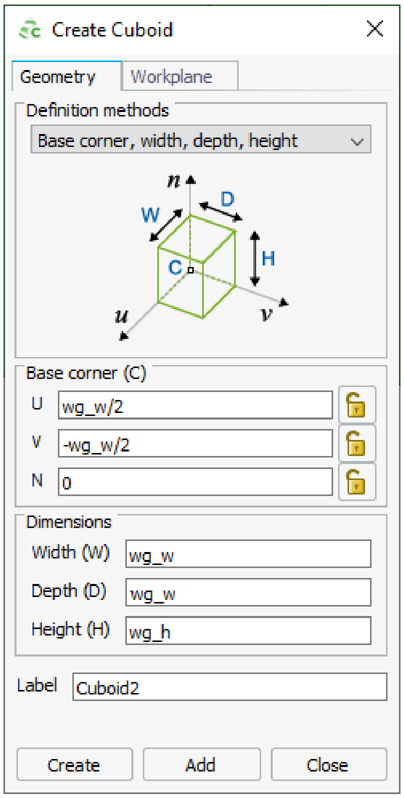


*Figure 139: The **Create Cuboid** dialog.*

d) Click **OK** to create the waveguide section and to close the dialog.

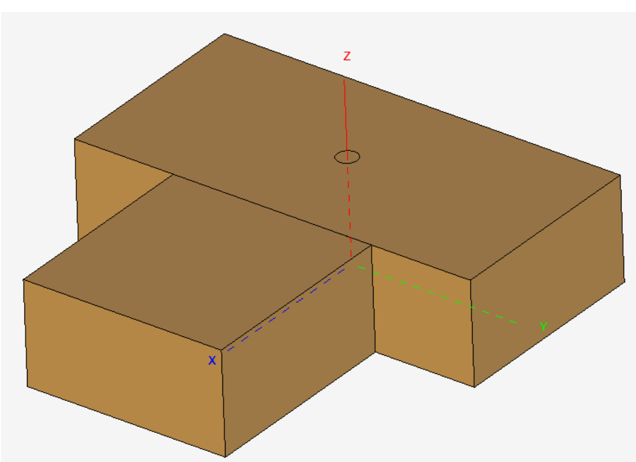


Figure 140: The two waveguide sections and power dividing pin.

6.4.7 Unioning the Geometry for Mesh Connectivity

Union the geometry (*Cuboid1* and *Cuboid2*) to create a single geometry part. A single geometry part will ensure mesh connectivity when the model is meshed.

1. In the model tree, select **Cuboid1** and **Cuboid2**[31].

   Tip: To select multiple objects, press and hold Ctrl while you click the items.

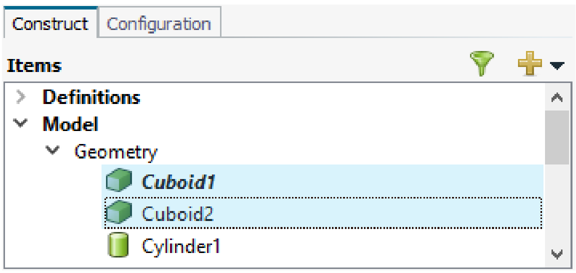


Figure 141: Cuboid1 and Cuboid2 selected in the model tree.

2. On the **Construct** tab, in the **Modify** group, click the **Union** icon.

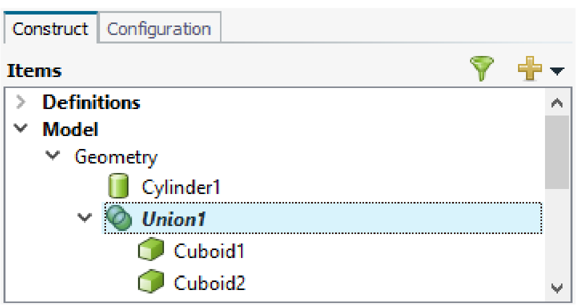


*Figure 142: The model tree showing **Union1** (the union between **Cuboid1** and **Cuboid2**).*

31. Alternative method is to select the items in the 3D view.

6.4.8 Removing Redundant Faces

Use the simplify tool to remove redundant faces.

1. In the model tree, select **Union1**.
2. On the **Transform** tab, in the **Simplify** group, click the **Simplify** icon.
3. Use the default settings on the **Simplify** dialog.

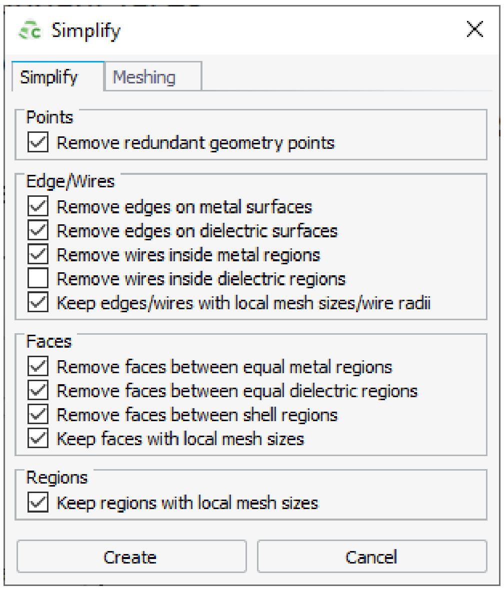


*Figure 143: The **Simplify** dialog.*

4. Click **Create** to simplify **Union1** and to close the dialog.

The redundant face at the junction between the two cuboids is removed.

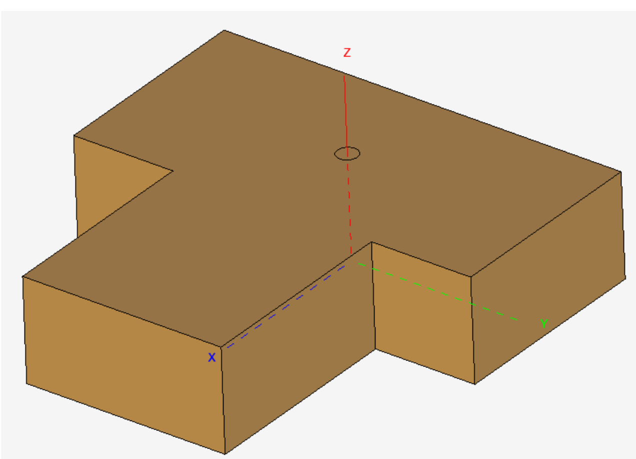


Figure 144: The waveguide section after the redundant faces were removed.

6.4.9 Changing the Waveguide to a Shell (Hollow) Part

Change the solid waveguide part to a shell (hollow) part with metal walls.

1. In the model tree, select **Union1**.
2. In the details tree, under **Regions**, select **Region1**.
3. From the right-click context menu, select **Properties**.

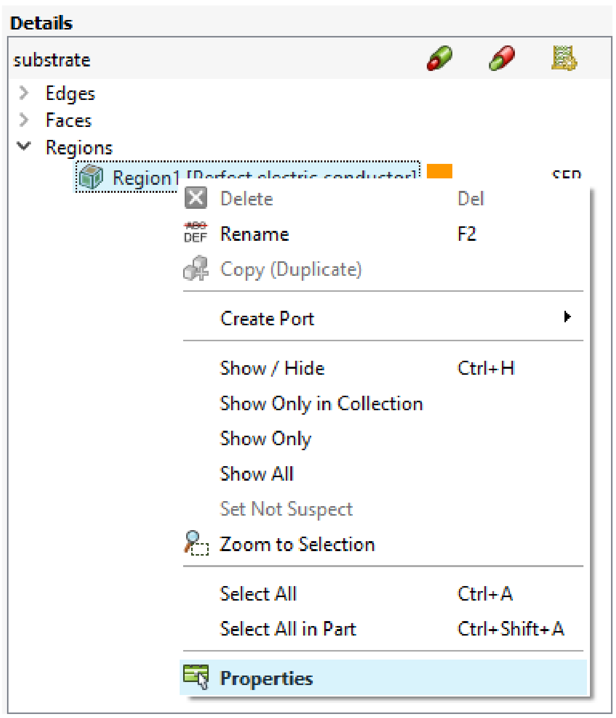


Figure 145: The right-click context menu options available for regions.

4. On the **Modify Region** dialog (**Properties** tab), set **Medium** to **Free space**.

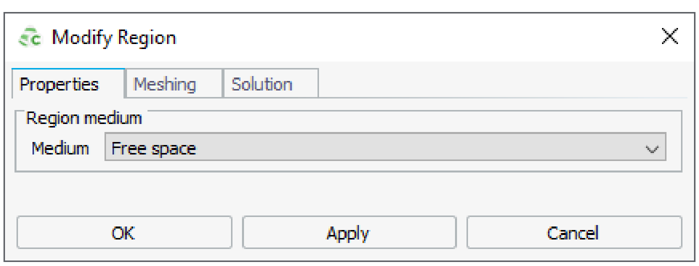


*Figure 146: The **Modify Region** dialog.*

5. Click **OK** to modify the region properties and to close the dialog.

6.4.10 Unioning the Waveguide and Power Dividing Pin

Create a single geometry part from the waveguide and power divider pin to ensure mesh connectivity when the model is meshed.

