ALTAIR

ONLY FORWARD

Altair® AcuSolve® 2025

Tutorials

Updated: 11/21/2024

Introduction to AcuSolve Tutorials

This guide contains a collection of simulation cases with explicit instructions for setting up, solving and post-processing. A range of tutorials are included to illustrate the basic AcuSolve workflow and to provide guidance for setting up a variety of problems.

Some tutorials can be completed with a choice of pre-processor. Therefore, two sets of tutorials are provided here: one for HyperMesh CFD and one for SimLab.

Objectives

The objectives of the AcuSolve tutorials are to:

- Present the basic AcuSolve workflow.
- Introduce the capabilities of AcuSolve to new users.
- Provide guidance for the use of AcuSolve with industrial applications.

Prerequisites

No experience with AcuSolve is needed prior to running through the tutorials included in this guide. It is expected that you have some exposure to basic principles of engineering and computer-aided engineering.

For the HyperMesh CFD based tutorials, it is strongly recommended that you go through the introductory tutorial ACU-T: 1000 UI Introduction before proceeding to the other tutorials. This tutorial includes the steps to import the geometry model, define surface mesh controls and boundary layer parameters, and so on, which are not included in the subsequent tutorials. For these remaining tutorials, you will be provided a HyperMesh database (`.hm` file) which has the pre-defined mesh controls. Instructions related to post-processing have been included in all the tutorials.

For the SimLab based tutorials, it is recommended that you go through the introductory tutorial ACU-T: 1000 UI Introduction before proceeding to the other tutorials. This tutorial provides an overview of the panels, ribbons, and toolbars in the SimLab user interface followed by demonstration of the CFD workflow. All of the SimLab tutorials start from importing the CAD geometry, generating mesh, and setting up the simulation to post-processing the results. All of the tutorials are video-based and you need to have an internet connection to view the videos and to download the files.

Basic Workflow

HyperMesh CFD Based Tutorials

The basic workflow for HyperMesh CFD based tutorials consists of the following tasks:

1. Problem description
2. Start HyperMesh CFD and open the HyperMesh database
3. Validate the model
4. Set up the simulation parameters and solver settings
5. Assign material properties to the fluid and solid regions
6. Define any porous media, body force, and/or reference frames
7. Assign flow boundary conditions, such as inlet, outlet, slip, symmetry, and so on
8. Define any radiation parameters
9. Define any mesh motion and mesh boundary conditions
10. Define surface mesh controls, boundary layer and volume mesh controls, and generate the mesh
11. Define nodal output frequencies, surface and volume monitors, and nodal initial conditions
12. Compute the solution using AcuSolve
13. Post-process results with the plot utility (to visualize data related to the progress of the calculations) and/or with HyperView to visualize simulation data with contours, vectors, streamlines, and the like

SimLab Based Tutorials

The basic workflow for SimLab based tutorials consists of the following tasks:

1. Problem description
2. Learning outcome
3. Define surface mesh controls, boundary layer and volume mesh controls, and generate the mesh
4. Select the physics to be solved
5. Assign material properties to the fluid and solid regions
6. Define any porous media, body force, and/or reference frames
7. Assign flow boundary conditions, such as inlet, outlet, slip, symmetry, and so on
8. Define any radiation parameters
9. Define any mesh motion and mesh boundary conditions
10. Set up the simulation parameters and solver settings
11. Define nodal output frequencies, surface and volume monitors, and nodal initial conditions
12. Compute the solution using AcuSolve
13. Post-process results (to visualize data related to the progress of the calculations and/or to visualize simulation data with contours, vectors, streamlines, and the like)
14. Summary

> **Note:** Validation (comparison with experimental results) and verification (determination of a mesh independent solution) approaches are not covered in this manual. Refer to the *AcuSolve Validation Manual* for validation cases.

For tutorials that illustrate more complicated problems, such as those involving transient physics or mesh motion, details of additional tasks are provided in the tutorial.

Supporting Files

Supporting files for each HyperMesh CFD tutorial are located in a compressed `.zip` archive.

Download the following and save its contents in a local directory. You can also download individual files directly from each tutorial.

HyperMeshCFD_tutorial_inputs.zip

For SimLab, you can download the files using the links provided on the page for each tutorial.

Typographical Conventions Used in this Manual

Different type styles are used to indicate workspace items and inputs, for example:

Set the Turbulence equation to **Spalart Allmaras**.
Enter `Symmetry` as the name of the surface.

A mini flow chart is used to indicate menu selections that lead to a specific command or dialog box, for example,

Open **View** > **Defined Views** and set the view to **+Z**.

This indicates that the **View** menu is clicked, then the **Defined Views** menu item is selected, and that the button labeled **+Z** is clicked when the dialog opens.

A mini flow chart is also used to indicate items in the Data Tree to be expanded, for example,

Expand the **Model** > **Surfaces** > **Small Inlet** tree item.

This indicates that first the **Model** tree item is expanded, then the **Surfaces** item underneath it is expanded, and finally, the **Small Inlet** item under **Surfaces** is expanded.

Contents

Introduction to AcuSolve Tutorials ii

User Interface Introduction 14

ACU-T: 1000 UI Introduction 15
HyperMesh CFD 15
SimLab 43

Flow 44

ACU-T: 2000 Turbulent Flow in a Mixing Elbow 45
HyperMesh CFD 45
SimLab 60
ACU-T: 2100 Turbulent Flow Over an Airfoil Using the SST Turbulence Model 61
HyperMesh CFD 61
SimLab 94
ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building 96
HyperMesh CFD 96
SimLab 111
ACU-T: 2400 Supersonic Flow in a Converging-Diverging Nozzle 112
HyperMesh CFD 112
SimLab 126
ACU-T: 2500 Floating Surface Monitor 127
SimLab 127

Heat Transfer 128

ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection 129
SimLab 129
ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow 130
HyperMesh CFD 130
SimLab 148
ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow 149
HyperMesh CFD 149
SimLab 163
ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping 164
HyperMesh CFD 164
SimLab 187
ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model 189
HyperMesh CFD 189
SimLab 212

ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial 213
HyperMesh CFD 213
SimLab 241
ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model 242
HyperMesh CFD 242
SimLab 255
ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model 257
HyperMesh CFD 257
SimLab 270
ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model 272
HyperMesh CFD 272
SimLab 304
ACU-T: 3300 Modeling of a Heat Exchanger Component 305
HyperMesh CFD 305
SimLab 320
ACU-T: 3310 Single Phase Nucleate Boiling 321
HyperMesh CFD 321
SimLab 334
ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model 335
HyperMesh CFD 335
SimLab 350
ACU-T: 3400 AcuSolve -Flux Integration 352
SimLab 352
ACU-T:3401 Electric Motor Analysis using AcuSolve - Flux One-Way Coupling 353
SimLab 353
ACU-T: 3500 Electric Potential - Automotive Fuse 354
SimLab 354
ACU-T: 3510 Battery Thermo – Electric: 2S2P Module 355
SimLab 355
ACU-T: 3511 Battery Thermo-Electric Pouch Cell 356
SimLab 356
ACU-T: 3512 Battery Thermal Runaway 357
SimLab 357
ACU-T: 3513 ARC-Based Thermal Runaway: 1S1P Module 358
SimLab 358
ACU-T: 3600 Melting of Diesel Exhaust Additive within an Enclosed Tank 359
HyperMesh CFD 359

Multiphase 378

ACU-T: 4000 Dam Break Simulation 379
HyperMesh CFD 379
SimLab 392
ACU-T: 4001 Water Filling in a Tank 393
HyperMesh CFD 393

SimLab 407
ACU-T: 4002 Sloshing of Water in a Tank 409
HyperMesh CFD 409
SimLab 421
ACU-T: 4003 Freely Falling Water Droplet 423
HyperMesh CFD 423
SimLab 440
ACU-T: 4100 Multiphase Flow using Algebraic Eulerian Model 442
HyperMesh CFD 442
SimLab 453
ACU-T: 4101 T-Junction Flow using the Eulerian Multiphase Model 455
HyperMesh CFD 455
SimLab 469
ACU-T: 4102 Fluidized Bed using the Granular Multiphase Model 470
HyperMesh CFD 470
SimLab 486
ACU-T: 4200 Humidity – Pipe Junction 487
HyperMesh CFD 487
SimLab 501
ACU-T: 4201 Condensation & Evaporation - Air Box 502
HyperMesh CFD 502
SimLab 521
ACU-T: 4300 Species Transport Modeling 523
HyperMesh CFD 523
SimLab 538
ACU-T: 4400 Mixing Flows Using the Gas Kinetics Model 540
SimLab 540

Moving Body 541

ACU-T: 5000 Blower - Steady (Rotating Frame) 542
HyperMesh CFD 542
SimLab 558
ACU-T: 5001 Blower - Transient (Sliding Mesh) 560
HyperMesh CFD 560
SimLab 573
ACU-T:5002 Centrifugal Blower – Hybrid (MRF + MM) 575
SimLab 575
ACU-T: 5100 Modeling of a Fan Component Using the Fan Component - Coefficient Method 576
HyperMesh CFD 576
SimLab 598
ACU-T: 5101 Modeling of a Fan Component Using the Fan Component - PQ Method 599
HyperMesh CFD 599
ACU-T: 5200 Rigid-Body Dynamics of a Check Valve 619
HyperMesh CFD 619
SimLab 650

ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve.....651
HyperMesh CFD.....651
SimLab.....683
ACU-T: 5202 Flow Closing Valve.....684
SimLab.....684
ACU-T: 5400 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (P-FSI).....685
SimLab.....685
ACU-T: 5403 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction.....686
HyperMesh CFD.....686
SimLab.....710

Particle Tracing.....712

ACU-T: 6000 Static Mixer Simulation – AcuTrace.....713
HyperMesh CFD.....713
ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM.....743
HyperMesh CFD.....743
SimLab.....775
ACU-T: 6101 Particle Separation in a Windshifter using AcuSolve - EDEM Unidirectional Coupling.....776
HyperMesh CFD.....776
SimLab.....817
ACU-T: 6102 Particle Separation in a Windshifter using AcuSolve - EDEM Bidirectional Coupling.....818
HyperMesh CFD.....818
SimLab.....862
ACU-T: 6103 AcuSolve - EDEM Bidirectional Coupling with Heat Transfer.....863
HyperMesh CFD.....863
SimLab.....900
ACU-T: 6104 AcuSolve - EDEM Bidirectional Coupling with Non-Spherical Particles.....901
HyperMesh CFD.....901
SimLab.....936
ACU-T: 6105 Single Particle Sedimentation – Effect of Lift and Torque.....937
HyperMesh CFD.....937
SimLab.....961
ACU-T: 6106 AcuSolve - EDEM Bidirectional Coupling with Mass Transfer.....962
HyperMesh CFD.....962
SimLab.....989
ACU-T: 6107 AcuSolve - EDEM Bidirectional 3-Phase Coupling.....990
SimLab.....990
ACU-T: 6108 AcuSolve - EDEM Bidirectional Coupling with User-Defined Drag Model.....991
HyperMesh CFD.....991

Porous Medium.....1015

ACU-T: 6500 Flow Through Porous Medium....1016
HyperMesh CFD....1016
SimLab....1031
ACU-T: 6501 Flow Through Porous Medium with Physical Velocity....1032
HyperMesh CFD....1032

Optimization....1045

ACU-T: 7200 Topology Optimization....1046
SimLab....1046
ACU-T: 7201 Topology Optimization with Flow Distribution....1047
SimLab....1047
ACU-T: 7202 CFD Topology Optimization of Multiple Flow Paths....1048
SimLab....1048
ACU-T: 7300 DOE using Inspire, SimLab and HyperStudy....1049
SimLab....1049
ACU-T: 7301 Solution DOE....1050
SimLab....1050
Intellectual Property Rights Notice....1051
Technical Support....1057

Index....1058

User Interface Introduction

1

This chapter covers the following:

- ACU-T: 1000 UI Introduction (p. 15)

ACU-T: 1000 UI Introduction

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up a Computational Fluid Dynamics (CFD) analysis using Altair HyperMesh CFD. HyperMesh CFD is a powerful tool which provides a single, streamlined platform for performing a CFD analysis, starting from importing CAD through post-processing results. In this tutorial, you will learn how to use HyperMesh CFD for setting up a CFD analysis while exploring different capabilities available within the software for importing a geometric model, validating the geometry, setting up simulation parameters and boundary conditions, and generating a mesh. You will then launch AcuSolve simulation directly from HyperMesh CFD and post-process the results using HyperMesh CFD Post.

Prerequisites

To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T1000_manifold.x_t`

Analyze the Problem

An important step in any CFD simulation is to examine the engineering problem at hand and determine the important parameters that need to be provided to AcuSolve. Parameters can be based on geometrical elements, such as inlets, outlets, or walls, and on flow conditions, such as fluid properties, velocity, or whether the flow should be modeled as turbulent or as laminar.

The system being simulated here is a manifold pipe, analogous to an inlet manifold in an engine. An inlet manifold distributes the incoming flow to multiple outputs. As can be seen in the image below, the pipe has a single inlet and multiple outlets, thus distributing a fraction of the flow among each outlet. Ideally in an inlet manifold used in an engine, the manifold design is such that it ensures near-equal distribution of flow among all the outlets. However, the geometry being used here is purely a demonstration case and not an optimized manifold geometry.

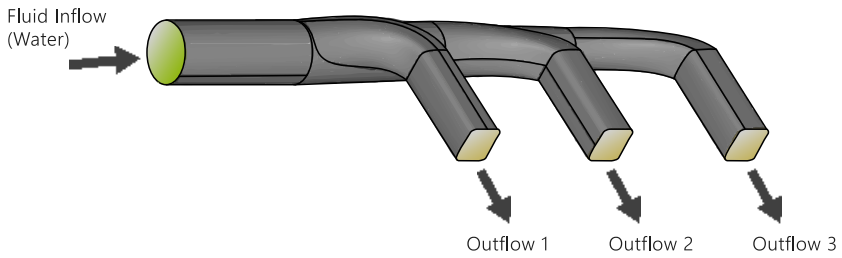


Figure 1: Schematic of the Problem

Introduction to HyperMesh CFD

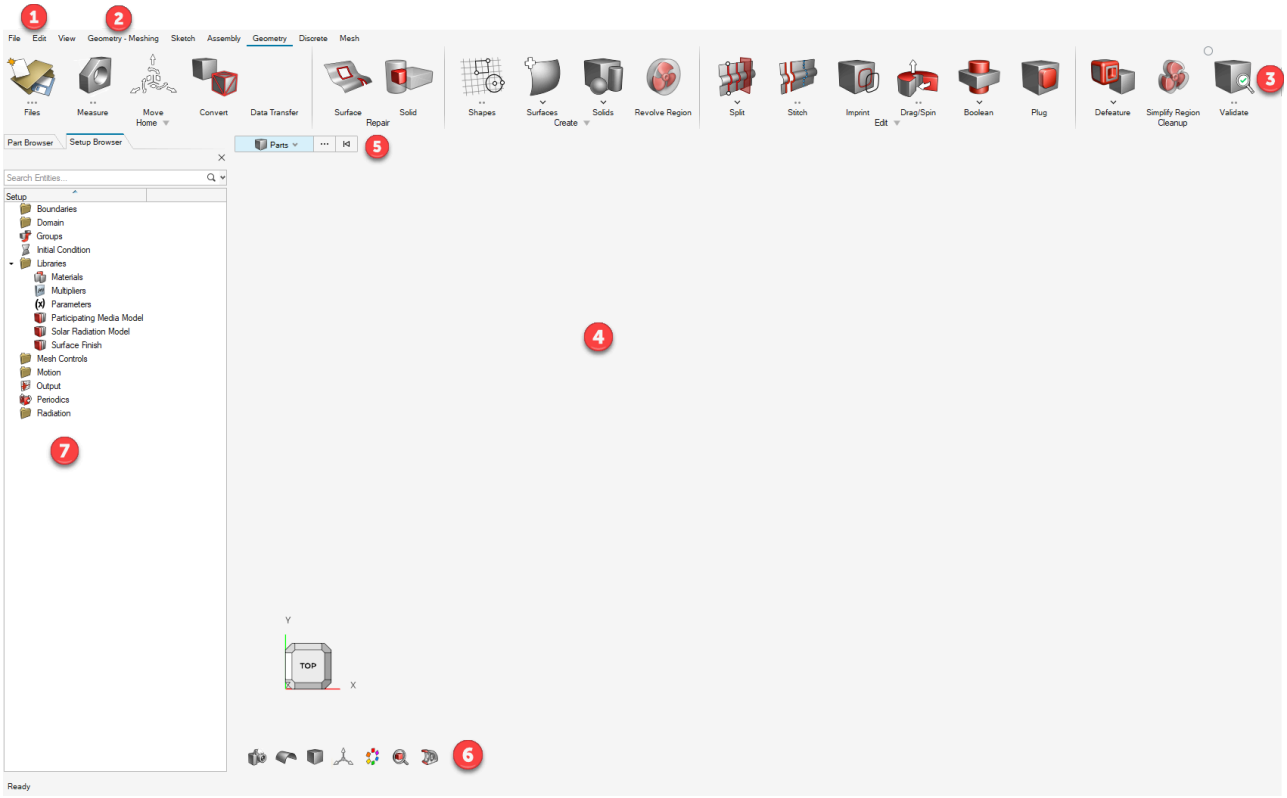


Figure 2:

The HyperMesh CFD graphical user interface can be divided into seven general categories as shown in the figure above.

1. The menu bar contains the drop-down menus for File input-output, Edit, and View operations.
2. The modeling environment switcher is used to populate relevant ribbons and tools based on modeling tasks.
3. Ribbons contain various functionalities and tools available in HM-CFD. Navigate between various ribbons by clicking the ribbon tabs to the right of the modeling environment switcher. After

selecting a ribbon, the corresponding tool icons are displayed on the screen. The functionalities of various ribbons and corresponding tools are briefly explained in this section.