1. In the model tree, select **Cylinder1** and **Union1**[32].

   Tip: To select multiple objects, press and hold Ctrl while you click the items.

2. On the **Construct** tab, in the **Modify** group, click the **Union** icon.

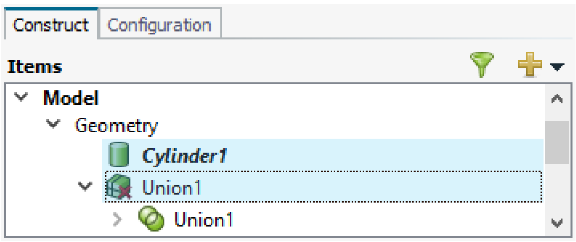


*Figure 147: Select **Cylinder1** and **Union1** in the model tree.*

32. Alternative method is to select the items in the 3D view.

6.4.11 Ports, Sources and Loads in CADFEKO

Voltage sources and discrete loads are applied to ports and not directly to the model geometry or mesh. A port must be defined before a source or load can be added.

Adding Waveguide Ports

Define waveguide ports with the correct orientation.

> **Note:** A port is a mathematical representation of where energy can enter (source) or leave a model (sink). Use a port to add sources and discrete loads to a model.
>
> Waveguide ports without sources are considered to be absorbing waveguide terminations.

1. Add the first waveguide port on the face at the most positive X position.
   a) In the 3D view, repeatedly left-click until the face is highlighted.

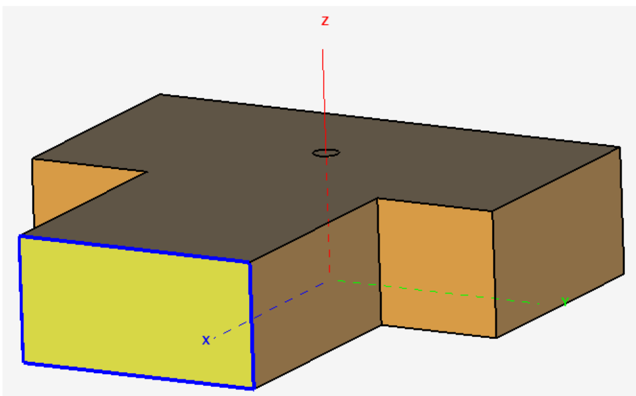


Figure 148: The face at the most positive X position is selected.

   b) Open the **Create Waveguide Port** dialog using one of the following workflows:
   - On the **Source/Load** tab, in the **Ports** group, click the **Waveguide Port** icon.
   - In the details tree, a right-click context menu is available on the face. From the list, click **Create Port** > **Waveguide Port**.

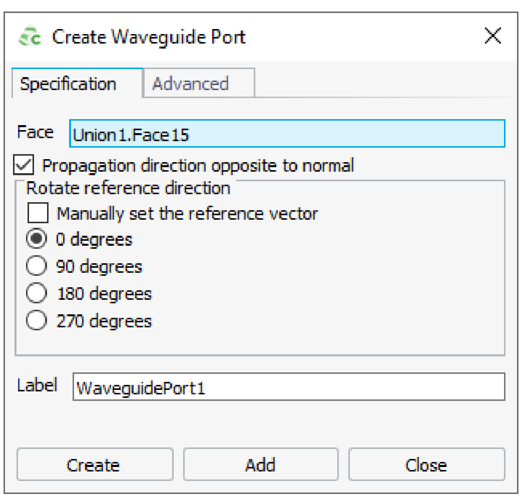


Figure 149: The ***Create Waveguide Port*** *dialog.*

   c) Use the default settings for the port.

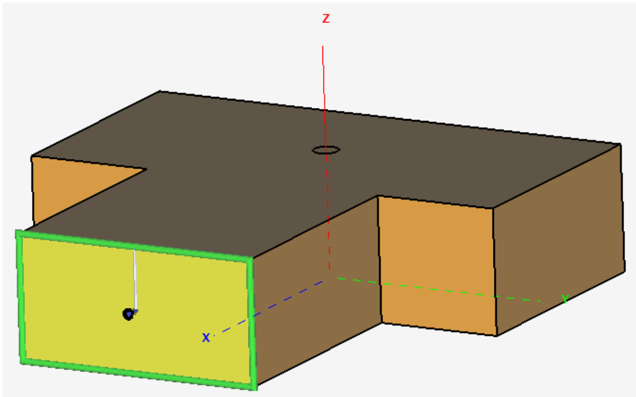


Figure 150: The preview of the waveguide port is displayed in green.

> **Note:** The white line is the reference vector and shows the direction of *m*, where *m* is the number of half-wavelengths across the width of the waveguide.

d) Click **Add** to add the waveguide port, but do not close the dialog.

2. Add the second waveguide port at the face at the most negative Y position.

a) On the dialog, click on the **Face** user input field to make it active. An active field is highlighted in blue (see Figure 149).

b) In the 3D view, repeatedly left-click until the face is highlighted.

c) Click **Add** to add the waveguide port, but do not close the dialog.

3. Add the third waveguide port at the face at the most positive Y position.

a) On the dialog, click on the **Face** field to make it active. An active field is highlighted in blue.

b) In the 3D view, repeatedly left-click until the face is highlighted.

c) Click **180 degrees** to ensure the correct reference direction.

d) Click **Create** to add the port and to close the dialog.

4. Enable the port annotations. On the **3D View** context tab, on the **Display Options** tab, in the **Entity Display** group, click the **Port Annotations** icon.

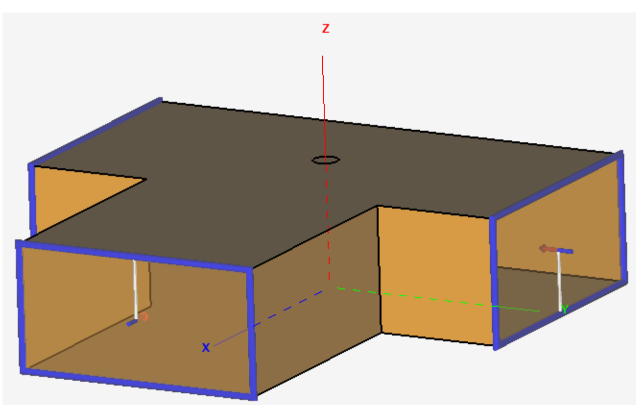


Figure 151: The waveguide power divider with three defined waveguide ports.

Adding a Waveguide Source

Add a waveguide source to the first port using the fundamental mode.

> **Note:** Default values are used in this example. The fundamental mode for this source will be excited (TE10).
>
> Add multiple modes as a single source by selecting **Specify modes manually**.

1. On the **Source/Load** tab, in the **Sources on Ports** group, click the **Waveguide Source** icon.
2. In the drop-down list, select **WaveguidePort1**.

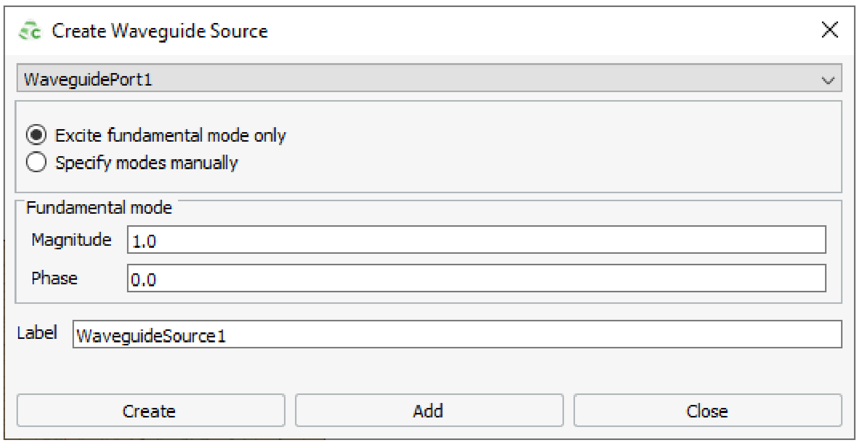


*Figure 152: The **Create Waveguide Source** dialog.*

3. Click **Create** to create the source and to close the dialog.

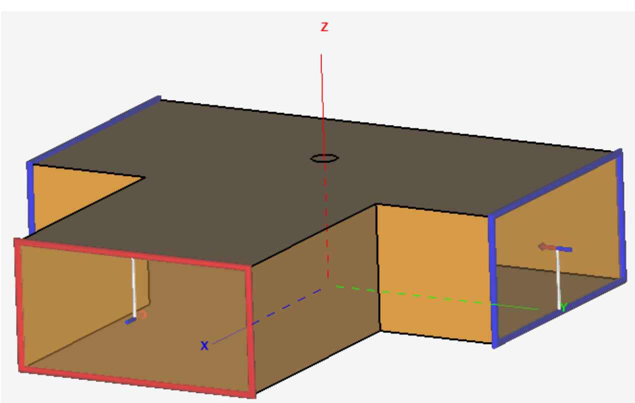


Figure 153: A port with a source is indicated in red.