Sketch ribbon

The Sketch ribbon provides multiple ways to define sketching plane by selecting planes aligned with global coordinate systems. You can also get references of existing geometry by cutting geometry or projecting geometry to a plane. It provides capabilities to create sketches using multiple interactive tools. A dimensioning capability enables you to parametrize dimensions of sketches. One of big use case for CFD is to get cutting lines of rotating geometry and create a non–cylindrical axisymmetric MRF region around them.

Figure 3:

Geometry ribbon

The Geometry ribbon consists of tools for repairing, creating, editing, and validating the geometry.

When a geometry file is imported, the Repair tools can be used to detect any defects present in the CAD model like intersections, free edges, duplicates, sliver surfaces, and so on and fix those errors.

The tools available under the Create sub-section can be used to create geometric entities like points, lines, surfaces, and solids.

The tools required for performing operations like plugging cavities, stitching surfaces, and so on are available under the Edit sub-section.

The Defeature tool can be used to resolve defects or model a new geometry, while the Validate tool can be used to detect any defects present in the CAD model. This process is usually known as CAD cleanup.

Figure 4:

Discrete ribbon

The Discrete ribbon consists of tools used for working with FE geometry. You can cap openings, connect geometry, define local or proximity-based wrap controls, enclose the model, remesh the enclosed results, and fix the mesh quality.

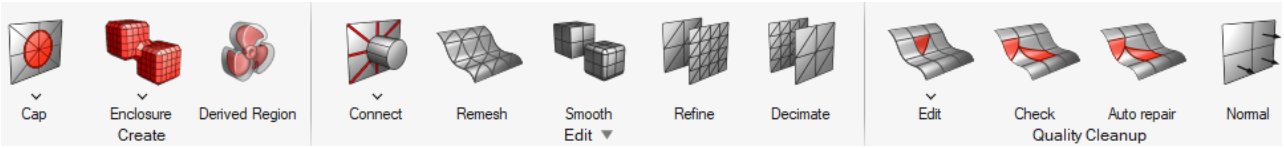


Figure 5:

Assembly ribbon

The Assembly ribbon is useful for finding, managing, and organizing the parts in your model.

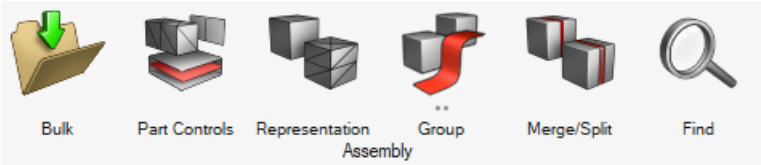


Figure 6:

Flow ribbon

The Flow ribbon contains tools for setting up simulation parameters, solver settings, and reference properties such as material properties, heat sources, porous media, and so on. The Setup sub-section is where you set up the physics equations and solver settings as well as create material models, multiplier functions, and parameters.

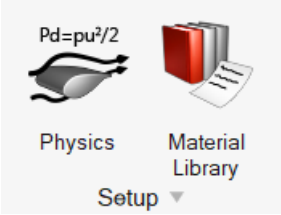


Figure 7:

The Domain sub-section contains tools for assigning reference properties such as materials, heat and momentum sources, and reference frames to volumes.

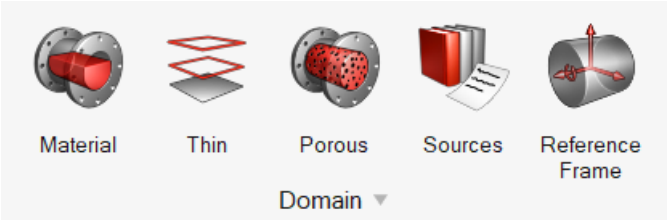


Figure 8:

Surface boundary conditions such as inlets, outlets, and far fields, can be assigned using the tools under the Boundaries sub-section. By default, all the surfaces are assigned a boundary condition of type 'auto_wall' and are placed under Default wall. Refer to the AcuSolve Surface Processing manual for more information about auto_wall. As you assign boundary conditions to the surfaces, they are moved into the respective group.

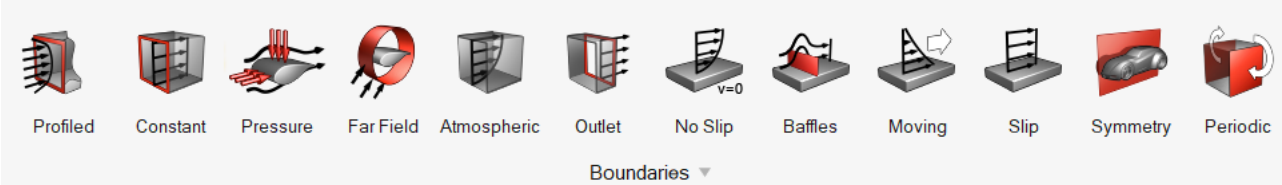


Figure 9:

Radiation ribbon

The Radiation ribbon is where you define radiation physics, create thermal, solar, and participating media models, and apply radiation parameters.

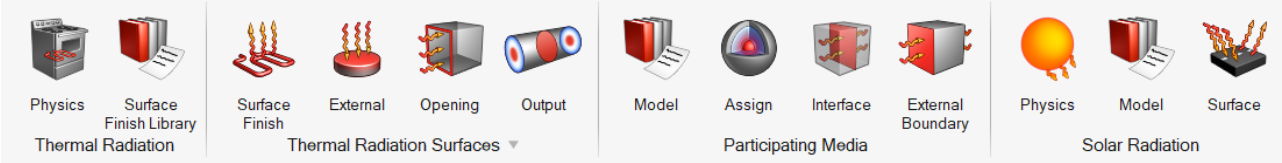


Figure 10:

Motion ribbon

Mesh boundary conditions and mesh-motion-related parameters can be defined using the tools available in the Motion ribbon. Parameters such as mesh motion type and mesh displacement constraints can be defined here. In addition to the mesh boundary conditions, code coupling with external codes such as OptiStruct and MotionSolve can be defined here

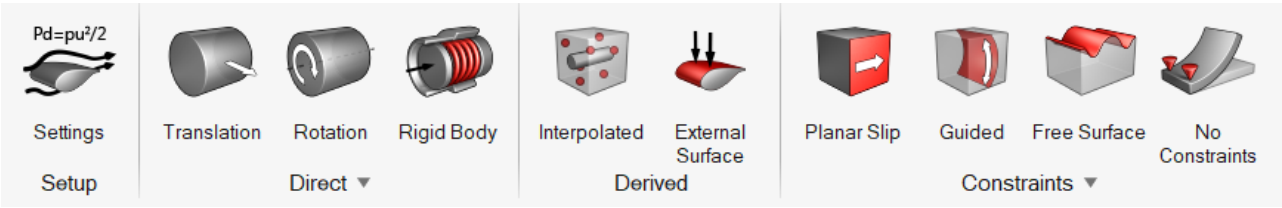


Figure 11:

AcuTrace ribbon

The AcuTrace ribbon computes particle traces as a series of segments solving particle motion. It computes traces for unsteady as well as steady flow fields, for flows with mesh motion as well as without, and for flows computed on meshes with interface surfaces. To solve a problem with AcuTrace, you must first run AcuSolve. You can set up for finite mass and massless particles.

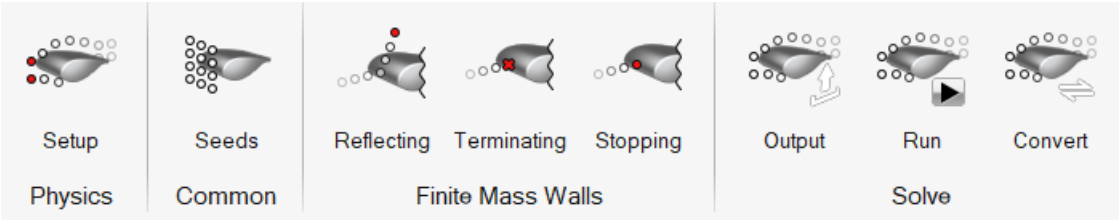


Figure 12:

Mesh ribbon

Meshing parameters such as surface mesh controls, boundary layer parameters, volume mesh parameters, and zone meshing parameters can be defined here. This ribbon also has tools for local remeshing. Once all the mesh controls are defined, you can generate the mesh using the Volume tool.

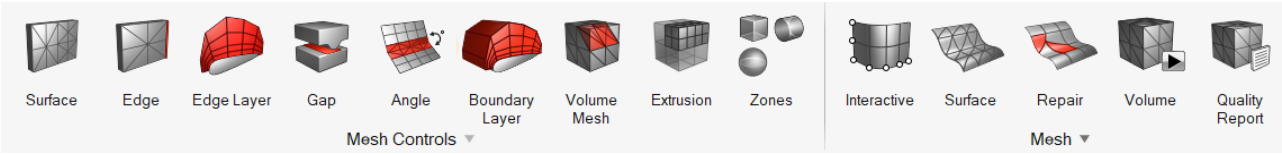


Figure 13:

Aerodynamics and Aeroacoustics Setup

The setup ribbon is used carry out external aerodynamics and fan noise simulations with the ultraFluidX solver.

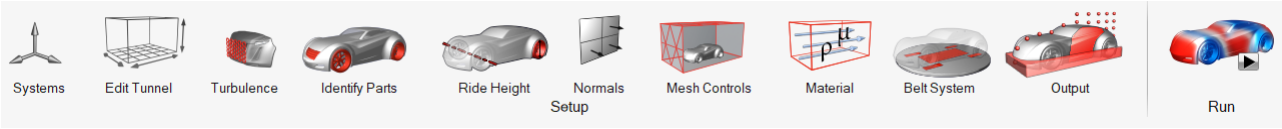


Figure 14:

Solution ribbon

The Solution ribbon is used to set up monitors for any individual point, surface, or volume set output. The Field tool is used to set the nodal output frequency for the entire model. The Initialize tools are used to set the nodal initial conditions for variables like pressure, velocity, and variables specific to each turbulence model.

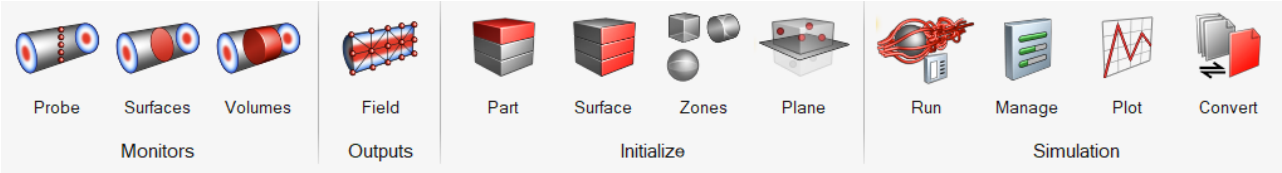


Figure 15:

Once the complete set up is done, the Run tool is used to launch AcuSolve. Once the AcuSolve run parameters are set, the simulation can be started, and you can monitor the status of the run using the run manager.

Post ribbon

The Post ribbon is where you can post-process the results. The Visualize tools under the Post ribbon can be used to create things like plots, streamlines, iso surfaces, and section cuts. The Measures tools can be used to probe variables at desired locations.

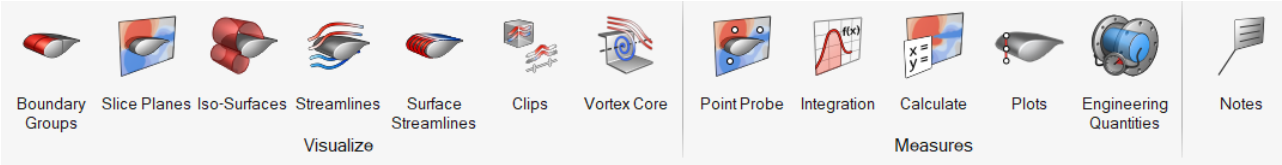


Figure 16:

Morphing Ribbon

The Morphing ribbon is used to morph mesh or FE geometry. You can define constraints to fix some nodes or define relative movement.
Once morphing is done, you can create shapes and design variables using the “Shape” tool and submit DOE studies using “DOE Study” tool.

Figure 17:

4. The modeling window is where the model is displayed. The model display can be manipulated using the view controls shown in the table below. Clicking on the model will highlight the entity being selected and right-clicking on an entity will give you additional options for the operations that can be done based on the context. Some of the functions available using right-click are Show, Hide, Isolate, Select, Advanced select, Create groups, and so on.

Button	**Operation**
Middle mouse scroll	Zoom in and out
Right-click hold and drag	Pan the model
Middle mouse click hold and drag	Rotate the model
Left-click	Select entity
Ctrl + Left-click	Select multiple entities
Left-click hold and drag	Window select
Shift + Left-click	Deselect entities

5. The entity selector enables you to control what entities can be selected using the left-mouse button. The selector can be set to any of the entities shown in the figure below. When you open any tool, the selector is automatically set to the entity (entities) which are appropriate for that command.

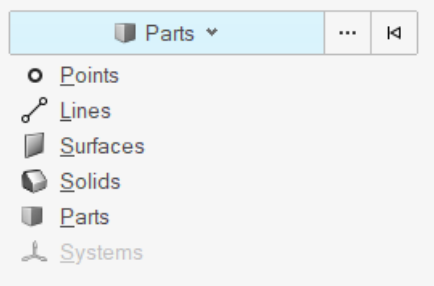


Figure 18:

6. The visualization of the model can be controlled using the tools available in View Controls toolbar. The display of mesh, model coloring, section cuts, standard views, and so on can be controlled using these tools.

Figure 19:

7. The browsers show the entities and setup parameters in the model and list them in a tree structure. They can be turned on or off from the View menu. Some common functions that can be performed in all browsers are show, hide, and isolate.

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.

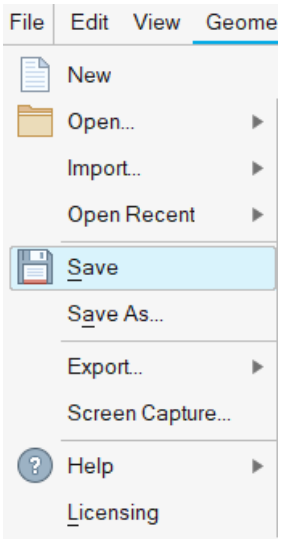


Figure 20:

- From the Home tools, Files tool group, click the **Save As** tool.

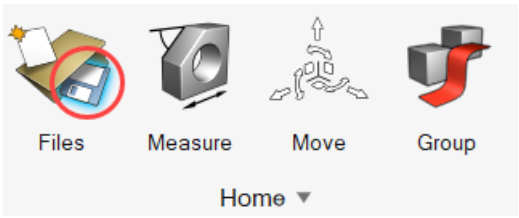


Figure 21:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database. This will be your problem directory and all the files related to the simulation will be stored in this location.
4. Enter a name for the database (`Manifold`, for example) and then click **Save**.

Import and Validate the Geometry

Import the Geometry

Before you start this step, copy the input file, `ACU-T1000_manifold.x_t`, to your problem directory.

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory and then select **ACU-T1000_manifold.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged and then click **Import**.

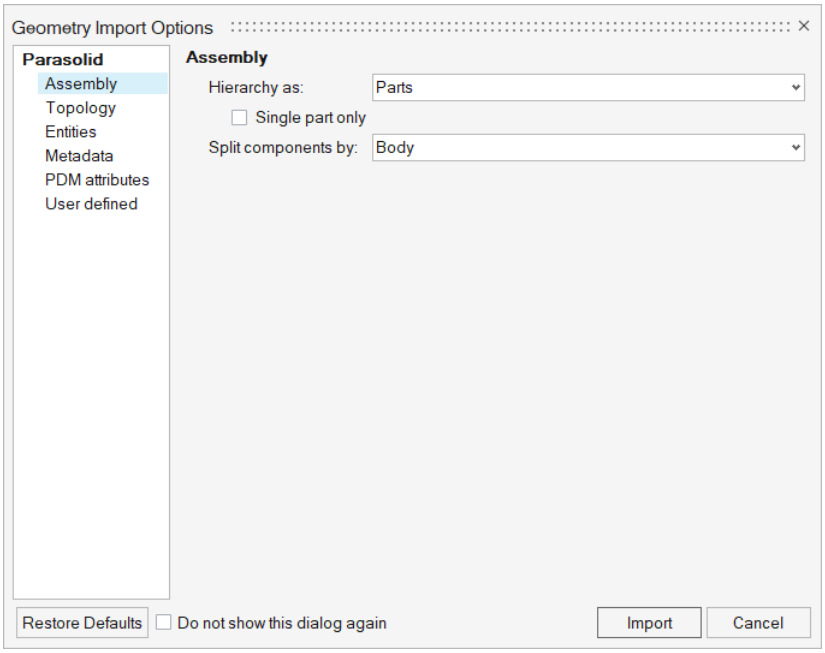


Figure 22:

4. Once the geometry is loaded, rotate and observe the features of the model.

   The view of the model displayed in the modeling window can be controlled using the following view controls.

Button	View Control
Middle mouse scroll	Zoom in and out
Right-click hold and drag	Pan the model
Middle mouse click hold and drag	Rotate the model

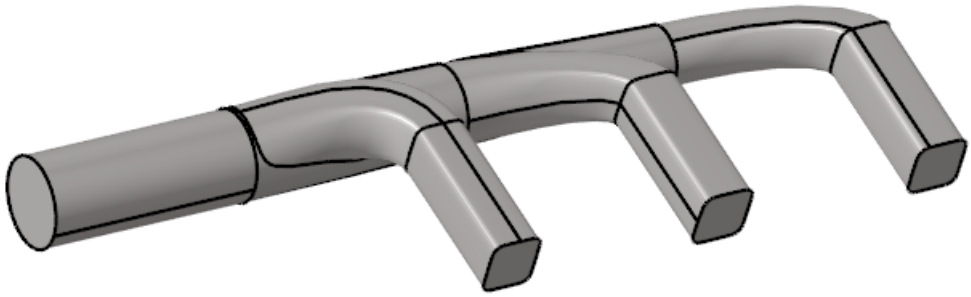


Figure 23:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 24:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 25:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up the Problem

In this step, you will set the general simulation parameters such as equation, solver settings, and so on. Then, you will assign the surface boundary conditions and material properties to the fluid domain.