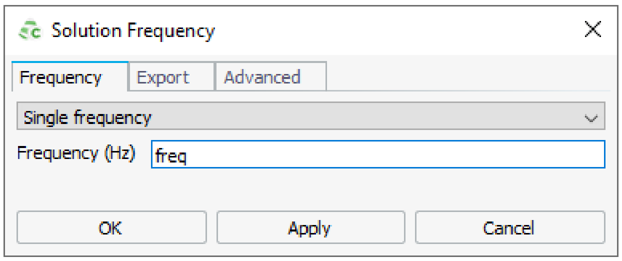
6.4.12 Setting the Simulation Frequency

Specify the frequency range of interest.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.

A variable was created at the beginning of the example that contains the solution frequency.

2. In the **Frequency (Hz)** field, enter *freq*.



3. Click **OK** to close the dialog.

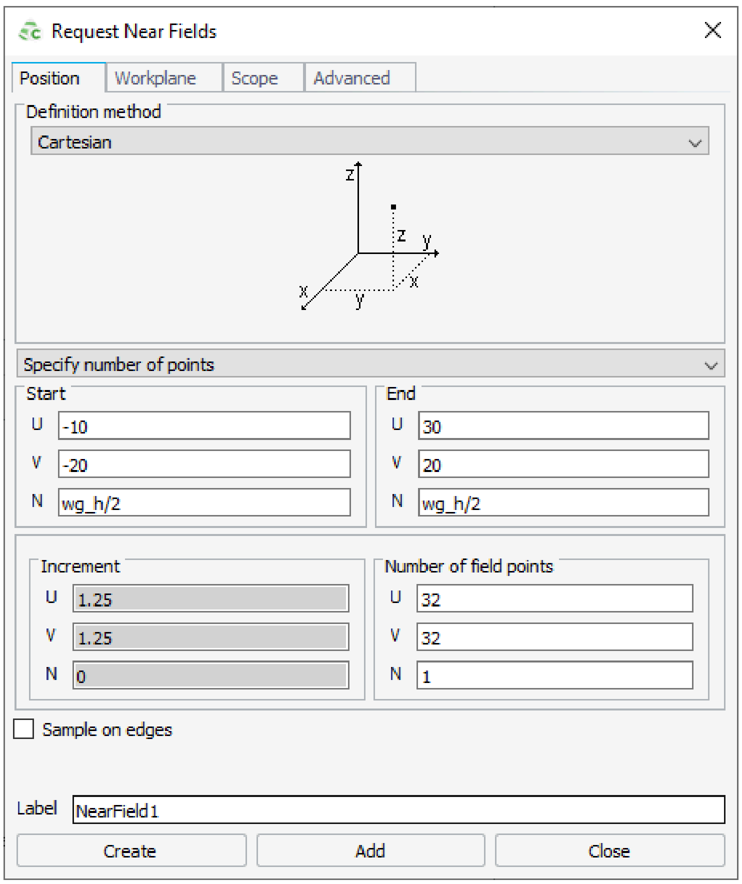
   With the frequency set to *freq*, the actual frequency is 9 GHz. In the model tree, click the **Configuration** tab to view the specified simulation frequency.

6.4.13 Adding a Near Field Request

Add a near field request to calculate the fields on a surface through the centre of the waveguide.

1. On the **Request** tab, in the **Solution Requests** group, click the **Near Fields** icon.
2. On the **Request near fields** dialog, enter the values as indicated.

Dimension	Start	End	Number of Field Points
U	-10	30	32
V	-20	20	32
N	wg_h/2	wg_h/2	1



3. Clear the **Sample on edges** check box.

> **Note:** A near field request on PEC boundaries will result in a warning by the Solver.

4. In the **Label** field, use the default (*NearField1*).
5. Click **Create** to add the near field request and to close the dialog.

6.4.14 Setting Local Mesh Sizes for Waveguide Port Faces

Refine the mesh locally at the waveguide port faces. The faces for the waveguide ports require a finer mesh to represent the highest mode that should be taken into account. The higher order modes are typically evanescent modes.

1. In the 3D view, select the three waveguide port faces.
2. From the right-click context menu, select **Properties**.
3. On the **Multiple Entities** dialog (**Meshing** tab), specify the following:
    a) Select the **Local mesh size** check box.
    b) Set the **Mesh size** to *lambda/20*.

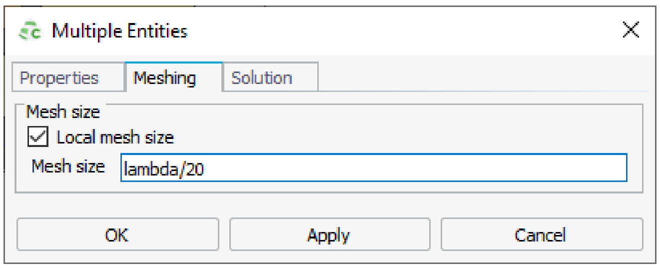


*Figure 154: The **Multiple Entities** (**Meshing** tab) dialog.*

4. Click **OK** to apply the properties and to close the dialog.

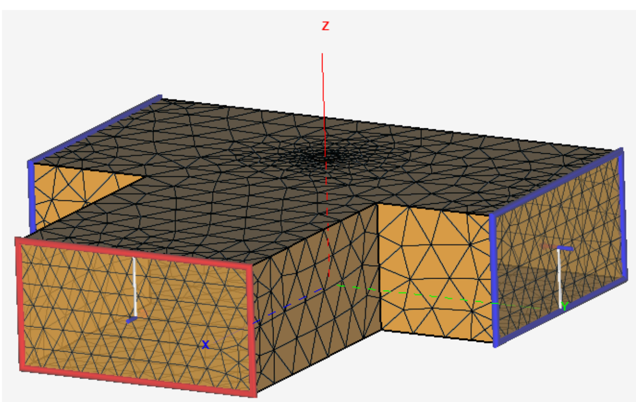
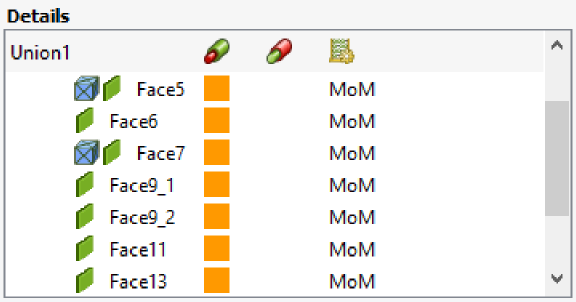


Figure 155: Note the localised mesh refinement at the waveguide port faces.

Note: The icon in the details tree indicate that a local mesh setting is applied.



6.4.15 Saving the Model

Save the model to a CADFEKO `.cfx` file.

1. Save the model using one of the following workflows:
    - On the **Home** tab, in the **File** group, click the **Save** icon.
    - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Waveguide_Divider.cfx`.
3. Click **Save** to close the dialog.

6.5 Launching the Solver

Launch the Solver to calculate the results. No requests were added to this model since impedance and current information are calculated automatically for all voltage and current sources in the model.

1. Launch the Solver using one of the following workflows:
    - On the **Solve/Run** tab, in the **Run/Launch** group, click the **Feko Solver** icon.
    - On the application launcher toolbar, click the **Feko Solver** icon in the group.
    - Press Alt+4 to use the keyboard shortcut.

If the model contains unsaved changes, the **Save Model** dialog is displayed.

2. Click **Yes** to save the model and to close the **Save Model** dialog.

The Feko Solver is launched and the **Executing runfeko** dialog is displayed. The dialog gives step-by-step feedback as the simulation progresses.

3. Click **Details** to expand the **Executing runfeko** to view the step-by-step feedback.

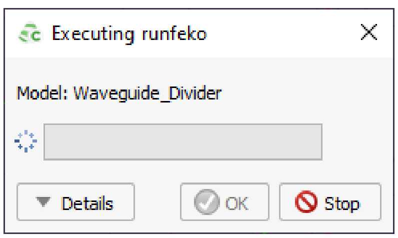


*Figure 156: The **Executing runfeko** dialog.*

6.6 Reviewing POSTFEKO and Launching OPTFEKO

Open POSTFEKO from within CADFEKO.

Use one of the following workflows to launch POSTFEKO:

- On the **Solve/Run** tab, in the **Run/Launch** group, click the **POSTFEKO** icon.
- On the application launcher toolbar, click the **POSTFEKO** icon in the group.
- Press Alt+3 to use the keyboard shortcut.