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

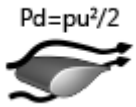


Figure 26:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that the **Incompressible** option is selected under Single phase flow.
   b) Set the Time marching to **Steady**.
   c) Select **Spalart-Allmaras** as the Turbulence model.

      The robustness and accuracy of the Spalart Allmaras turbulence model makes it an excellent choice for simulation of steady state flows.

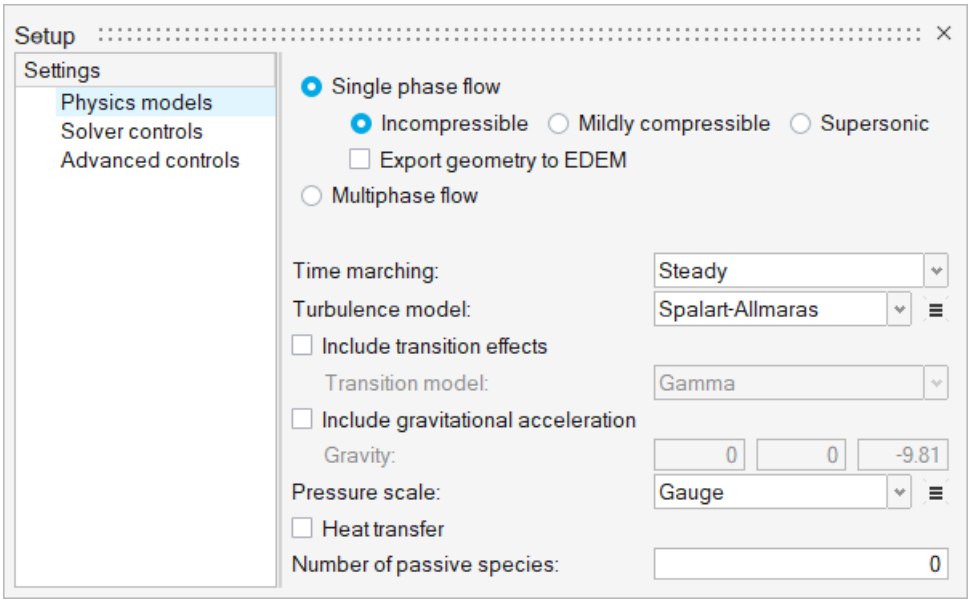


Figure 27:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

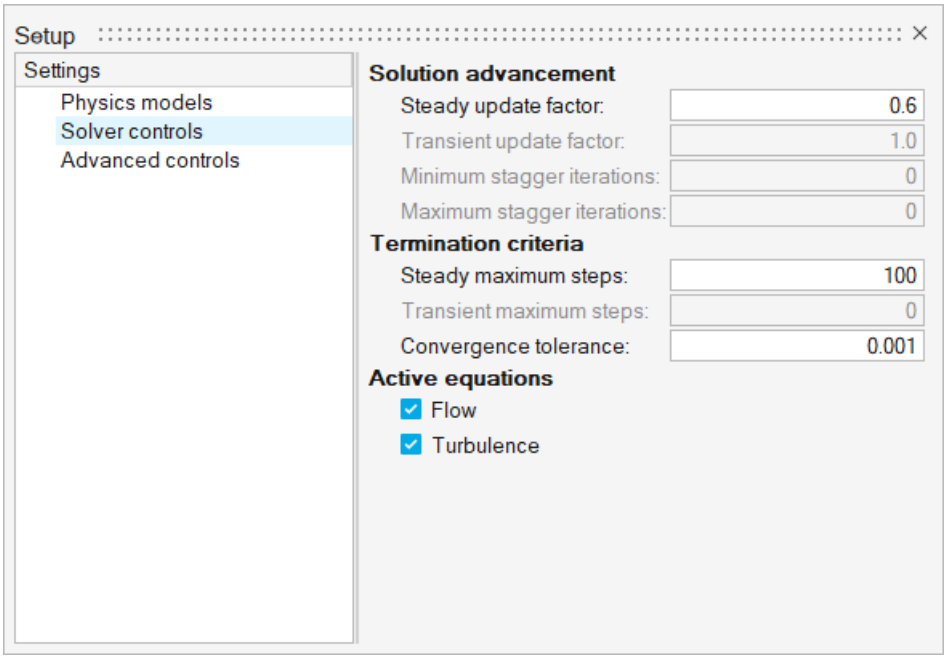


Figure 28:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 29:

2. From the modeling window, click anywhere on the manifold.
   The entire manifold geometry is highlighted and a Material microdialog appears.

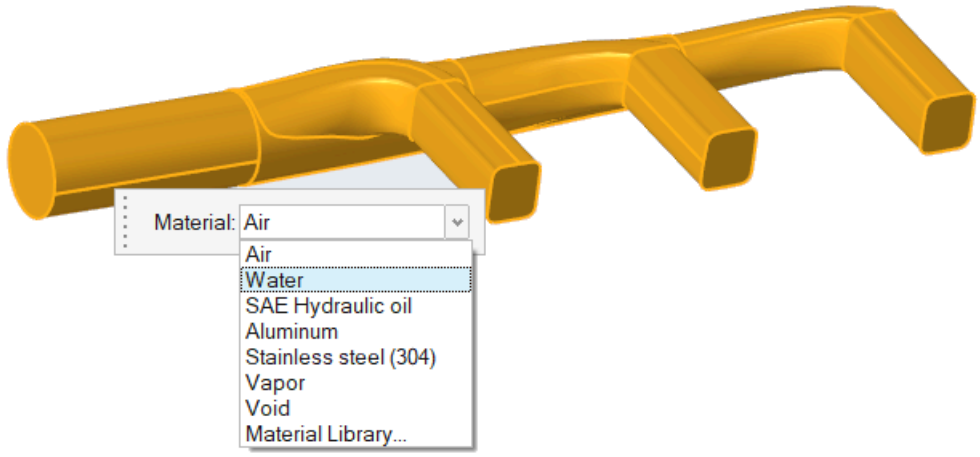


Figure 30:

3. Click the drop-down menu and select **Water** from the list of materials.
4. On the guide bar, click ▸ to execute the command.

   The changes made in the tool are not effective until they are executed by clicking the icon.

   Once the command is executed, the color of the geometry changes to indicate the material assigned to the volume. In this case, there is only one volume in the geometry so there is a single color. Alternatively, if there are multiple volumes with different materials assigned, the model will be displayed accordingly with distinct colors for each material assigned.

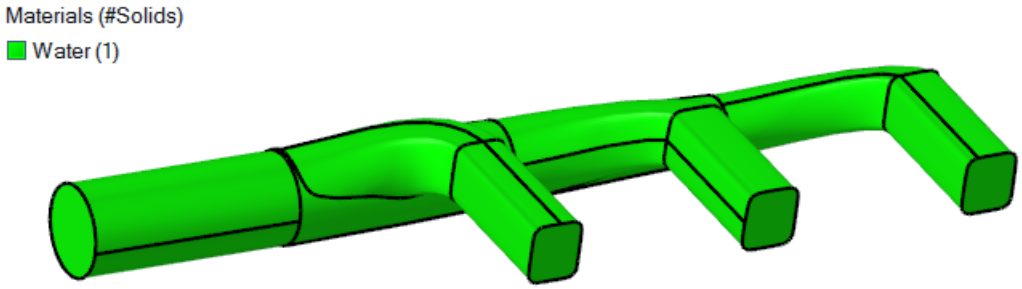


Figure 31:

5. On the guide bar, click ✓ to exit the tool.

Assign the Flow Boundary Conditions

The current model has one inlet, three outlets, and walls for the rest of the surfaces. When a geometry model is imported into HyperMesh CFD, all the surfaces are placed in the Default Wall (i.e. Type = auto_wall). As you start assigning the surface boundary conditions, those surfaces are moved into a

new boundary condition group. All the surface boundary condition tools are placed under the Boundaries sub-section of the Flow ribbon.

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 32:

2. In the modeling window, click the surface of the inlet.

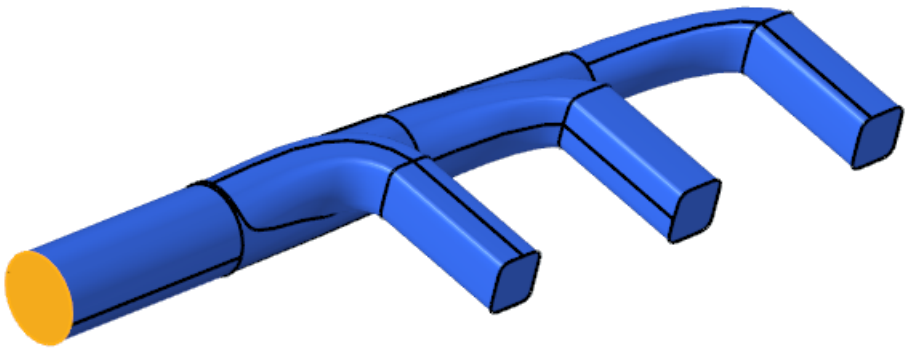


Figure 33:

Observe that a new group named "Inlet" is created under the Boundaries list in the top-left corner of the modeling window. Once the current command is executed, the highlighted surface will be moved into the Inlet group.

3. In the microdialog, enter a value of `2.0` m/sec for Average velocity.

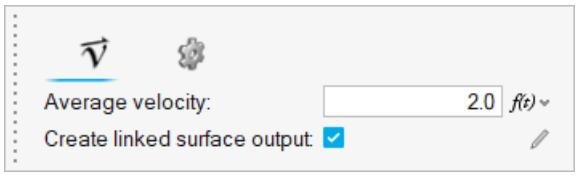


Figure 34:

4. On the guide bar, click ▸ to execute the command.
The color of the inlet surface in the modeling window is updated.

> **Note:** The color assigned to the surfaces is random. Therefore, the color of the surfaces shown in the images below might be different from what you see on your screen.

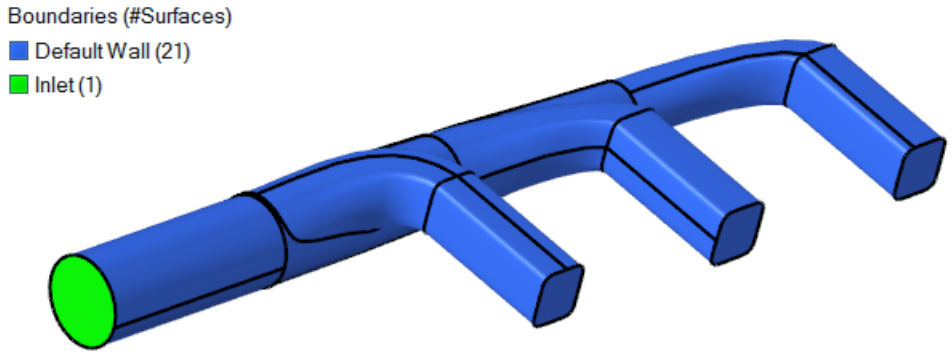


Figure 35:

5. Click the **Outlet** tool.

Figure 36:

6. Click the three outlet surfaces shown in the figure below.

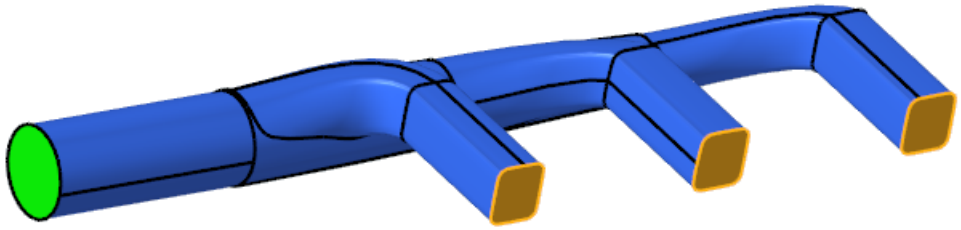


Figure 37:

7. Leave the default options in the dialog unchanged then click ▸ on the guide bar to execute the command.
The color of the outlet surfaces change and the list of boundaries on the left are updated. The number of surfaces under each group are shown in the brackets.

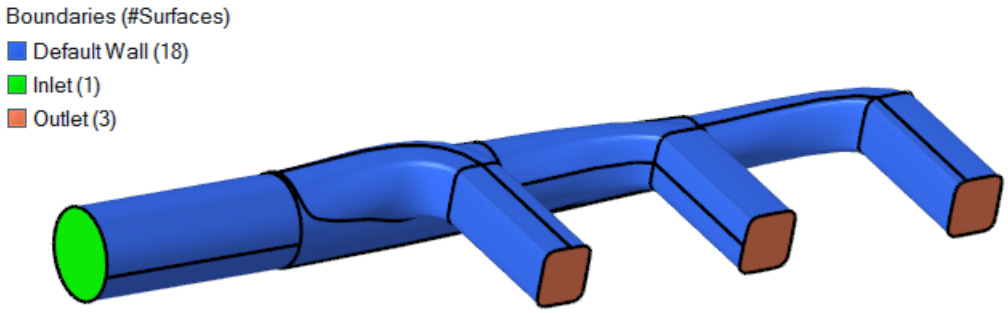


Figure 38:

Note:

- To update the color of any group, click on the colored-square on the left of the group name and select the color of your choice from the palette.
- To update the name of any group, right-click on the group name and select **Rename** from the context menu.

8. On the guide bar, click ✓ to exit the tool.
9. Save the model.

Define Mesh Controls

Now that you have assigned the material properties and boundary conditions, you will define the meshing parameters for the model and then generate the mesh.

Define the Surface Mesh Controls

1. From the Mesh ribbon, click the **Surface** tool.

Figure 39:

2. Right-click in the modeling window and go to **Select** > **All**.

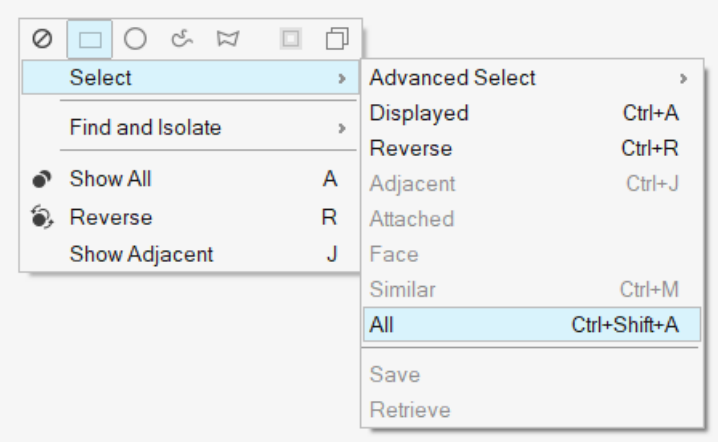


Figure 40:

All the surfaces in the model are highlighted and a dialog for surface mesh parameters appears.

3. Enter `0.003` m for the Average element size.

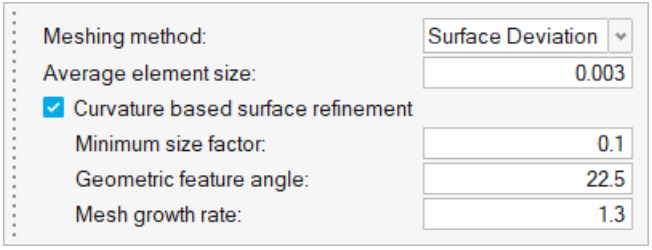


Figure 41:

4. Leave the default values for the remaining parameters unchanged then click ▸ on the guide bar to execute the command.

Define the Boundary Layer Mesh Parameters

1. From the Mesh ribbon, click the **Boundary Layer** tool.

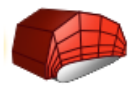


Figure 42:

2. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Boundaries** > **Default Wall**.

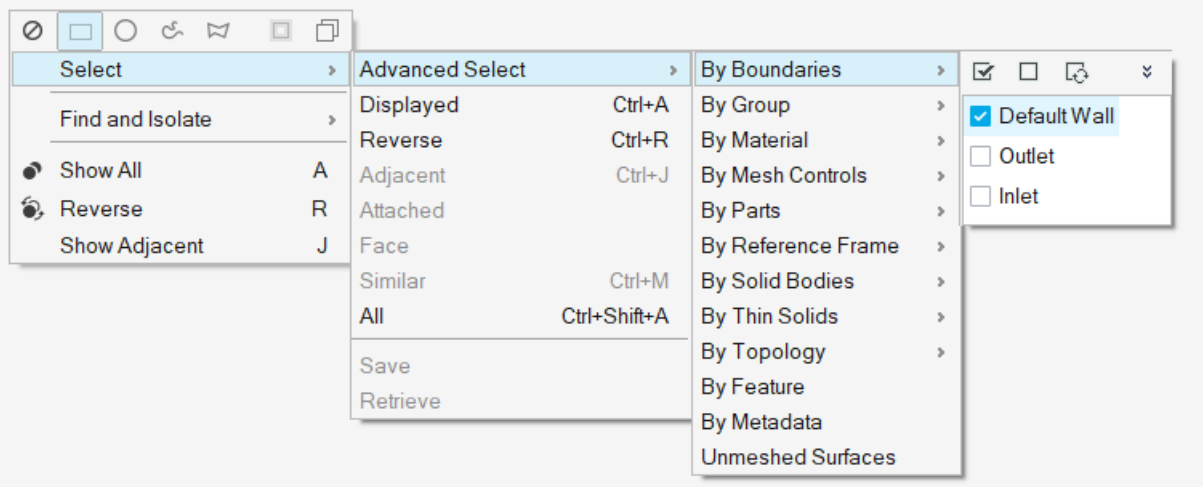


Figure 43:

All the wall surfaces are highlighted and a dialog for boundary layer parameters appears.

3. Enter the values in the dialog as shown in the figure below.

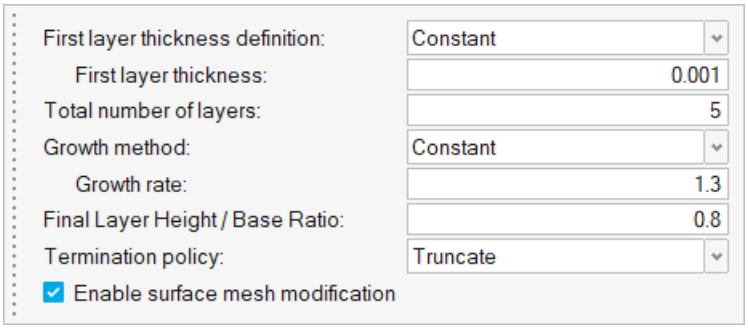


Figure 44:

4. On the guide bar, click ▸ to execute the command.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

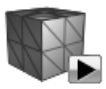


Figure 45:

2. In the **Meshing Operations** dialog, enter an Average element size of `0.005` m.

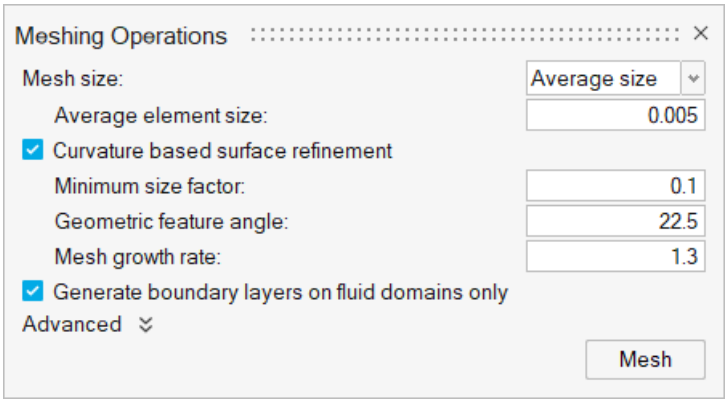


Figure 46:

3. Click **Mesh**.
   Once the meshing process has started, the **Run Status** dialog appears. To view the status of the meshing process, right-click on the process row and select **View log file**.

   Once the meshing is done, the run status is updated accordingly, and you are automatically moved to the Solution ribbon.

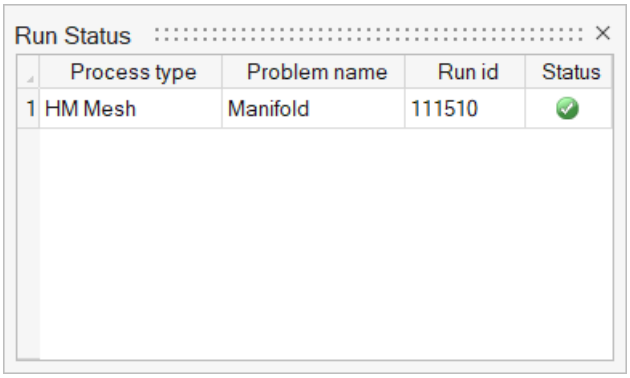


Figure 47:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 48:

2. In the **Launch AcuSolve** dialog, verify that the Problem directory and AcuRun path are pointing to the correct location.
3. Set the Parallel processing option to **Intel MPI**.

4. Optional: Set the number of processors to **4** or **8** based on availability.
5. Deactivate the **Automatically define pressure reference** option.
6. Expand the **Default initial conditions** menu.
7. Deactivate the **Pre-compute flow** option and enter `2.0` m/s for x-velocity.
8. Deactivate the **Pre-compute turbulence** flow option and enter `1e-5` m^2/s for Eddy viscosity.
9. Leave the remaining options as default and click **Run** to launch AcuSolve.

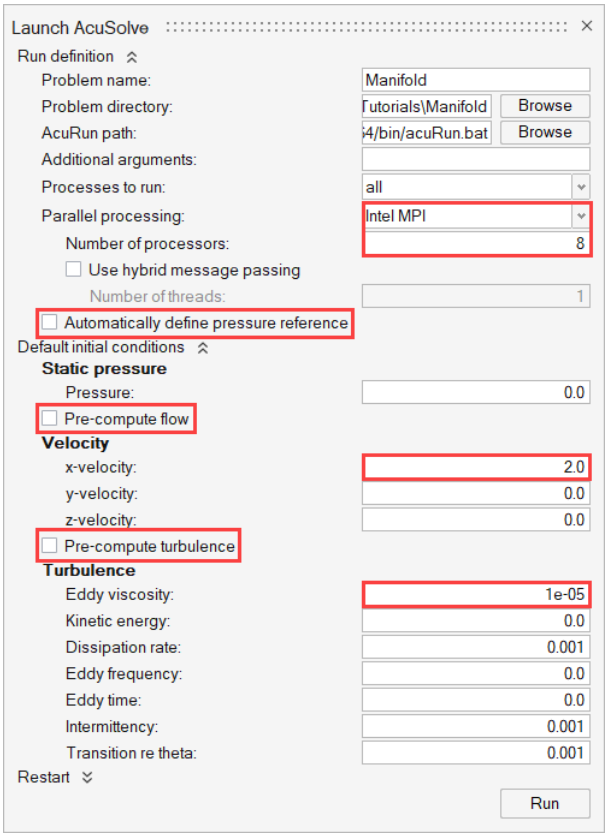


Figure 49:

The **Run Status** dialog opens again and the AcuSolve run appears on the list.

10. Right-click on the AcuSolve run and select **View log file**.

    A summary of the run printed in the dialog indicates that AcuSolve has finished running the simulation. Once the solution is converged, the Status will be updated accordingly.

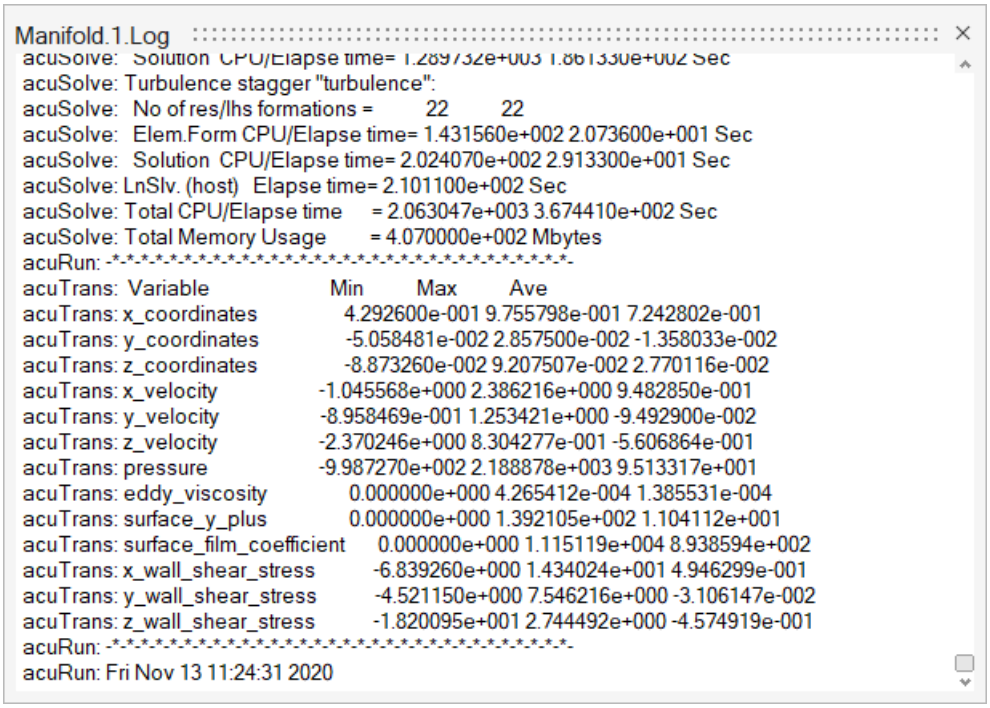


Figure 50:

11. In the **Run Status** dialog, right-click on AcuSolve run and select **Plot time history** to launch the **Plot Manager**.

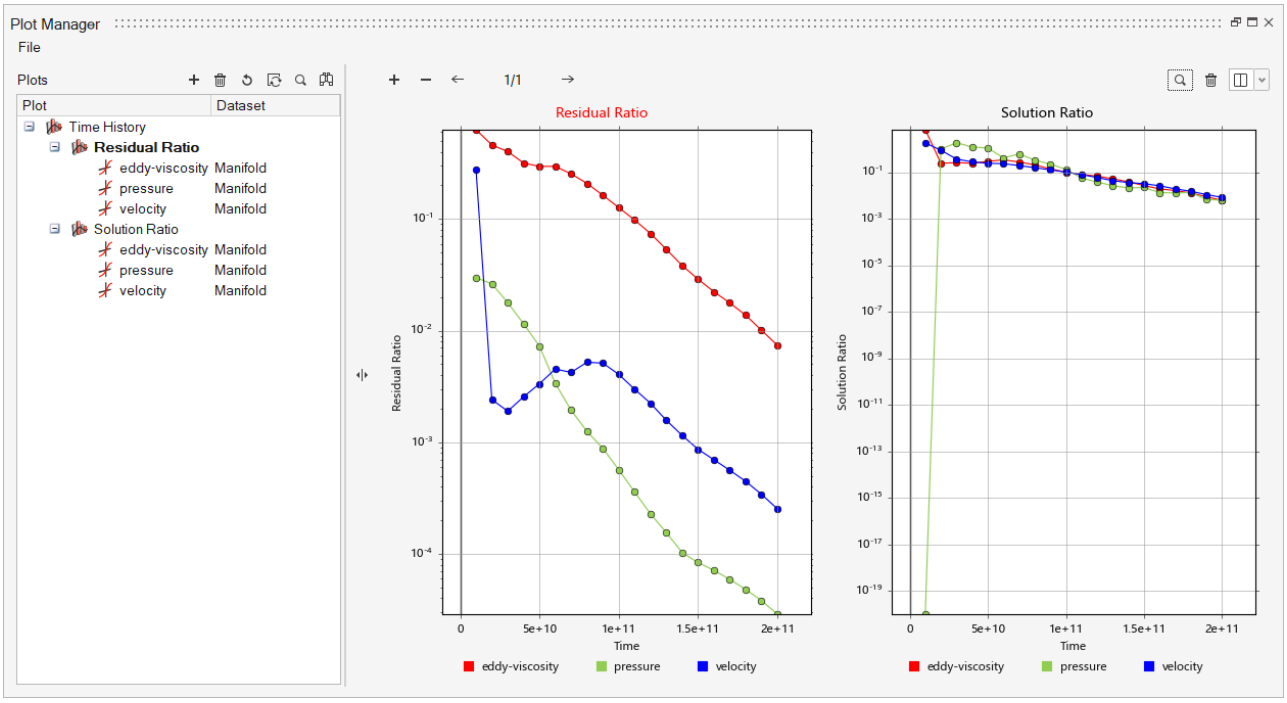


Figure 51:

The above plot shows the residual and solution ratios of the equations as the solution progresses through each time step. The convergence check is performed on the residual and solution increment ratios of each equation in the problem. For the residual, the ratio of the residual norm over the norm of the forces of the systems is used. For the solution increment, the ratio of the solution increment norm over the solution norm is used. These ratios are computed separately for each solution field, such as pressure and velocity.

Once the pressure and velocity residual ratios reach a value less than the specified convergence tolerance (0.001), the solution is considered to be converged. By default, the eddy viscosity convergence tolerance is set to a magnitude of one order higher than the specified convergence tolerance (0.01).

Once the pressure and velocity solution ratios reach a value less than the specified convergence tolerance (0.01), the solution is considered to be arrived at steady state. The eddy viscosity solution ratio convergence tolerance is set to a magnitude of one order higher than the specified solution ratio convergence tolerance (0.1).

Post-Process the Results with HyperMesh CFD Post

This part of the tutorial shows you how to work with a steady state solution using HyperMesh CFD Post.

Load the Results

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Browse to your working directory. Select the file `Manifold.1.Log` and click **Open**.
   The solid and all the surfaces are loaded in the Post Browser.

Create Pressure Contours on Boundary Surfaces

1. Click the **Boundary Groups** tool.

Figure 52:

2. Select all surfaces in the model by holding the left mouse button, dragging, and drawing a rectangle across the manifold.

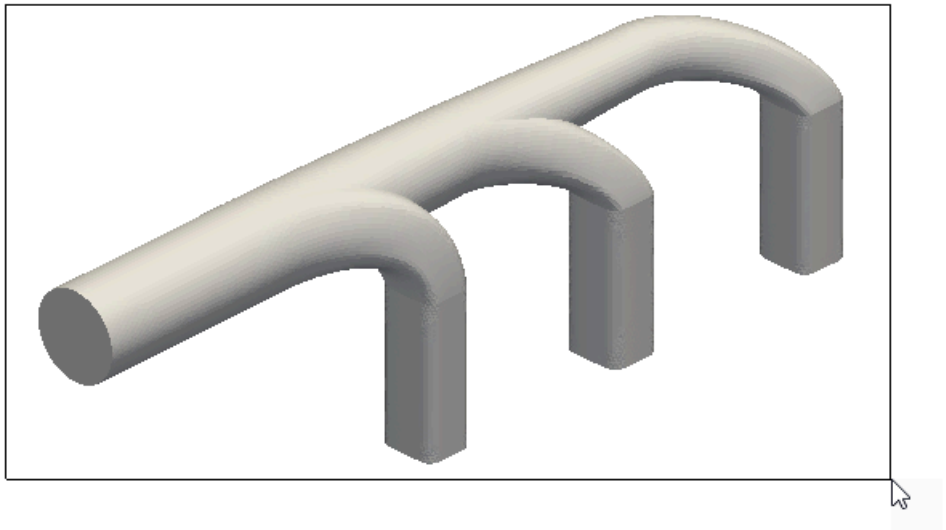


Figure 53:

3. In the microdialog, set the Display variable to **pressure**.
4. Activate the **Legend** toggle and click ⚡ to reset the range.
5. Click ≡ and set the Colormap Name to **Rainbow Uniform**.

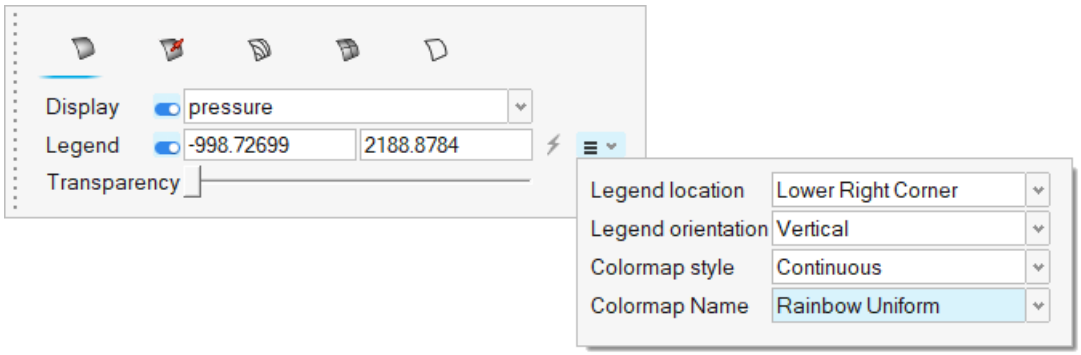


Figure 54:

6. Click ✓ on the guide bar.
   The pressure legend is in Pa.

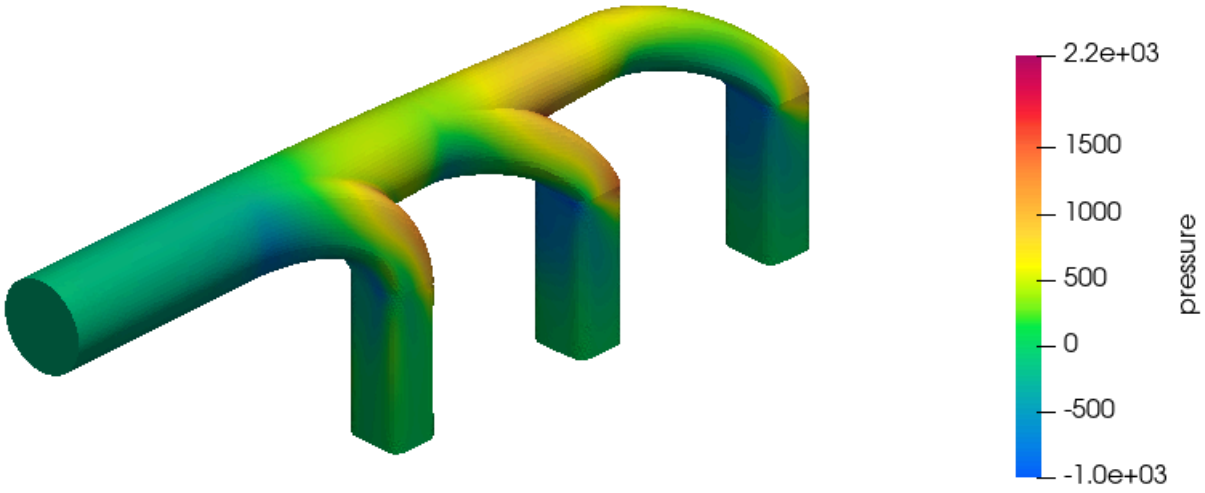


Figure 55:

7. In the Post Browser, right-click on **Boundary Group 4** and select **Rename**. Name it `Pressure_plot`.

Plot Velocity and Pressure Contours on a Section Cut

1. In the Post Browser, turn off the display of the boundary surfaces by clicking on the icon next to Flow Boundaries.

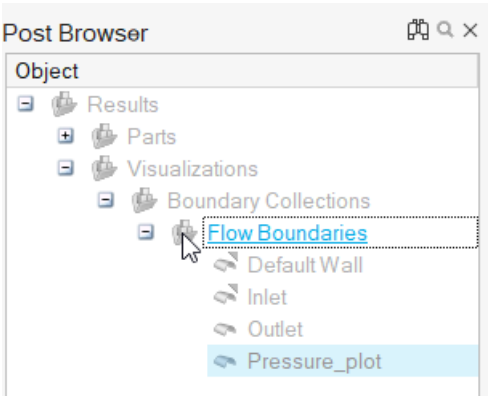


Figure 56:

2. Click the **Slice Planes** tool.

Figure 57:

3. Select the x-z plane in the modeling window.

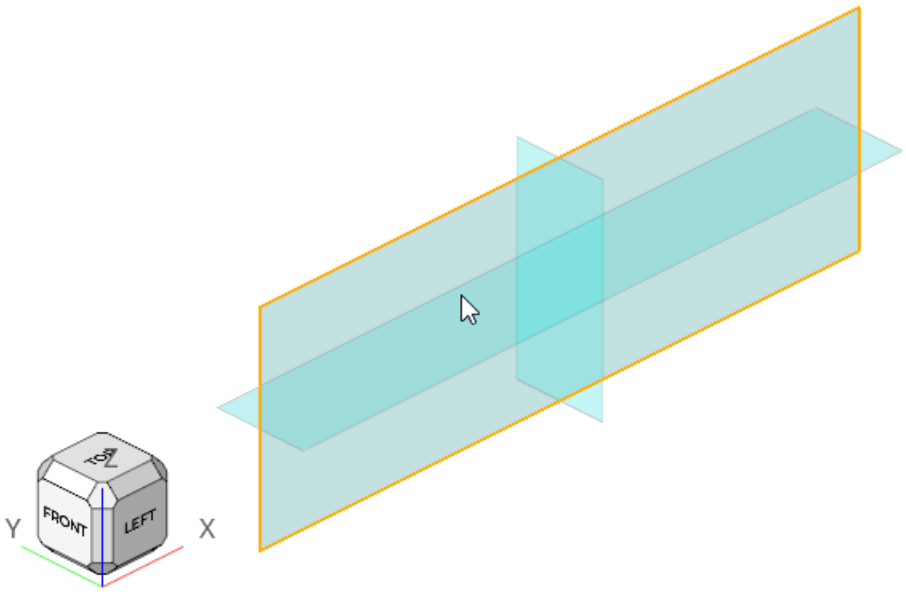


Figure 58:

4. In the microdialog, click to adjust the plane with the Vector tool.
5. In the Vector tool dialog, click XYZ to define the coordinates of the center of plane. Set the value of Y to `-0.015` and press **Enter**.