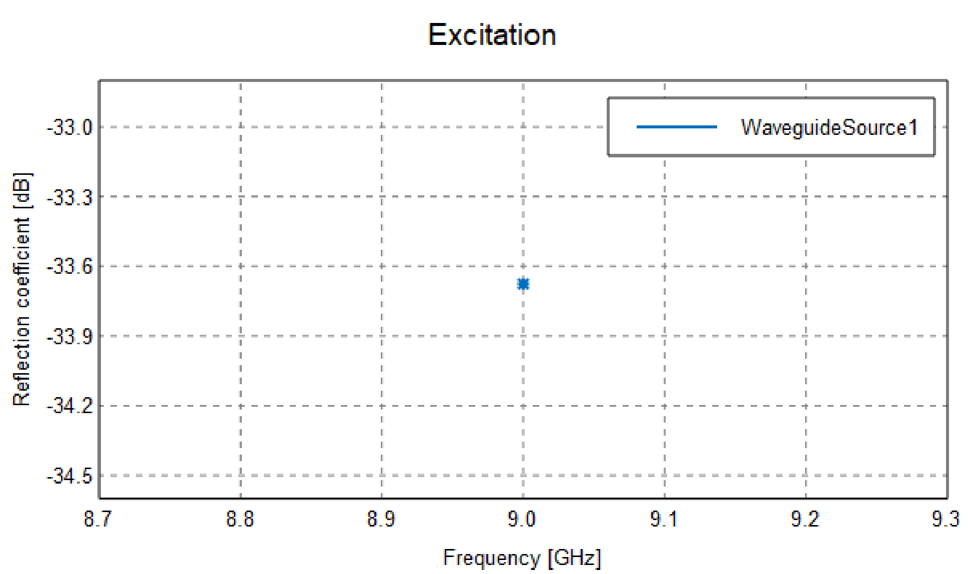
POSTFEKO opens by default with a single 3D view containing the model geometry.

6.6.1 Viewing the Input Reflection Coefficient

Plot the input reflection coefficient on a Cartesian graph in dB.

The model was solved at a single frequency only. Therefore the graph will contain a single data point.

1. Create a new Cartesian graph.
   a) On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
2. Add the input reflection coefficient to the Cartesian graph.
   a) On the **Home** tab, in the **Add results** group, click the **Source data** icon. From the drop-down list, select **WaveguideSource1**.
3. View the input reflection coefficient in dB.
   a) On the result palette, in the **Traces** panel, select **WaveguideSource1**.
   b) On the **Quantity** panel, select the **dB** check box.
4. Change the legend position to top-right.
   a) On the **Display** tab, in the **Legend** group, click the **Position** icon. From the drop-down list select **Overlay top right**.
5. Remove the graph footer.
   a) On the **Display** tab, in the **Display** group, click the **Chart text** icon.
   b) In the **Graph footer** field, clear the **Auto** check box and delete the text.
   c) Click **OK** to apply the text changes and to close the dialog.



6. [Optional] Add annotations to the graph.
   a) On the **Cartesian** context tab, on the **Measure** tab, on the **Custom annotations** group, click the **Point** icon. From the drop-down list, click **Global maximum**.

> **Note:** The power reflected back to Port1 is more than 30 dB lower than the power applied to the same port.

6.6.2 Viewing the Near Fields

Display the calculated near fields in the 3D view. Animate the instantaneous near field.

1. Select the **3D View1** window.
2. Enable mesh opacity to view the near field inside the waveguide.
   a) On the **3D View** contextual tabs set, on the **Mesh** tab, in the **Opacity** group, click the **Mesh opacity** icon. From the drop-down list, select a value of **40%**.
3. On the **Home** tab, in the **Add results** group, click the **Near Fields** icon. From the drop-down list, select **NearField1**.
4. Animate the phase of the near field.
   a) In the result palette, in the **Quantity** panel, select **Instantaneous magnitude**.
   b) On the **3D View** contextual tabs set, on the **Animate** tab, on the **Settings** group, click the **Type** icon. From the drop-down list, select the **Phase** icon.
5. Start the animation process.
   a) On the **3D View** contextual tabs set, on the **Animate** tab, on the **Control** group, click the **Play** icon.
   b) Stop the animation by clicking the **Play** icon again.

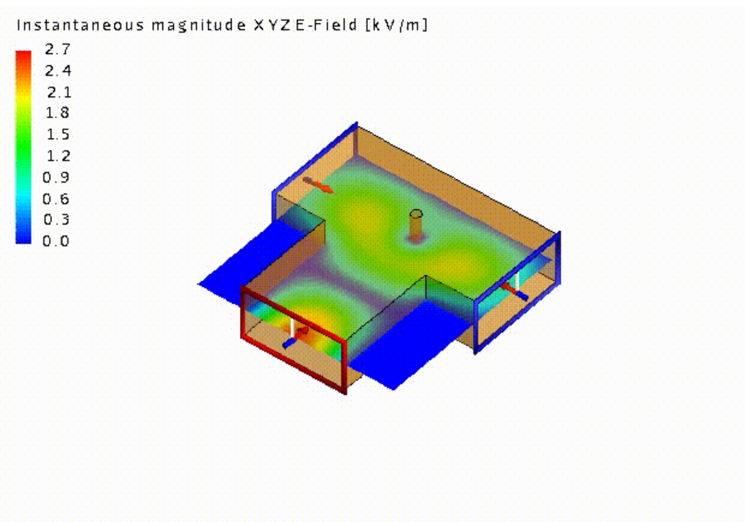


Figure 157: The near field in the power divider (refer to the WebHelp to view the animation of the instantaneous magnitude).

6.7 Final Remarks

This example showed the construction, configuration and solution of a waveguide power divider. The model consists of a hollow cuboidal sections with a cylinder (pin) in the centre. The near field and input reflection coefficient was calculated and displayed.

Optimisation of Bent Dipole and Plate

7

The example considers the optimisation of the gain of a bent dipole in front of a plate.

This chapter covers the following:

7.1 Example Overview

Calculate and maximise the gain of a bent dipole in front of a plate. Optimise the dipole-plate separation distance and the dipole bend-angle.

The goal is to maximize the maximum gain in the azimuth plane at a single frequency.

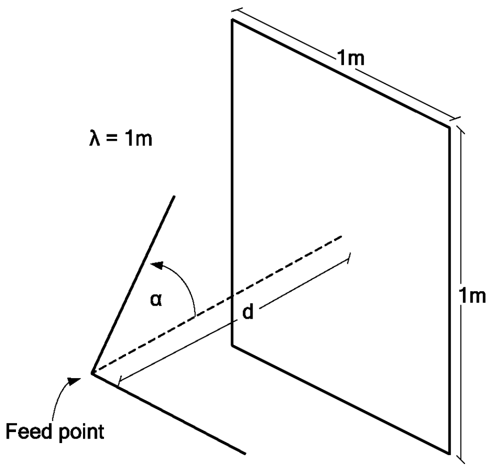


Figure 158: Sketch of the model showing dimensions and other relevant parameters.

7.2 Topics Discussed in this Example

Before starting this example, check if the topics discussed in this example are relevant to the intended application and experience level.

The topics discussed in this example are:

- CADFEKO
    - Define an optimisation search in CADFEKO.
    - Run the optimiser (OPTFEKO).
    - Add a wire port to a wire segment.
    - Specify symmetry to save computational resources.
    - Add a voltage source to a wire port.
    - Mesh the model.
    - Run CEM validate to ensure the model is electromagnetically validated.
    - Run the Solver
- POSTFEKO
    - View the optimisation results in POSTFEKO.

> **Note:** Follow the example steps in the order it is presented as each step uses its predecessor as a starting point.

> **Tip:** Find the completed model in the application macro library[33]:
>
> `GS 7: Optimisation of a Bent Dipole and Plate`

33. The application macro library is located on the **Home** tab, in the **Scripting** group. Click the **Application Macro** icon and from the drop-down list, select **Getting Started Guide.**

7.3 Example Prerequisites

Before starting this example, ensure that the system satisfies the minimum requirements.

The requirements for this example are:

- Feko 2025.1 or later should be installed.
- It is recommended that you watch the demo video before attempting this example.
- This example should not take longer than 20 minutes to complete.

> **Note:** When using CADFEKO over a remote desktop connection, you may need to enable 3D support for remote desktop[34] for the host's graphics card should a crash occur when clicking **New Project** in CADFEKO.

34. See the **Troubleshooting** section in the Appendix of the Feko User Guide for more details.

7.4 Creating the Model in CADFEKO

Create the model geometry using the CAD component, CADFEKO.

7.4.1 Launching CADFEKO (Windows)

There are several options available to launch CADFEKO in Microsoft Windows.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.