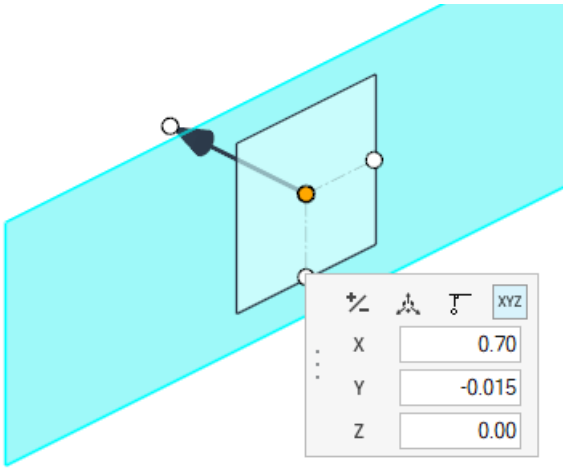


Figure 59:

6. Press **Esc** twice.
7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, set the Display variable to **pressure**.
9. Activate the **Legend** toggle and click to reset the range.
10. Click and set the Colormap Name to **Rainbow Uniform**.

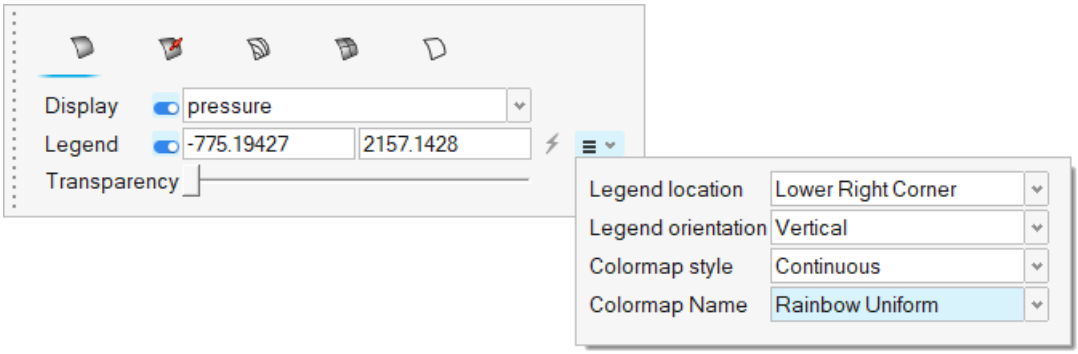


Figure 60:

11. Click on the guide bar.

    The pressure legend is in Pa.

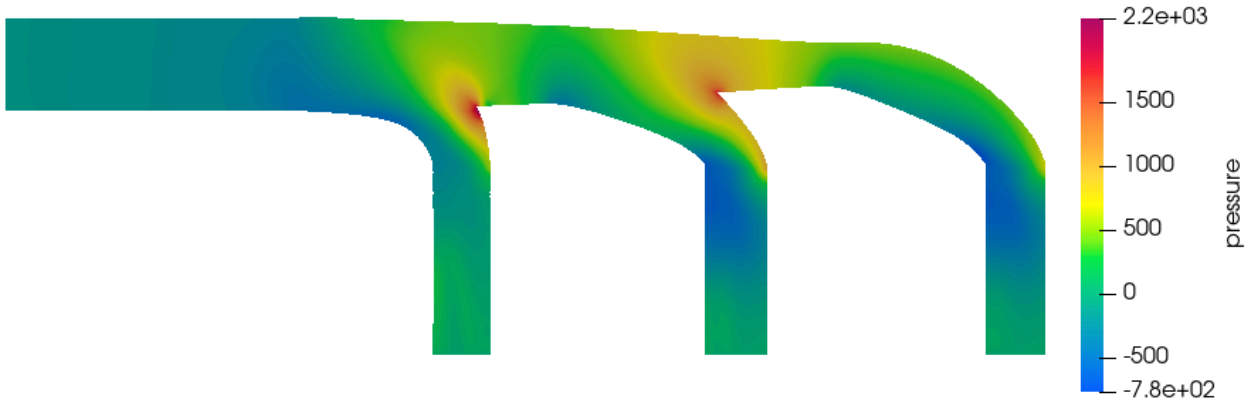


Figure 61:

12. In the Post Browser, right-click on **Slice Plane 1** and select **Edit**.

13. In the microdialog, change the Display variable to **velocity** (m/s) then click ✓ on the guide bar.

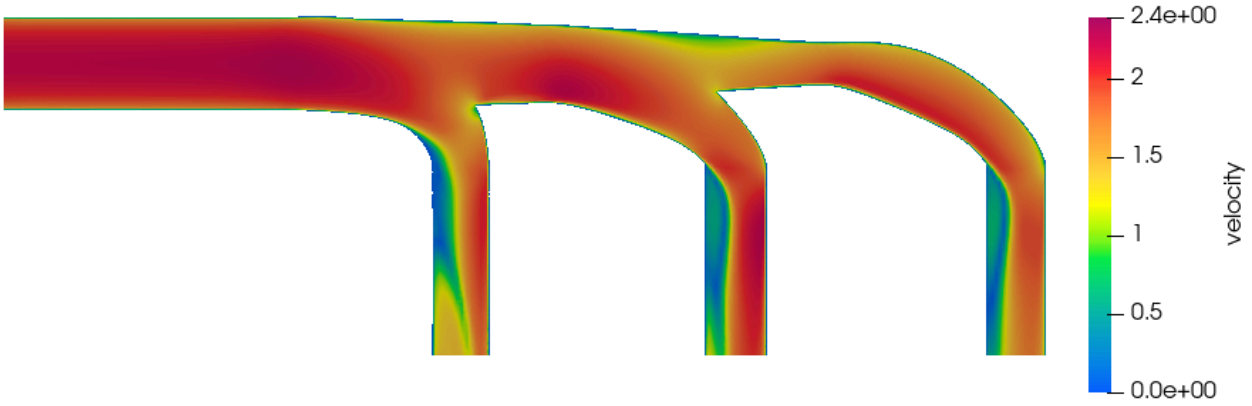


Figure 62:

Summary

In this tutorial, you worked through a basic workflow to carry out a CFD simulation and post-process the results using HyperMesh CFD. You started by importing the geometry and meshing the model in HyperMesh CFD. You also set up the model and launched AcuSolve directly from within HyperMesh CFD. Upon completion of the solution by AcuSolve, you post-processed the results in the Post ribbon. You learned how to create contours on the boundary surfaces and section cuts.

SimLab

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up a Computational Fluid Dynamics (CFD) analysis using Altair SimLab. SimLab is a process-oriented multidisciplinary simulation environment where multiple physics, such as CFD, structural, and thermal can be easily set up using highly automated modeling tasks.

In this tutorial, you will learn how to use SimLab for setting up a CFD analysis while exploring different capabilities available within the software for importing a geometric model, generating the mesh, setting up the simulation, launching AcuSolve, and post-processing the results.

To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. Before you begin, download the CAD geometry.

You will need an internet connection to watch the video tutorials and download the files.

Tutorial Video

https://fast.wistia.net/embed/iframe/sbcatkw2le

Additional Resources

For more tutorials, visit the SimLab Learning Center.

Flow

2

This chapter covers the following:

- ACU-T: 2000 Turbulent Flow in a Mixing Elbow (p. 45)
- ACU-T: 2100 Turbulent Flow Over an Airfoil Using the SST Turbulence Model (p. 61)
- ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building (p. 96)
- ACU-T: 2400 Supersonic Flow in a Converging-Diverging Nozzle (p. 112)
- ACU-T: 2500 Floating Surface Monitor (p. 127)

ACU-T: 2000 Turbulent Flow in a Mixing Elbow

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T2000_MixingElbow.hm`

Problem Description

The problem characteristics shown here determine if the flow is laminar or turbulent by calculating the Reynolds number in the pipe. The diameter of the large inlet is 0.1 m, and the inlet velocity (v) is 0.4 m/s. The diameter of the small inlet is 0.025 m, and the inlet velocity is 1.2 m/s.

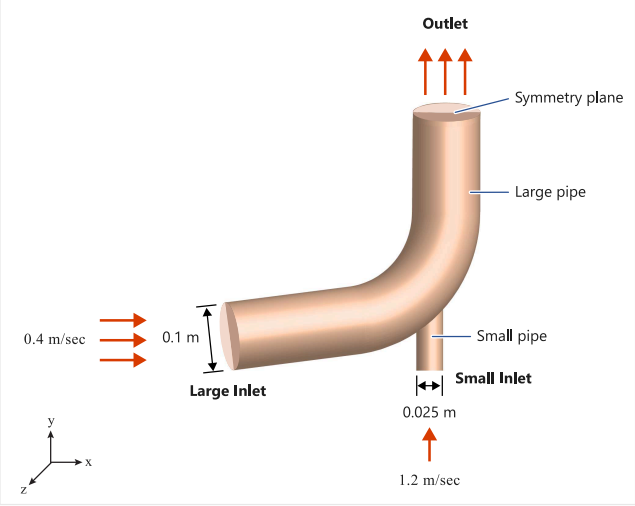


Figure 63: Schematic of Mixing Elbow

The fluid in this problem is water with the following properties that do not change with temperature: a density (ρ) of 1000 kg/m^3 and a molecular viscosity (μ) of 1 X 10^{-3} kg/m-sec.

Based on mass conservation, the combined flow rate (Q) yields a pipe velocity of 0.475 m/s downstream of the small inlet. The following equations are used to calculate the pipe velocity:

$$Q_{LargeInlet} = A_{LargeInlet} V_{LargeInlet}$$
$$Q_{SmallInlet} = A_{SmallInlet} V_{SmallInlet}$$
$$Q_{Total} = Q_{LargeInlet} + Q_{SmallInlet} \quad (1)$$
$$V_{pipe} = \frac{Q_{total}}{A_{LargeInlet}} = 0.475 m \quad s$$

This value is useful in determining the Reynolds number, which in turn can be used to determine if the flow should be modeled as turbulent, or if it should be modeled as laminar.

In order to determine whether the modeled flow would be turbulent or whether it would be laminar, the Reynolds number (Re) should be calculated. The Reynolds number is given by:

$$\mathrm{Re} = \frac{\rho V D}{\mu} \quad (2)$$

where ρ is the fluid density, *V* is the fluid velocity, *D* is the diameter of the flow region, and μ is the molecular viscosity of the fluid. When the Reynolds number is above 4,000, it is generally accepted that flow should be modeled as turbulent.

The Reynolds numbers of 40,000 at the large inlet, 30,000 at the small inlet, and 47,500 for the combined flow indicate that the flow is turbulent throughout the flow domain. The simulation will be set up to model steady state, turbulent flow.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 64:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T2000_MixingElbow.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `MixingElbow_Turbulent` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.

6. Enter `MixingElbow` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 65:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 66:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

Figure 67:

The **Setup** dialog opens.

2. Under the Physics models setting:

a) Verify that the **Incompressible** option is selected under Single phase flow.

b) Set the Time marching to **Steady**.

c) Select **Spalart-Allmaras** as the Turbulence model.

The robustness and accuracy of the Spalart Allmaras turbulence model makes it an excellent choice for simulation of steady state flows.

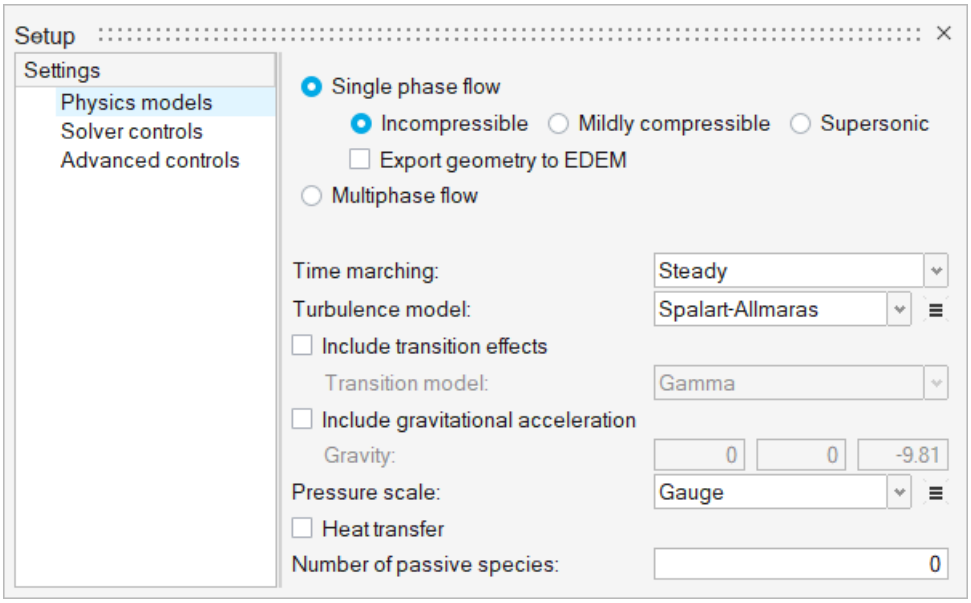


Figure 68:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

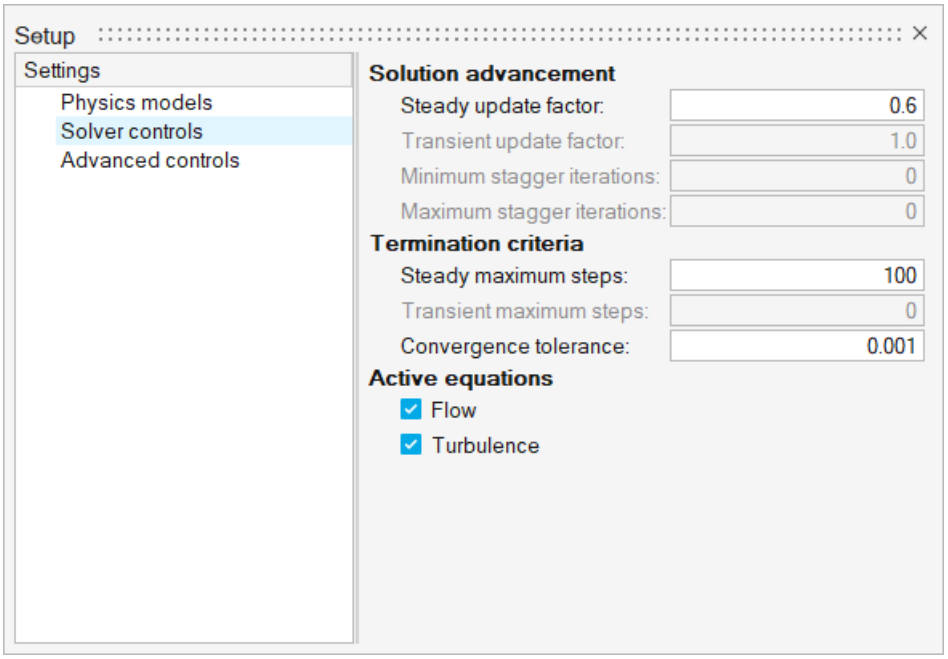


Figure 69:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 70:

2. Select the model body.
   The entire solid is highlighted.

Figure 71:

3. In the microdialog, click the drop-down menu next to Material and select **Water**.
4. On the guide bar, click ✓ to execute the command and exit the tool.

Assign Flow Boundary Conditions

Set Boundary Conditions for the Large Inlet

The large inlet of the mixing elbow will be defined as a profiled inlet based on average velocity.

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 72:

2. Click the face of the large inlet.

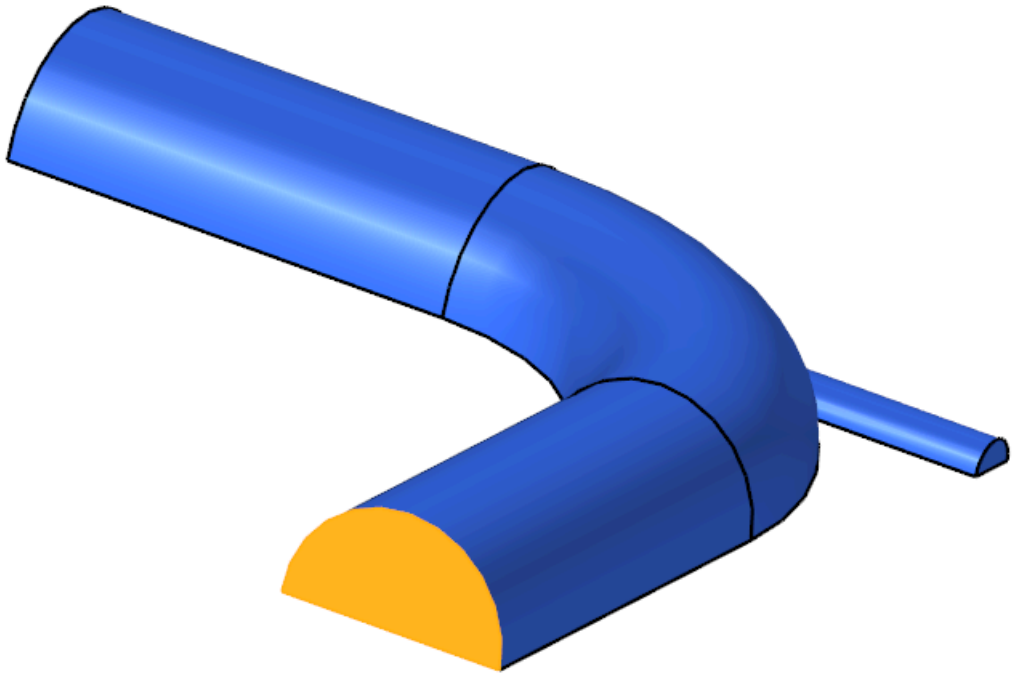


Figure 73:

3. In the microdialog, enter a value of `0.4` m/s for Average velocity.

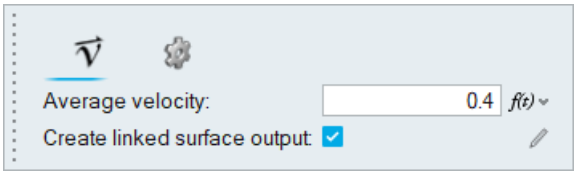


Figure 74:

4. Rename the inlet.
   a) From the legend on the left side of the modeling window, double-click on **Inlet**
   b) Type `Large_Inlet` and press **Enter**.
5. On the guide bar, click ▸ to execute the command and remain in the tool.

> **Note:** The number of inlets created appears in parenthesis on the top-right of the Profiled tool icon.

Set Boundary Conditions for the Small Inlet

The small inlet of the mixing elbow will also be defined using average velocity. The guide bar for the Profiled tool should still be open from the last step.

1. Click the face of the small inlet.

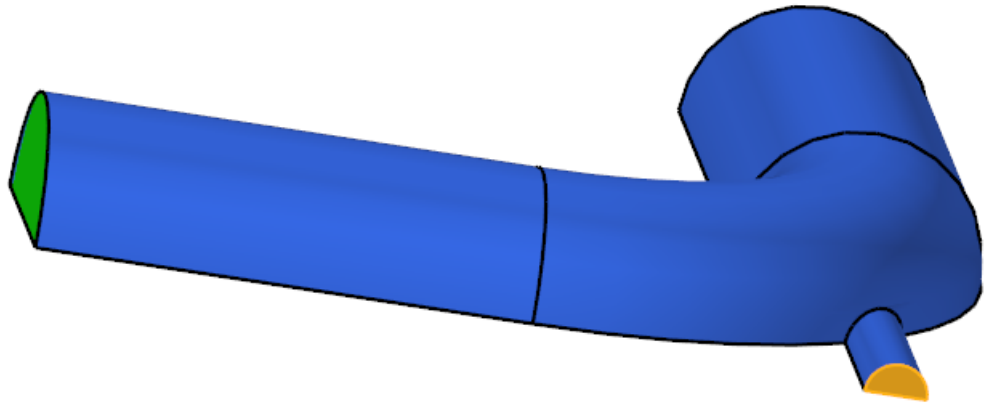


Figure 75:

2. In the microdialog, enter a value of `1.2` m/s for Average velocity.

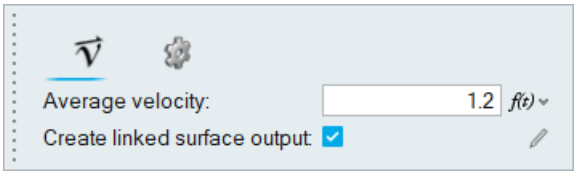


Figure 76:

3. Rename the inlet.
   a) From the legend on the left side of the modeling window, double-click on **Inlet**
   b) Type `Small_Inlet` and press **Enter**.
4. On the guide bar, click ✓ to execute the command and exit the tool.

Set Boundary Conditions for the Outlet

1. From the Flow ribbon, click the **Outlet** tool.

Figure 77:

2. Click the face of the outlet.

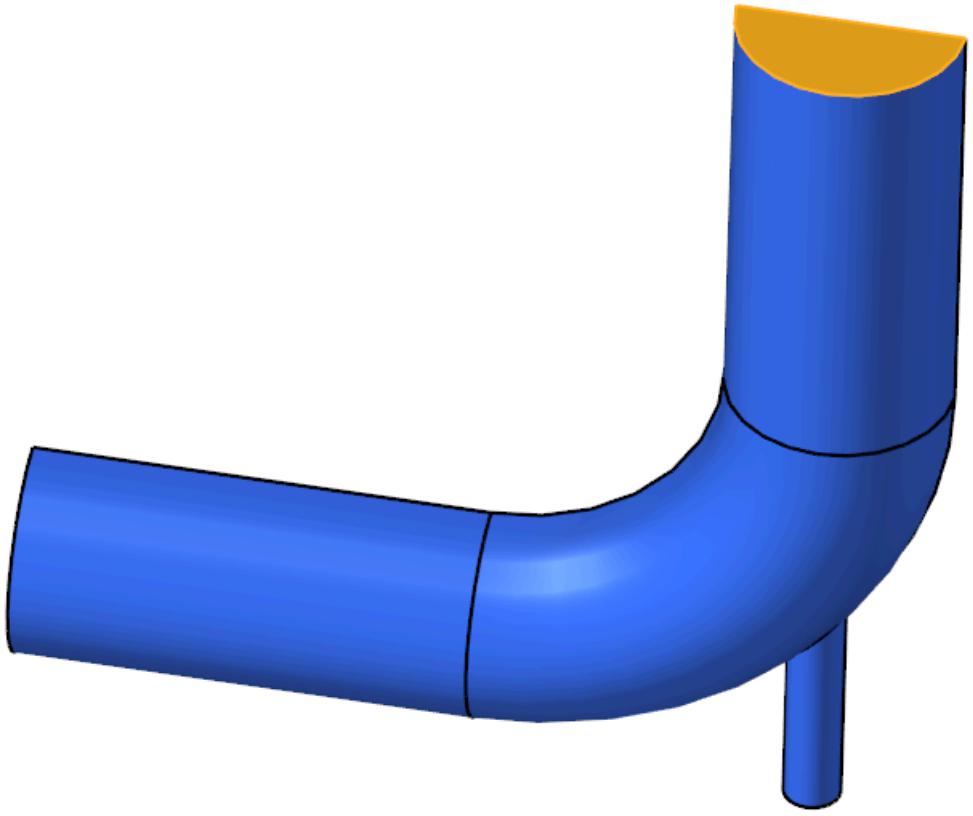


Figure 78:

3. In the microdialog, make sure the Static pressure is 0 Pa and the Pressure loss factor is also 0.

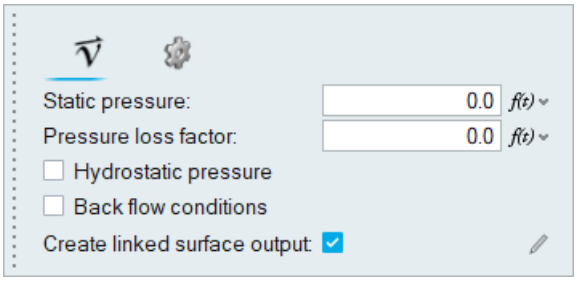


Figure 79:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Set Boundary Conditions for the Symmetry Plane

This geometry is symmetric about the XY midplane, and can therefore be modeled with half of the geometry. In order to take advantage of this, the midplane needs to be identified as a symmetry plane. The symmetry boundary condition enforces constraints such that the flow field from one side of the plane is a mirror image of that on the other side.

1. From the Flow ribbon, click the **Symmetry** tool.

Figure 80:

2. Select the face of the symmetry plane.

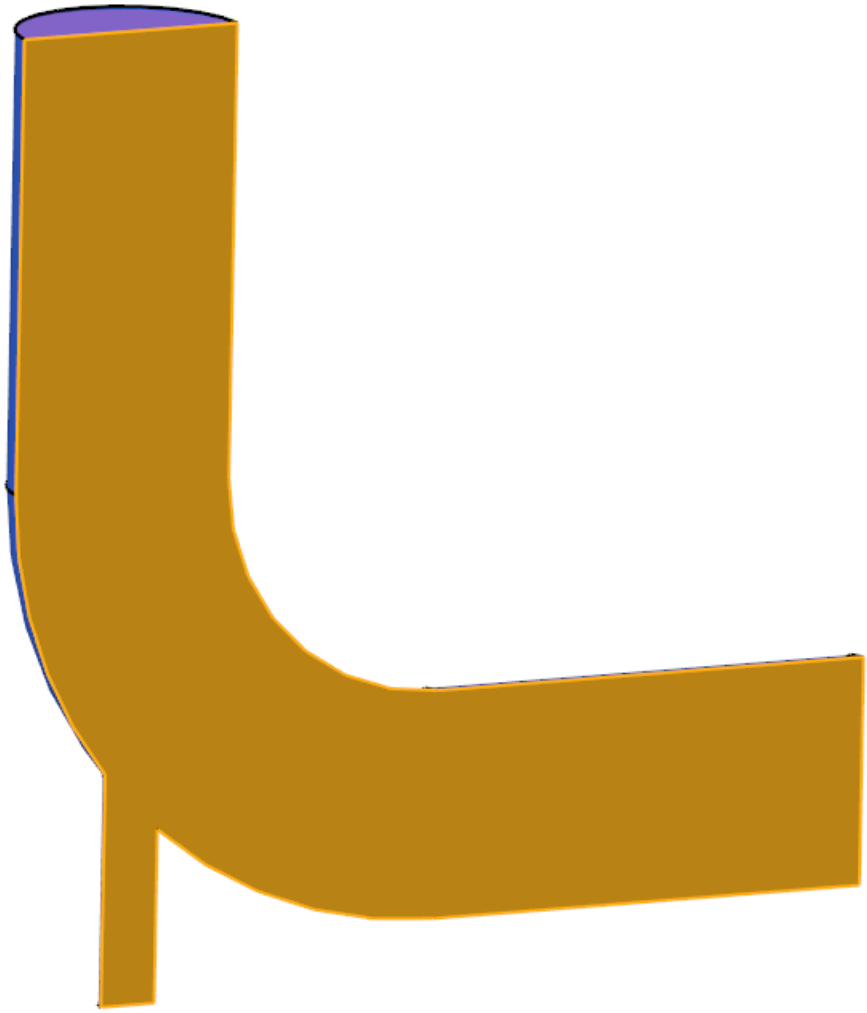


Figure 81:

3. In the microdialog, accept the default symmetry conditions.

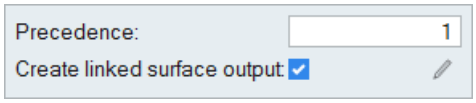


Figure 82:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Save the database.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

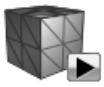


Figure 83:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.025 m.
3. Accept all other default parameters.

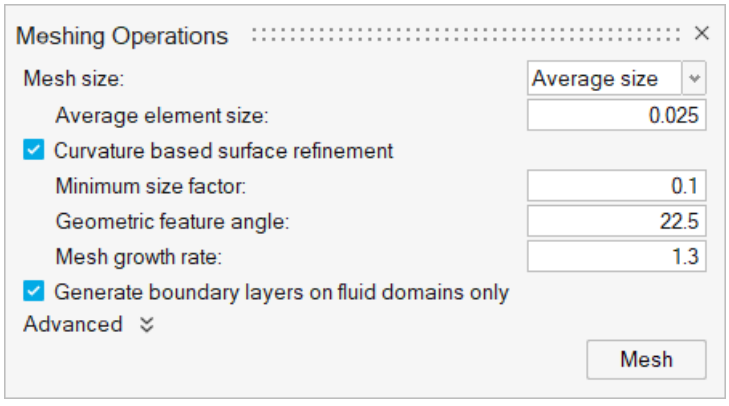


Figure 84:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

5. From the back side of the model, observe the refined mesh around the small inlet.

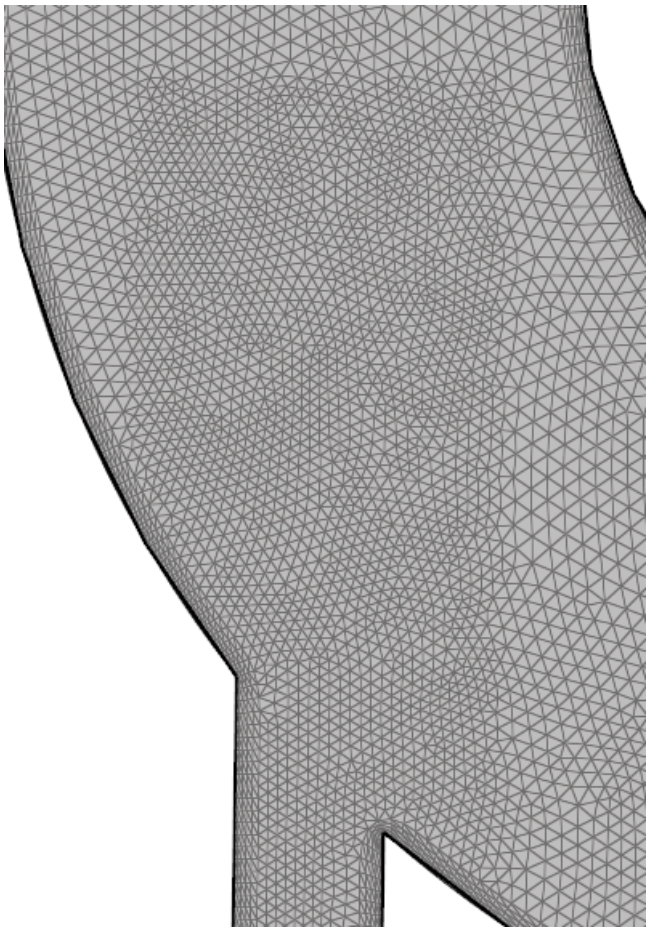


Figure 85:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 86:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.

3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

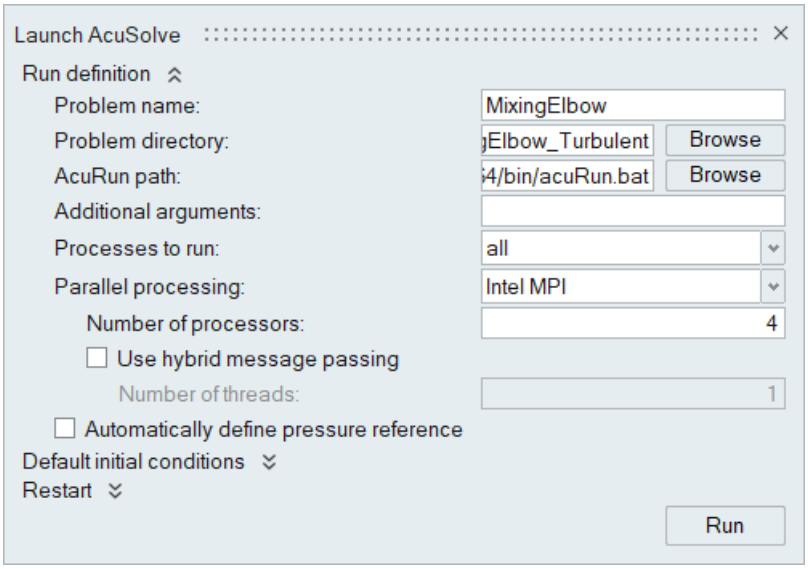


Figure 87:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Isolate the **Symmetry** flow boundary in the Post Browser.
5. Right-click on **Symmetry** and select **Edit**.
6. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
7. Click ≡ and set the Colormap Name to **Rainbow Uniform**.