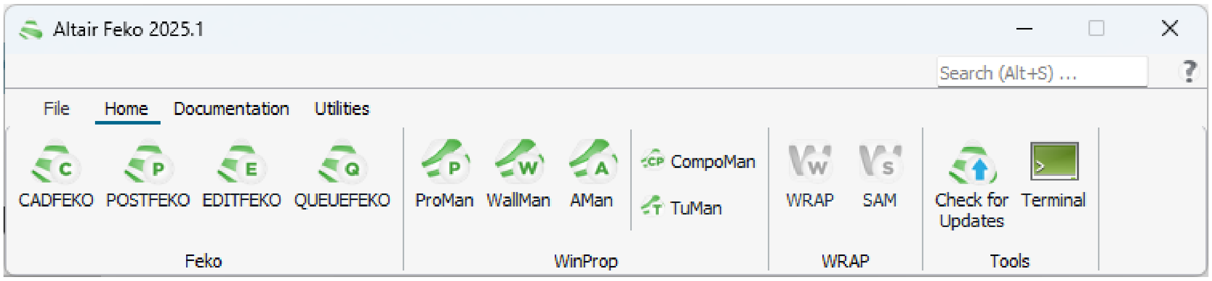


Figure 159: The Launcher utility.

- Open CADFEKO by double-clicking on a `.cfx`[35] file.
- Open CADFEKO from other components, for example, from inside POSTFEKO or EDITFEKO.

> **Note:** If the application icon is used to launch CADFEKO, no model is loaded and the start page is shown. Launching CADFEKO from other Feko components automatically loads the model.

7.4.2 Launching CADFEKO (Linux)

There are several options available to launch CADFEKO in Linux.

Launch CADFEKO using one of the following workflows:

- Open CADFEKO using the Launcher utility.
- Open a command terminal. Use the absolute path to the location where the CADFEKO executable resides, for example:

```
/home/user/2025.1/altair/feko/bin/cadfeko
```

- Open a command terminal. Source the "initfeko" script using the absolute path to it, for example:

```
. /home/user/2025.1/altair/feko/bin/initfeko
```

  Sourcing `initfeko` ensures that the correct Feko environment is configured. Type `cadfeko` and press Enter.

35. A `.cfx` file is created by CADFEKO and contains the meshed and/or unmeshed CADFEKO model as well as the calculation requests.

> **Note:** Take note that sourcing a script requires a dot (".") followed by a space (" ") and then the path to `initfeko` for the changes to be applied to the current shell and not a sub-shell.

7.4.3 Adding Variables

Define variables to create a parametric model.

A model is parametric when it is created using variable expressions. When a variable expression is modified, any items dependent on that variable are re-evaluated and automatically updated. It is the recommended construction method when creating a model, but not compulsory.

Defined variables are stored as part of the model in the `.cfx` file.

1. Open the **Create Variable** dialog using one of the following workflows:
    - On the **Construct** tab, in the **Define** group, click the **Add Variable** icon.
    - On the model tree, a right-click context menu is available on **Variables**. From the list, select **Add Variable**.

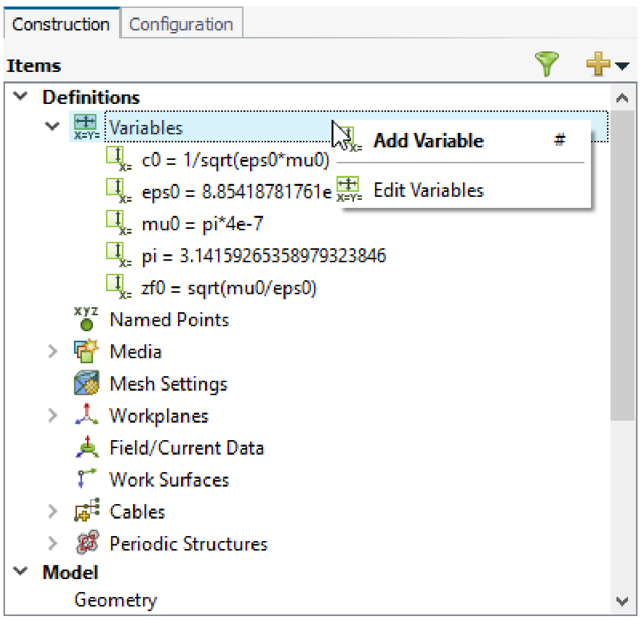


*Figure 160: The model tree (**Construction** tab).*

    - On the model tree, click the icon. From the drop-down list, select **Add Variable**.

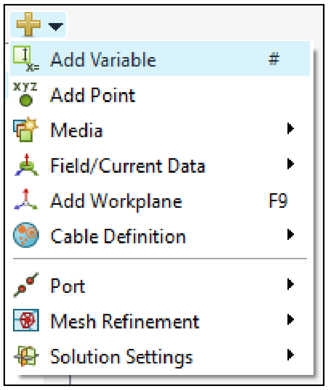


Figure 161: The drop-down list available in the model tree.

- Press # to use the keyboard shortcut.

2. Create the following variables:

Name	**Expression**
alpha	60
alpha_rad	*alpha**pi*/180
d	0.25
lambda	1
freq	*c0/lambda*

Tip:

- Click **Add** to keep the **Create Variable** dialog open and add more variables.
- Click **Create** to add a variable and close the **Create Variable** dialog.

7.4.4 Creating the Bent Dipole

Create the bent dipole using a polyline.

To illustrate the usage of custom workplanes, the bent dipole is created using a custom workplane.

1. On the **Construct** tab, in the **Create Curve** group, click the **Polyline** icon.

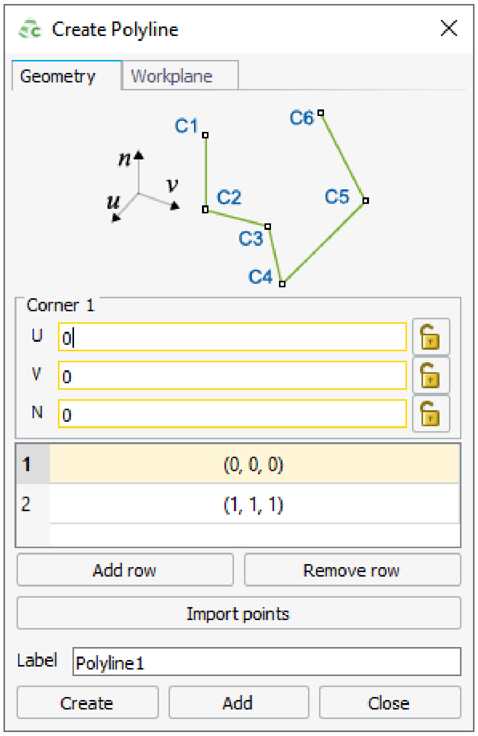


*Figure 162: The **Create Polyline** dialog showing the default values. Active fields are outlined in yellow.*

> **Note:** Default values are used on geometry creation dialogs to allow a preview in the 3D view. You may change the values as required.

> **Tip:** An active field allowing point-entry is indicated by a yellow outline.
>
> Point-entry allows a variable or named points to be entered by pressing Ctrl+Shift+left click on a variable or named point in the model tree.

2. Create a polyline.
   a) Under **Corner 1**, add the following coordinates:
      - Corner 1:
        - **U**: *lambda*/4*cos(*alpha_rad*)
        - **V**: 0
        - **N**: *lambda*/4*sin(*alpha_rad*)

   b) In the table, click on the second row to make **Corner 2** active. Add the following coordinates:

- Corner 2:
  - **U**: 0
  - **V**: 0
  - **N**: 0

c) Click **Add row** for **Corner 3**. Click on the third row to make **Corner 3** active. Add the following coordinates:

- Corner 3:
  - **U**: *lambda*/4*cos(*alpha_rad*)
  - **V**: 0
  - **N**: -*lambda*/4*sin(*alpha_rad*)

d) Set the **Label** to `Bent_dipole`.

3. Modify the origin and orientation of the polyline.

a) Click the **Workplane** tab.

b) Click **Custom workplane**.

c) Under **Origin**, in the **X field**, enter *d*.

d) Under **U Vector**, in the **X field**, enter *-1*.

4. Click **Create** to create the polyline and to close the dialog.

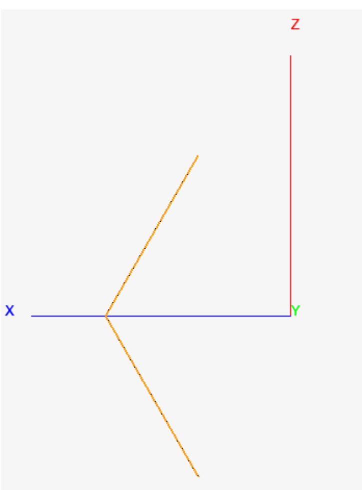


Figure 163: The bent dipole with corner point on the X axis.

7.4.5 Creating the Plate

Create the plate with a vertical orientation located at the origin using a rectangle.