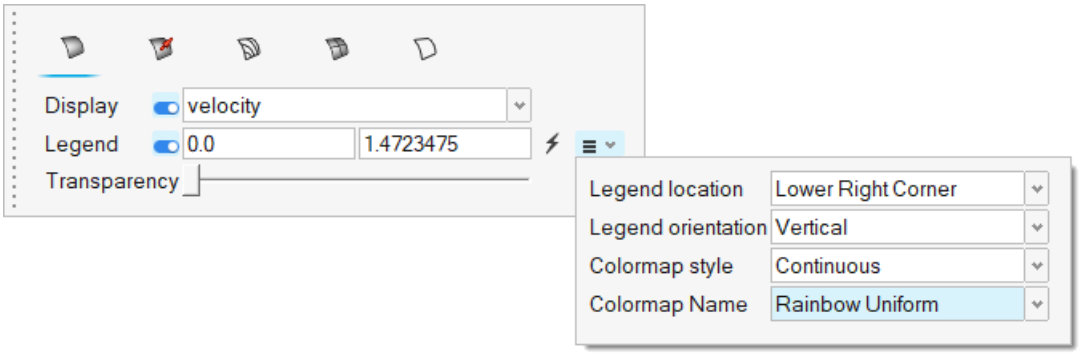


Figure 88:

8. Click ✓ on the guide bar to show the velocity magnitude (m/s) contour.

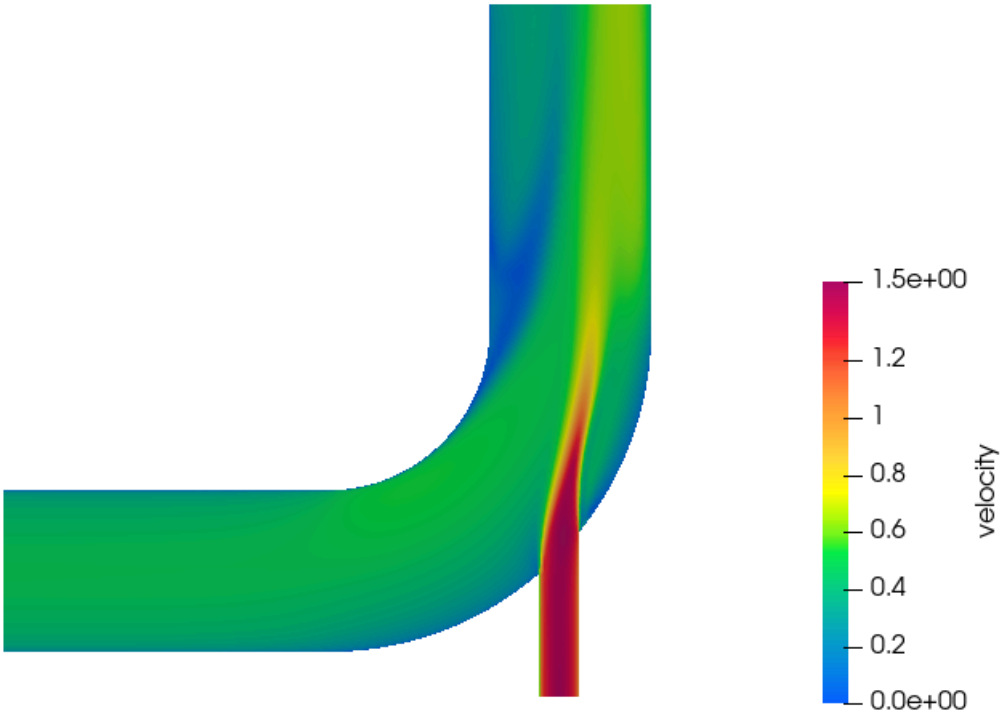


Figure 89:

9. Again, right-click on the **Symmetry** and select **Edit**.

10. Display the **pressure** (Pa).

11. Click ⚡ to reset the range.

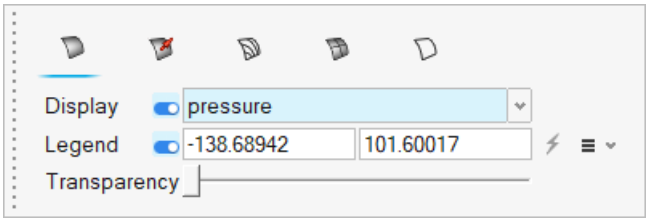


Figure 90:

12. Click ✓ on the guide bar to show the pressure (Pa) contours.

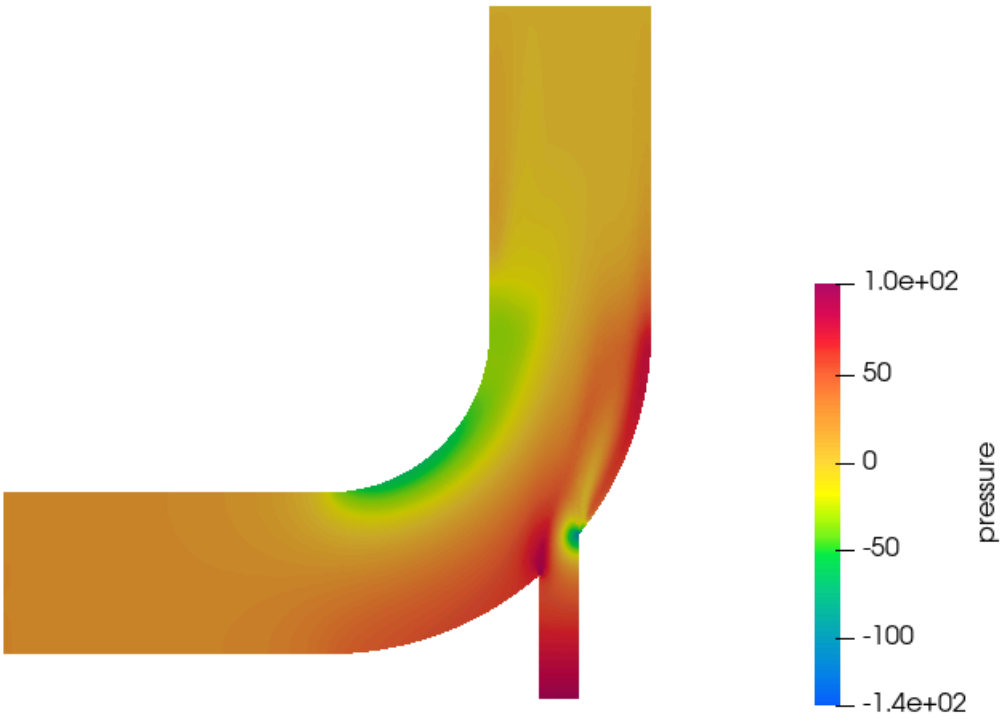


Figure 91:

Summary

In this tutorial, you worked through a basic workflow to set up a CFD model, carry out a CFD simulation, and post-process the results. You started by importing the model in HyperMesh CFD. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh CFD. Upon completion of the solution by AcuSolve, you used the Post ribbon to create contour plots.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/s6vec7wjai

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 2100 Turbulent Flow Over an Airfoil Using the SST Turbulence Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T2100_NACA0012airfoil.x_b`

Problem Description

The problem to be addressed in this tutorial is shown schematically in the following image. It consists of a cylindrical bounding region containing air that flows past a NACA0012 airfoil profile. The simulation is performed as 2D by including only a single layer of extruded elements in the airfoil span direction. The velocity vector at the far field boundary of the domain is specified to yield an angle of attack of 5 degrees and a Reynolds Number of 1.0e6. The airfoil chord is 1 meter, and standard air material properties are used for the simulation.

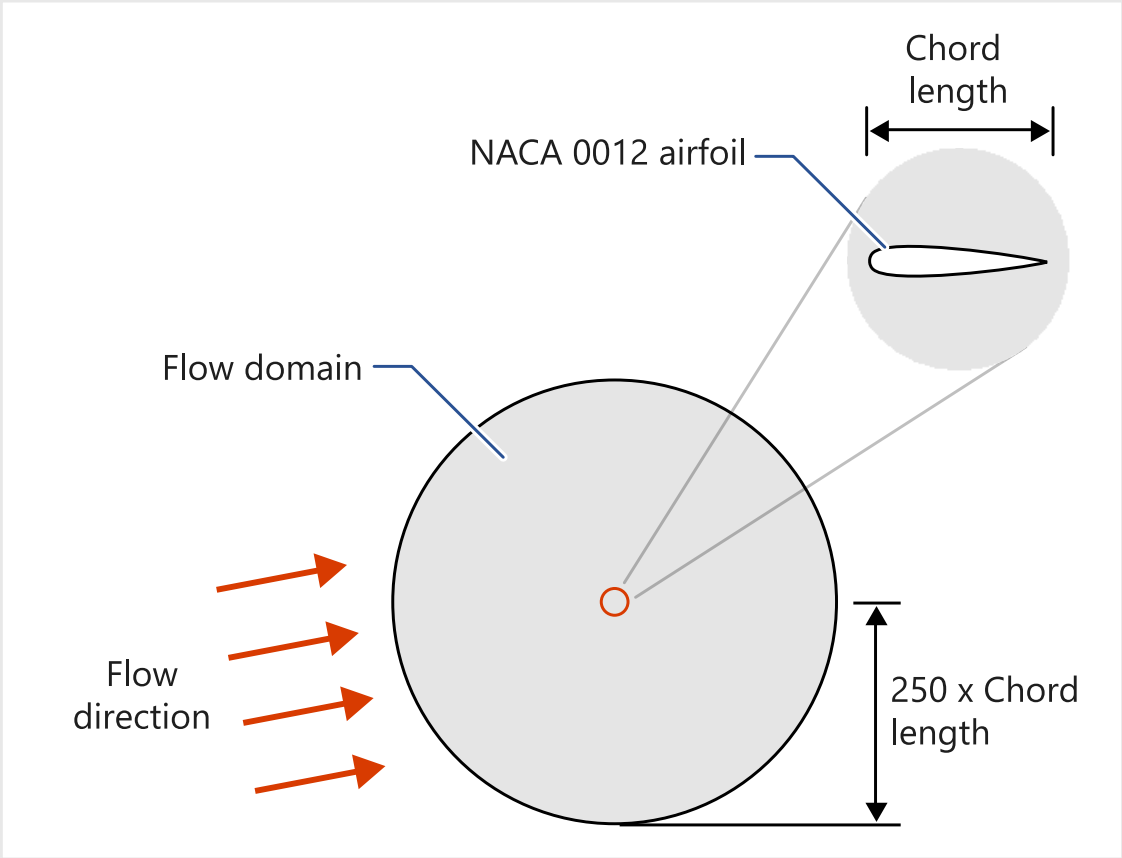


Figure 92:

The diameter of the cylindrical bounding volume for the airfoil is set to 500 times the airfoil chord. This large bounding volume is selected to ensure that the farfield boundaries are sufficiently far from the airfoil to prevent any influence of blockage of the domain on the solution.

Since the flowfield near the airfoil is assumed to be turbulent, the SST turbulence model will be used for the simulation. These simulation conditions correspond to a scenario where the boundary layer on the leading edge of the airfoil is tripped with some type of roughness elements to produce a fully turbulent boundary layer over the length of the airfoil.

Start HyperMesh CFD and Create the Simulation Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

2. Click **File** > **Save As** and create a new directory named `TubulentAirfoil_SST`, or any name of your preference, and navigate to the directory.

   This will be the working directory where all files related to the simulation will be saved.
3. Save the file as `Airfoil`, or any name of your preference.

   This file will be the database file for this case.
4. Click **Save** to create the database.

Import the Airfoil Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T2100_NACA0012airfoil.x_b** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

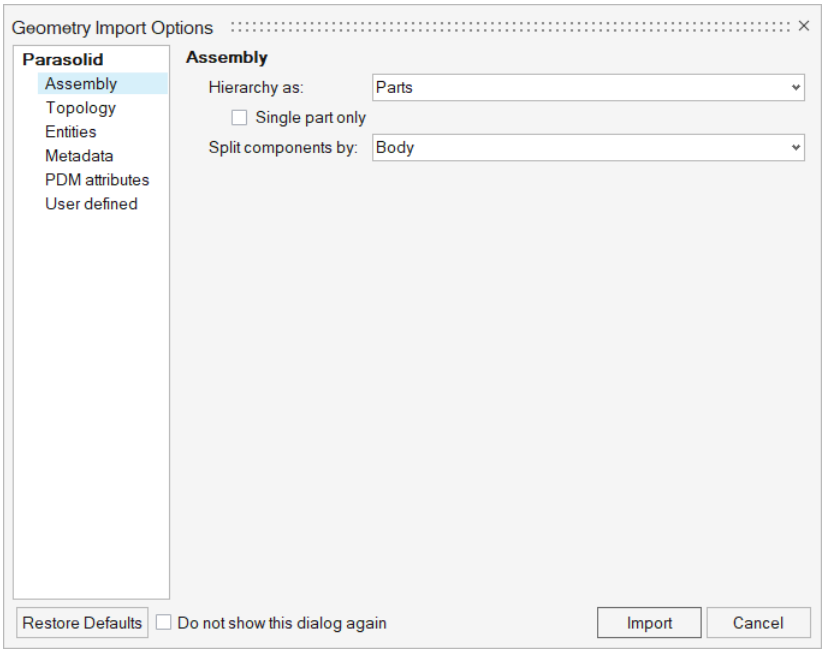


Figure 93:

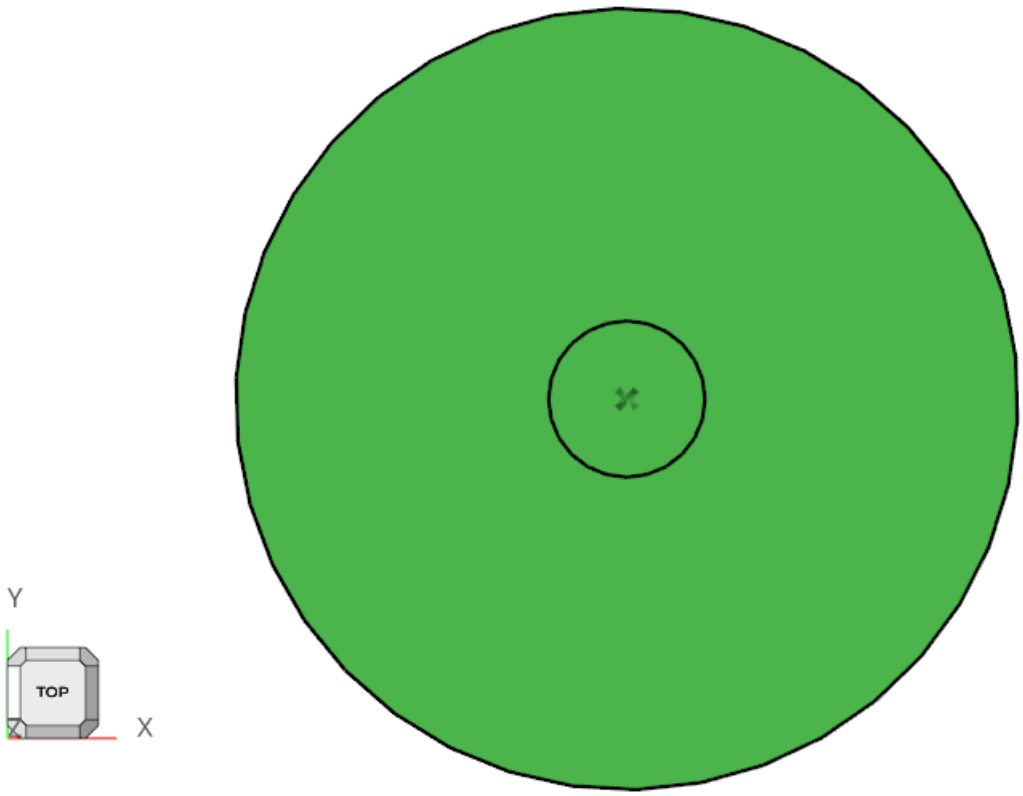


Figure 94:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 95:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 96:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

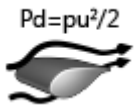


Figure 97:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that the **Incompressible** option is selected under Single phase flow.
   b) Set the Time marching to **Steady**.
   c) Select **SST** as the Turbulence model.

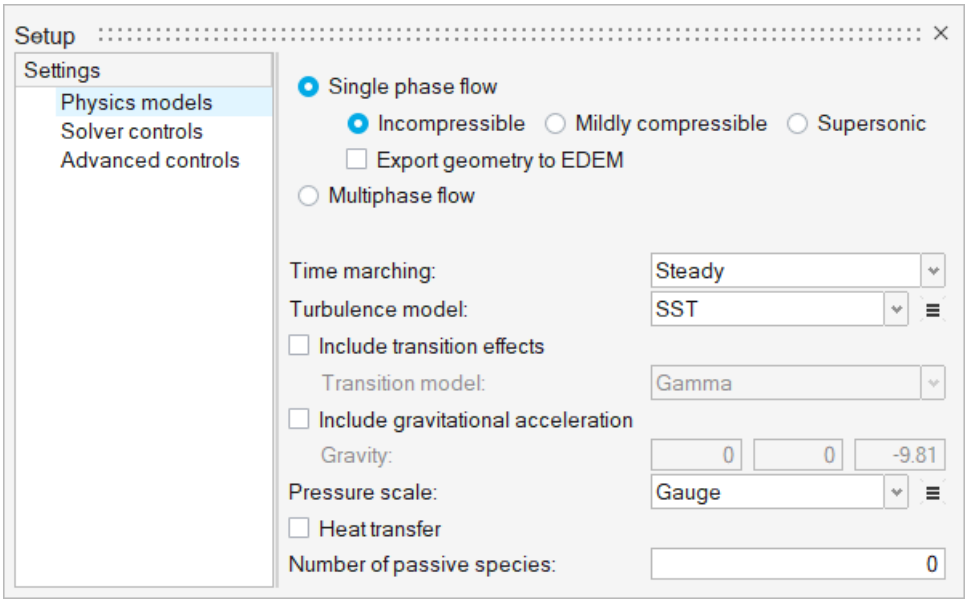


Figure 98:

3. Click the **Solver controls** setting.
4. Set the Steady update factor to `0.6`, the Steady maximum steps to `100`, and the Convergence tolerance to `0.001`.