1. On the **Construct** tab, in the **Create Surface** group, click the **Rectangle** icon.
2. Create a rectangle using the **Base centre, width, depth** definition method.
   a) **Base centre (C)**: (0, 0, 0)
   b) **Width (W)**: *lambda*
   c) **Depth (D)**: *lambda*
   d) **Label**: `Reflector`

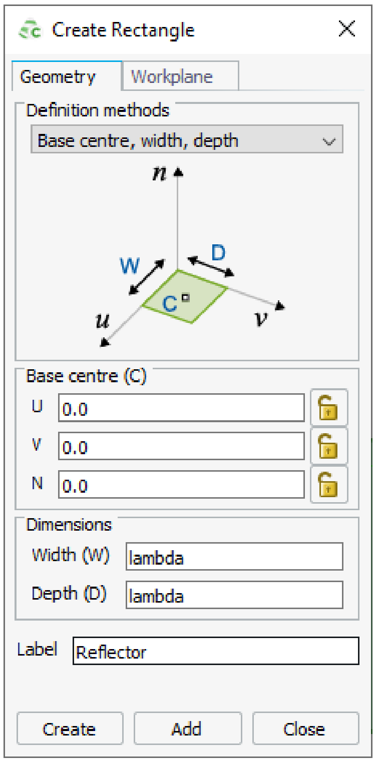


*Figure 164: The **Create Rectangle** dialog.*

3. Modify the orientation of the rectangle.
   a) On the **Create Rectangle** dialog, click the **Workplane** tab.
   b) Click **Predefine workplane**.
   c) From the drop-down list, select **Global YZ**.
4. Click **Create** to create the rectangle and to close the dialog.

7.4.6 Ports, Sources and Loads in CADFEKO

Voltage sources and discrete loads are applied to ports and not directly to the model geometry or mesh. A port must be defined before a source or load can be added.

Creating the Port

Define a wire port on the feed pin to excite the dipole. A voltage source will be added to this port.

> **Note:** A port is a mathematical representation of where energy can enter (source) or leave a model (sink). Use a port to add sources and discrete loads to a model.

1. Select the wire where the port is to be added.
   a) In the model tree, select **Bent_dipole**.
   b) In the details tree, select one of the wires of **Bent_dipole**.
2. Open the **Create Wire Port** dialog using one of the following workflows:
   - On the **Source/Load** tab, in the **Ports** group, click the **Wire Port** icon.
   - In the details tree, a right-click context menu is available on the wire. From the drop-down list, click **Create port** > **Wire Port**.
3. Under **Place port on**, click **Segment**.

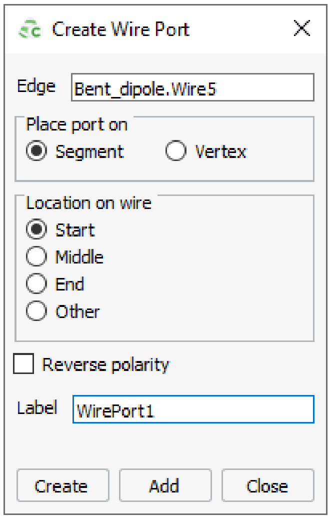


*Figure 165: The **Create Wire Port** dialog.*

4. Under **Location on wire**, select **End** to place the port on the X axis for the selected wire.
5. Click **Create** to create the port and to close the dialog.

Adding a Voltage Source

Add a voltage source to the port of the bent dipole.

1. On the **Source/Load** tab, in the **Sources on Ports** group, click the **Voltage Source** icon.
2. On the **Add Voltage Source** dialog, use the default settings.

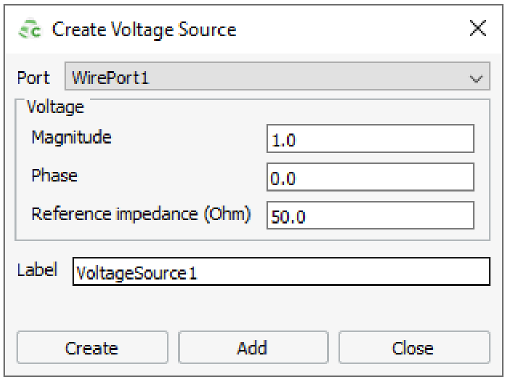


*Figure 166: The **Create Voltage Source** dialog.*

3. Click **Create** to create the voltage source and to close the dialog.

> **Note:** The **Configuration** tab was selected automatically when you defined the voltage source. You may also add sources, loads and set the frequency from here.

7.4.7 Setting the Simulation Frequency

Specify the frequency range of interest. For this example, a single frequency point is used.

1. On the **Source/Load** tab, in the **Settings** group, click the **Frequency** icon.

A variable was created at the beginning of the example that contains the solution frequency.

2. In the **Frequency (Hz)** field, enter *freq*.

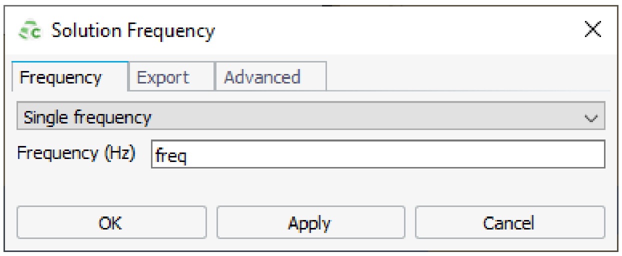


*Figure 167: The **Solution Frequency** dialog.*

3. Click **OK** to set the frequency and to close the dialog.

> **Note:** With the frequency set to *freq*, the actual frequency is 299.792 MHz. In the model tree, click the Configuration to view the specified simulation frequency.

7.4.8 Requesting Far Fields

Add a far field request (in the azimuth direction) to the model.

1. On the **Request** tab, in the **Solution Requests** group, click the **Far Fields** icon.
2. On the **Request Far fields** dialog, click **Horizontal cut (UV plane)**.

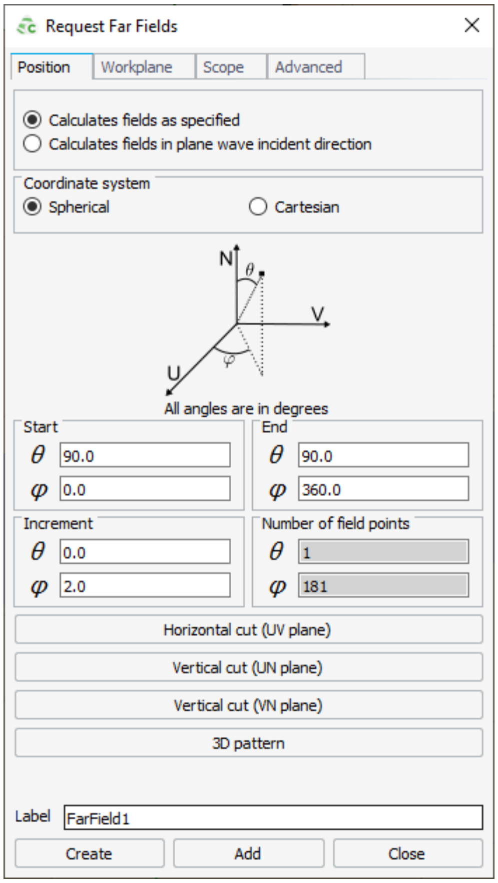


*Figure 168: The **Request Far Fields** dialog.*

3. Use the default label.
4. Click **Create** to create a far field request and to close the dialog.

7.4.9 Defining an Optimisation Search

Add an optimisation search to maximise the maximum gain in the azimuth direction.

1. On the **Request** tab, in the **Optimisation** group, click the **Add Search** icon.
2. On the **Create Optimisation Search** dialog, set the **Optimisation convergence accuracy** to **Normal (default)**.
3. Click **Create** to create the new optimisation search and to close the dialog.

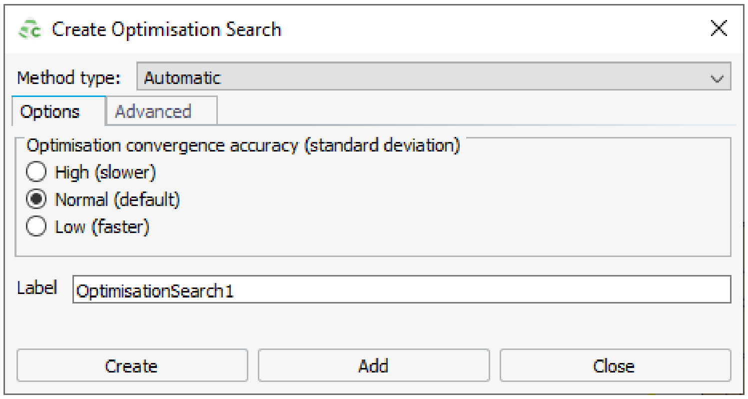


*Figure 169: The **Create Optimisation Search** dialog.*

7.4.10 Specifying the Optimisation Parameters

Specify the optimisation parameters. Choose from existing variables and specify their minimum and maximum values

1. In the model tree (**Construction** tab), select the relevant search. On the **Request** tab, in the **Optimisation** group, click the **Parameters** icon.
2. Populate the fields on the **Modify Optimisation Parameters** dialog as given in the table.

Variable	Min value	Max value	Start value
alpha	20	110	80
d	0.7	0.9	0.8

3. Click **OK** to set the parameters and to close the dialog.

7.4.11 Specifying the Far Field Goal

Specify the goal focus, the operations to perform on the goal, and the goal objective.

1. In the model tree, on the **Construction** tab, select **OptimisationSearch1**.
2. On the **Request** tab, in the **Optimisation** group, click the **Add Goal Function** icon. From the drop down list, select **Far Field Goal**.
3. Under **Goal focus**, specify the requested output from the Solver.
   a) Under **Focus source type**, select **Defined in CADFEKO**.
   b) In the **Focus source** field, from the drop-down list, select **FarField1**.
   c) In the **Focus type** field, from the drop-down list select **Gain**.
   d) In the **Polarisation** field, from the drop-down list select **Total**.
4. Under **Focus processing steps**, specify the processing to be performed prior to comparing with the objective.
   a) In the **Operation** column, from the drop-down list, select **Max**.
5. Under **Goal operator**, specify how the objective and focus is compared.
   a) In the **Operator type** drop-down list, select **Maximise** to maximise the gain.
      - The **Goal objective** is the value or values with which the focus is compared. For maximisation and minimisation the **Goal objective** is hidden.
      - For multiple goals, the **Weight** will determine the contribution of the particular goal to the global error function during the fitness evaluation.

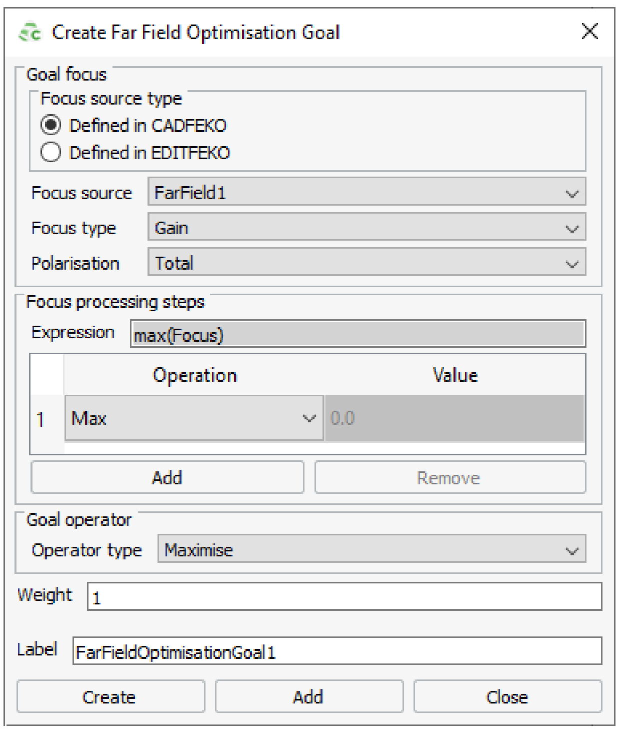


*Figure 170: The **Create Far Field Goal** dialog.*

6. Click **Create** to create the new far field goal and to close the dialog.

7.4.12 Modifying the Auto-Generated Mesh

When the frequency is set or local mesh settings are applied to the geometry, the automatic mesh algorithm calculates and creates the mesh automatically while the GUI is active using default mesh settings. When required, these mesh settings may be modified.

1. Open the **Modify Mesh Settings** dialog using one of the following workflows:
   - On the **Mesh** tab, in the **Meshing** group, click the **Create Mesh** icon.
   - Press Ctrl+M to use the keyboard shortcut.
2. On the **Modify Mesh Settings**, set the **Mesh size** to **Coarse**.
3. Set the **Wire segment radius** to `0.001`.

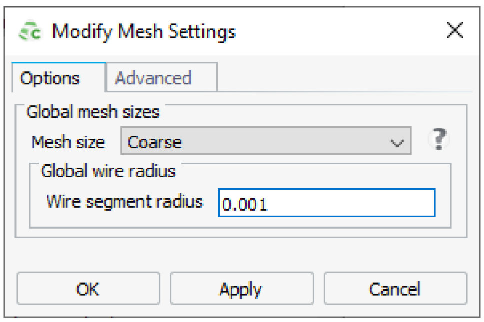


Figure 171: The ***Modify Mesh Settings*** *dialog.*

4. Click **OK** to create the mesh and to close the dialog.

7.4.13 Saving the Model

Save the model to a CADFEKO `.cfx` file.

1. Save the model using one of the following workflows:
    - On the **Home** tab, in the **File** group, click the **Save** icon.
    - Press Ctrl+S to use the keyboard shortcut.
2. Save the model as `Dipole_Optimisation.cfx`.
3. Click **Save** to close the dialog.

7.5 Launching the Solver

Launch the Solver to calculate the results. No requests were added to this model since impedance and current information are calculated automatically for all voltage and current sources in the model.

1. Launch the Solver using one of the following workflows:
    - On the **Solve/Run** tab, in the **Run/Launch** group, click the **Feko Solver** icon.
    - On the application launcher toolbar, click the **Feko Solver** icon in the group.
    - Press Alt+4 to use the keyboard shortcut.

If the model contains unsaved changes, the **Save Model** dialog is displayed.

2. Click **Yes** to save the model and to close the **Save Model** dialog.

The Feko Solver is launched and the **Executing runfeko** dialog is displayed. The dialog gives step-by-step feedback as the simulation progresses.

3. Click **Details** to expand the **Executing runfeko** to view the step-by-step feedback.

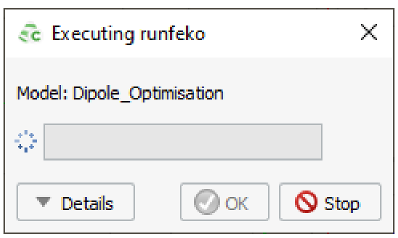


*Figure 172: The **Executing runfeko** dialog.*

7.6 Reviewing POSTFEKO and Launching OPTFEKO

Open POSTFEKO from within CADFEKO.

Use one of the following workflows to launch POSTFEKO:

- On the **Solve/Run** tab, in the **Run/Launch** group, click the **POSTFEKO** icon.
- On the application launcher toolbar, click the **POSTFEKO** icon in the group.
- Press Alt+3 to use the keyboard shortcut.

POSTFEKO opens by default with a single 3D view containing the model geometry.

7.6.1 Setting Up POSTFEKO to View Optimisation Progress

Create a 3D view with the far field results as well as two Cartesian graph to view the distance parameter and far field goal.

When POSTFEKO is opened, the model is displayed in a single 3D view.

1. Display the far field in the 3D view.
   a) On the **Home** tab, in the **Add results** group, click the **Far field** icon.
   b) From the drop-down list select **FarField1**.

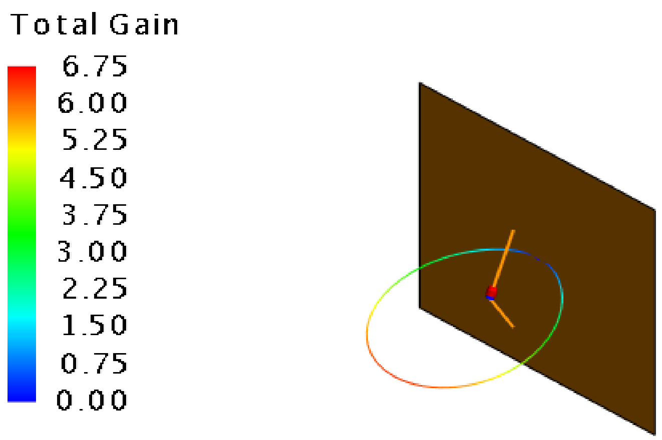


Figure 173: The far field result for the bent dipole and plate.

2. Add a Cartesian graph and view the optimised parameter, *alpha*.
   a) On the **Home** tab, in the **Create new display** group, click the **Cartesian** icon.
   b) On the **Home** tab, in the **Add results** group, click the **Optimisation** icon. From the drop-down list, select **Optimisation**.
   c) In the result palette, in the **Trace** list, select **alpha**.

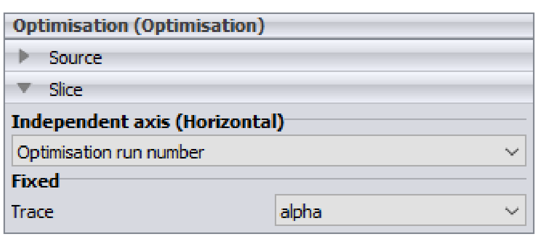


*Figure 174: The **Optimisation** panel in the result palette.*

3. Duplicate the first graph (to create a second graph) to view the optimised parameter, *d*.

a) On the **Cartesian** context tab, on the **Display** tab, in the **Duplicate** group, click the **Duplicate view** icon.

b) In the result palette select the trace, **Optimisation**.

c) In the result palette, in the **Trace** field, select **d**.

4. Duplicate the first graph (to create the third graph) and view the far field goal versus optimisation run number.

   a) On the **Cartesian** context tab, on the **Display** tab, in the **Duplicate** group, click the **Duplicate view** icon.

   b) In the result palette select the trace, **Optimisation**.

   c) In the result palette, in the **Trace** field, select **optimisationsearch1.goals.farfieldoptimisationgoal1**.

5. [Optional] Arrange (tile) the four windows to view the multiple windows at once.

   a) On the **View** tab, in the **Window**, click the **Tile** icon.

7.6.2 Launching OPTFEKO

Run OPTFEKO and view the output of the optimisation progress.

The following steps are performed in POSTFEKO, but the optimiser can also be launched from CADFEKO

1. On the **Home** tab, in the **Run/Launch** group, click the **OPTFEKO** icon.

> **Note:** The **Executing optfeko** dialog is displayed in a condensed format.
>
> Click **Details** to view any problems encountered during the optimisation process.

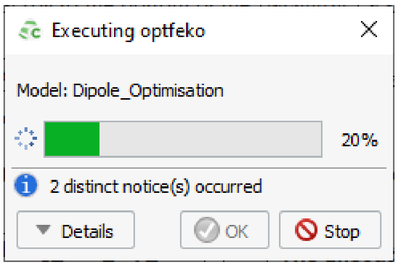


Figure 175: The Executing optfeko dialog in condensed format.

2. Scroll to the bottom of the **Executing optfeko** window output to view the convergence information, as well as the optimal parameters.

> **Note:** Sensitivity information is included if sufficient data is available for the analysis.

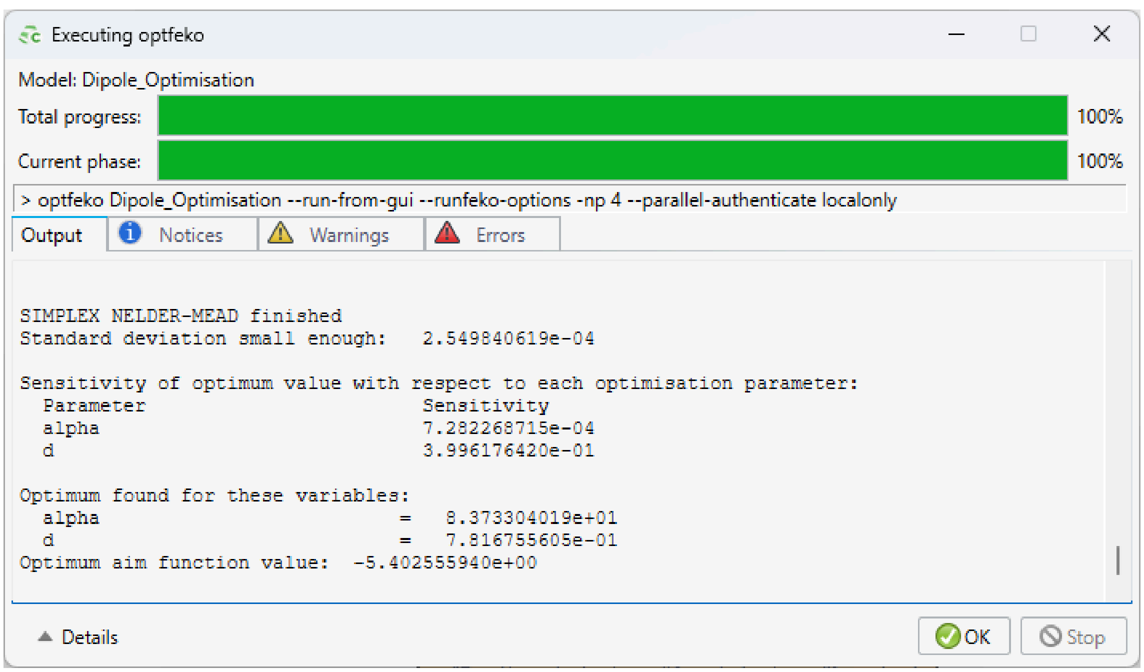


Figure 176: The Executing optfeko dialog with details.

The **Executing optfeko** window displays the optimum values as follows:

- *alpha*: 83.733°
- *d*: 0.782 meters

7.6.3 Viewing the Optimisation Results

View the optimisation results obtained by OPTFEKO.

The graphs have already been configured to view the progress of the optimisation process.

View the final results of OPTFEKO for the optimisation parameters on the previously defined 3D view and 2D graphs.

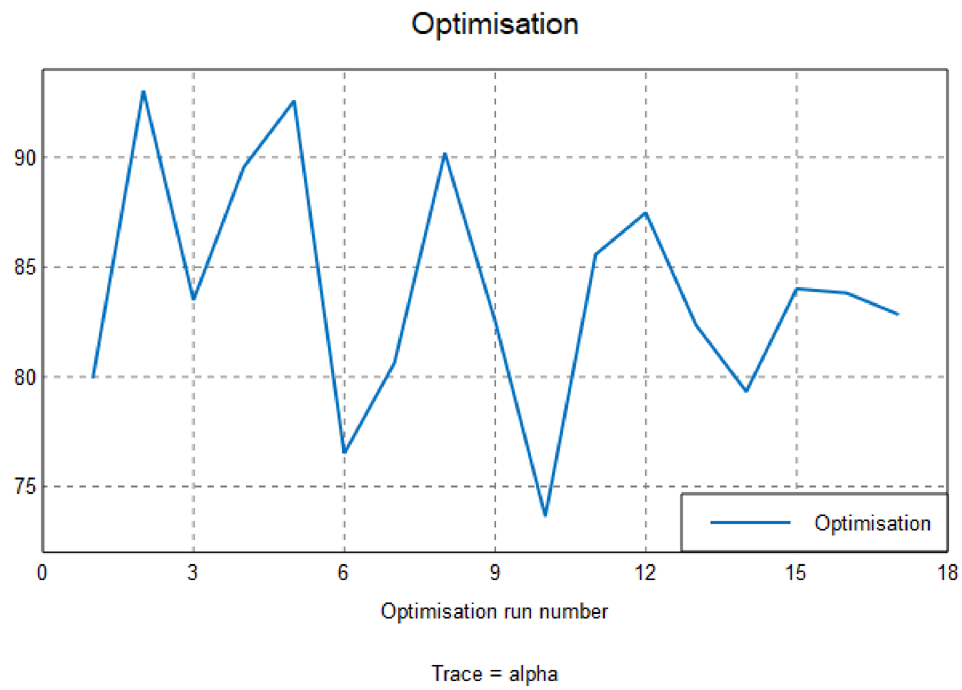


Figure 177: The optimisation run number on a Cartesian graph.

> **Note:** For more details on the optimisation process, view the log file, `Dipole_Optimisation.log`, that was created in the same directory as the current model.

OPTFEKO creates multiple CADFEKO (.cfx) models for each iteration, located in the same directory as the current model. For example:

```
Dipole Optimisation_opt_1.cfx
Dipole Optimisation_opt_2.cfx
Dipole Optimisation_opt_3.cfx
.
.
Dipole Optimisation_opt_17.cfx
```

including the model with the variables set to the optimum values (indicated by the `_optimum` file extension).

```
Dipole_Optimisation_optimum.cfx
```

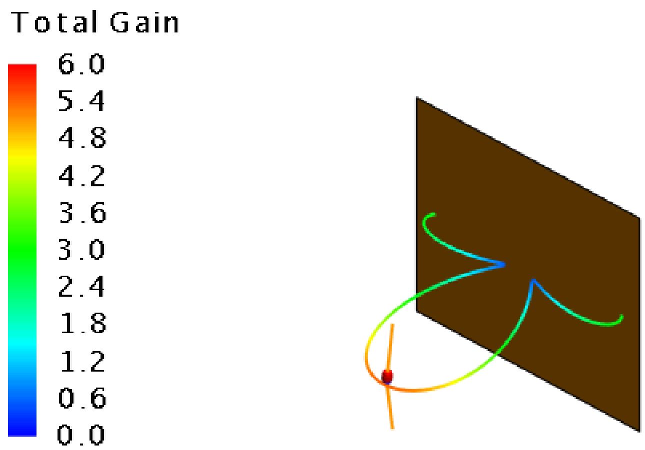


Figure 178: The optimised far field for the bent dipole and plate.

7.7 Closing Remarks

This example showed the construction and optimisation of a bent dipole in front of a plate.

Many concepts were introduced in this simple example that are applicable to models commonly created in CADFEKO. This example has demonstrated how to configure a CADFEKO model as well as how optimisation in CADFEKO is executed. The optimisation process as well as the optimum values for the model parameters were displayed in POSTFEKO, but can also be viewed in the `.log` file.

Index

Special Characters

Numerics

A

B

C

L

M

N

R

S

T

U

V

W

Z