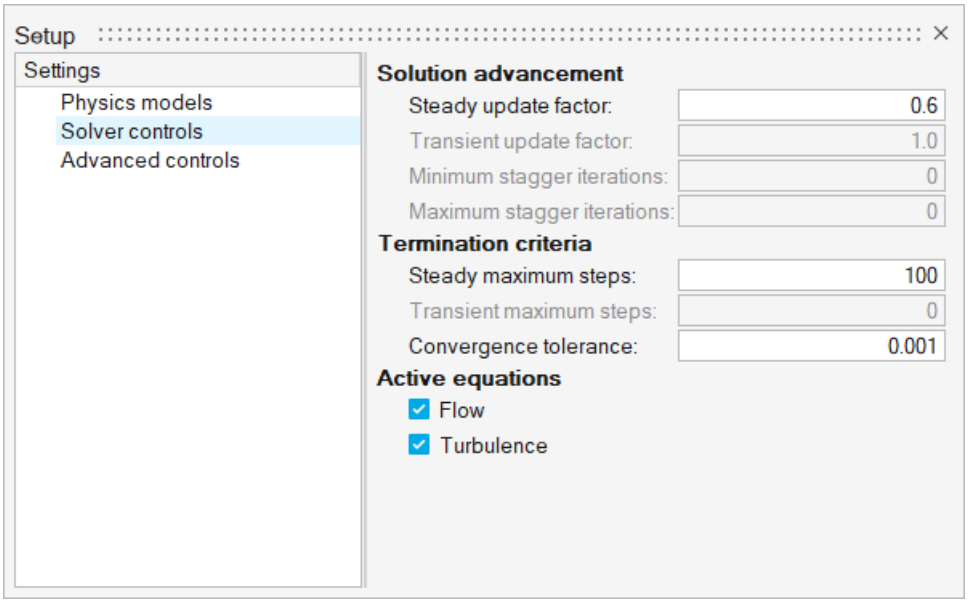


Figure 99:

5. Close the dialog and save the model.

Verify the Material Selection

1. From the Flow ribbon, click the **Material** tool.

Figure 100:

2. Select the model body.
   The entire solid is highlighted.

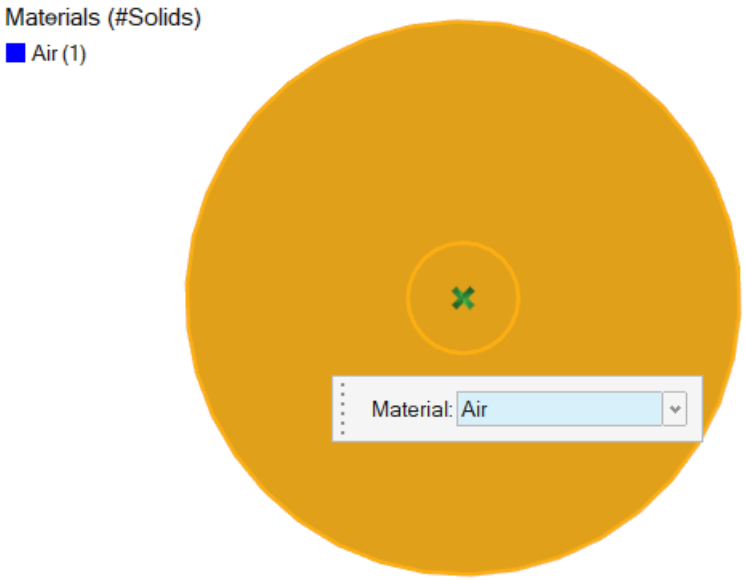


Figure 101:

3. In the microdialog check that the material **Air** is assigned by default.
4. Click the **Material Library** tool to verify the material properties of air.

Figure 102:

Air should have the following properties.

Density: 1.225
Specific Heat: 1005
Viscosity: 1.781e-5
Thermal Conductivity: 0.02521

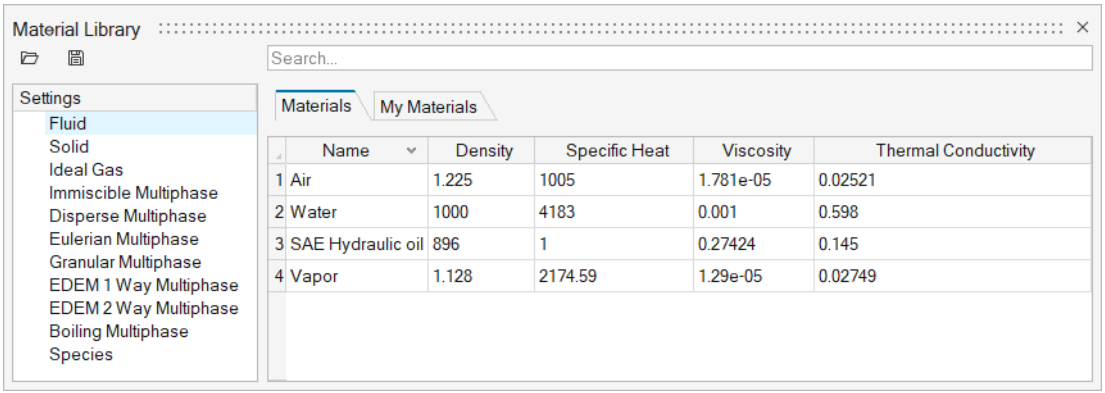


Figure 103:

5. Close the dialog.

Define Parameters

1. From the Flow ribbon, click the arrow next to the Setup tool set, then select **Parameters**.

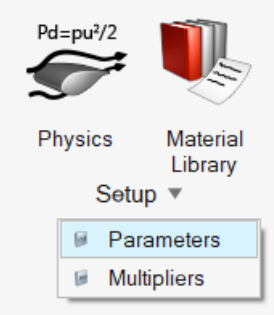


Figure 104:

The **Parameter Manager** opens, which allows you to create variables and expressions which can be used later in the setup.

2. Click + to create a new parameter. Set the Name to `xvelocity` and the Expression to `14.48345108`.
3. Create another new parameter: `yvelocity = 1.26713777823`

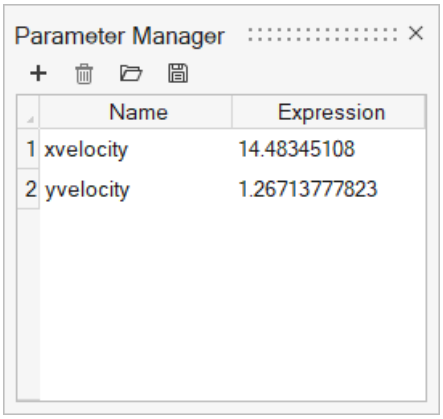


Figure 105:

4. Close the dialog.

Assign Flow Boundary Conditions

Set a Boundary Condition for the Far Field

1. From the Flow ribbon, click the **Far Field** tool.

Figure 106:

2. Rotate the geometry slightly so that the side-surfaces are visible. Click on the two half-side surfaces as shown in the figure below. In the microdialog, type `xvelocity` and `yvelocity` in the respective velocity fields. Accept all other default parameters

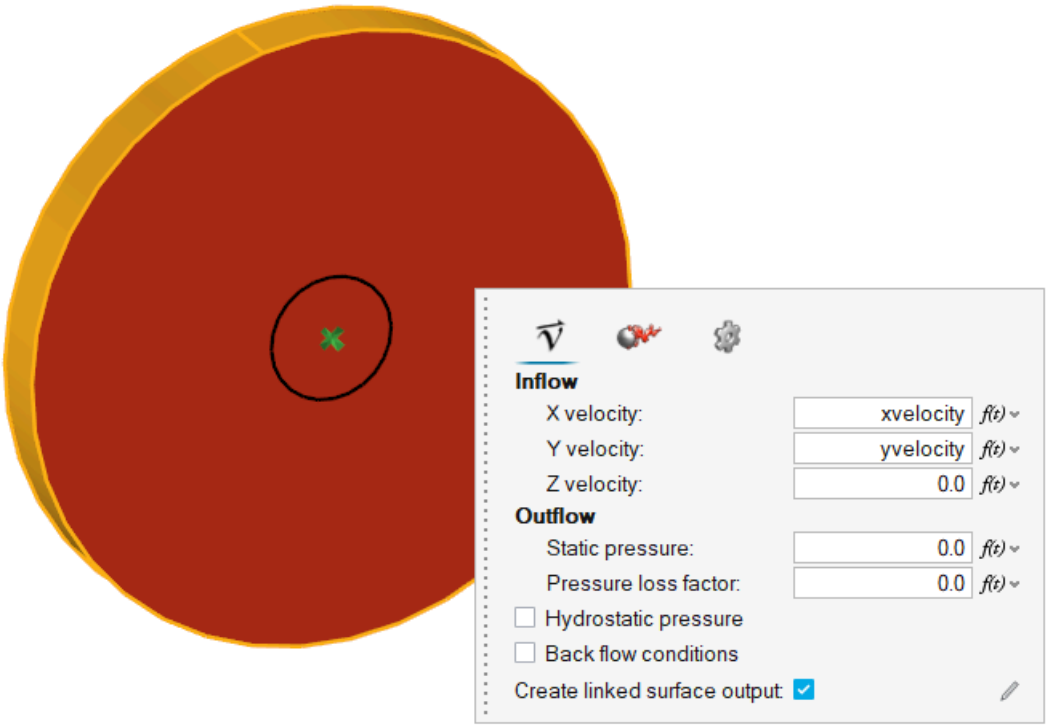


Figure 107:

3. Click the **Turbulence** tab in the microdialog and select **External** for the Turbulence flow type. Accept all other default parameters.

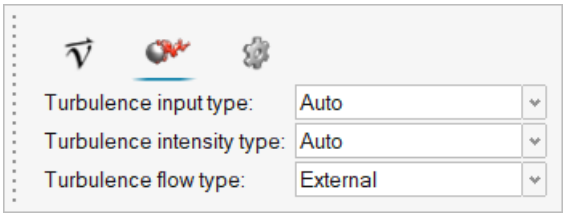


Figure 108:

4. Click ▸ on the guide bar.
The Boundaries legend shows the status of the updated event.

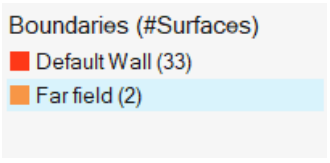


Figure 109:

5. On the guide bar, click ✓ to execute the command and exit the tool.

Set Slip Boundary Conditions

1. Click the **Top** face of the View Cube to orient the geometry.

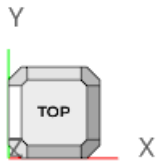


Figure 110:

2. Click the **Slip** tool.

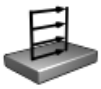


Figure 111:

3. Select the two surfaces shown in the figure below. Accept all default parameters in the microdialog.

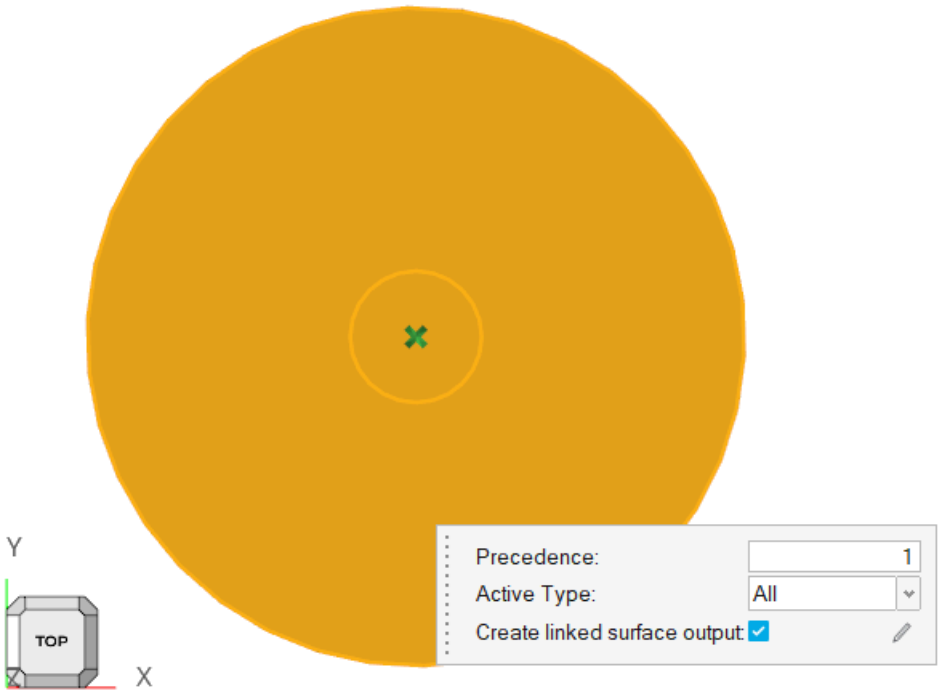


Figure 112:

4. Click ▸ on the guide bar.

5. Click the **Top** face of the View Cube once more to orient the geometry to the bottom view.
   Once the view is set, clicking the view again aligns the geometry to the opposite view.
6. Select the surface and accept all default parameters in the microdialog.

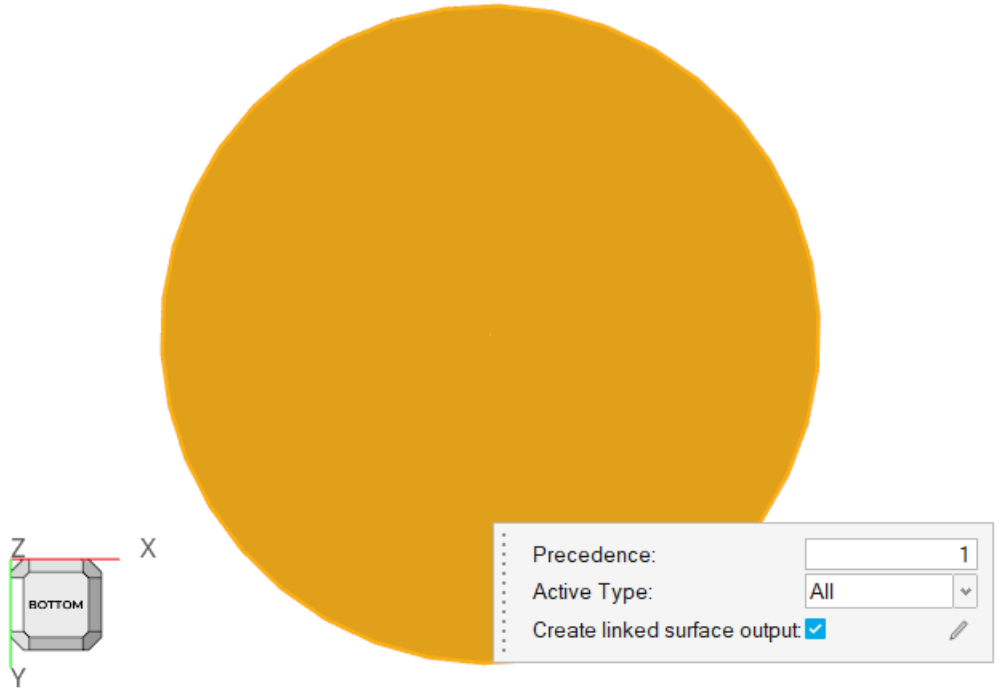


Figure 113:

7. Click ▸ on the guide bar.
   The Boundaries legend should now look like the following:

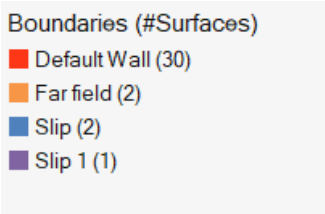


Figure 114:

8. Click the **Bottom** face of the View Cube to re-align the geometry with the top view.
9. Using the Boundaries legend, rename Slip (2) to `+Z slip` and Slip (1) to `-Z slip`.
10. On the guide bar, click ✓ to execute the command and exit the tool.

Set Wall Boundary Conditions

1. Click the **No Slip** tool.

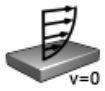


Figure 115:

2. Scroll to zoom in so that the airfoil surface is clearly visible.
3. Right-click in the modeling window and choose **Select** > **Advanced Selection** > **By Boundaries** > **Default Wall** to select the entire airfoil profile.

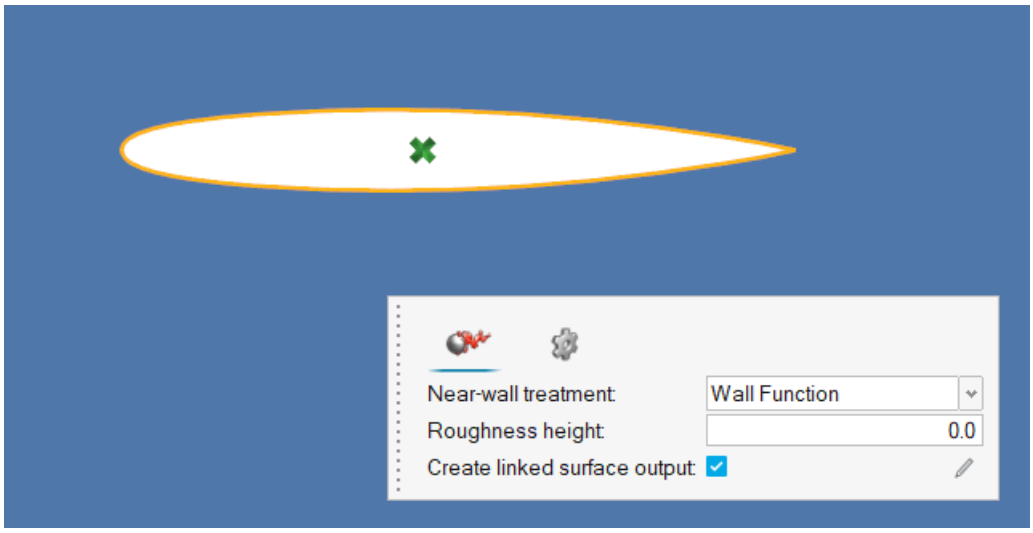


Figure 116:

4. Click ▸ on the guide bar.
5. Using the Boundaries legend, rename Wall (30) to `Airfoil`.
6. On the guide bar, click ✓ to execute the command and exit the tool.
7. Click the **Top** face of the View Cube to re-align the model.

Define Mesh Properties

Set Surface Mesh Controls

1. From the Mesh ribbon, click the **Surface** tool.

Figure 117:

2. Select the outer circular surface shown in the figure below.

3. In the microdialog, set the Average element size to `20` and make sure the Minimum size factor (0.1), Geometric feature angle (22.5), and Mesh growth rate (1.2) are entered as shown below.

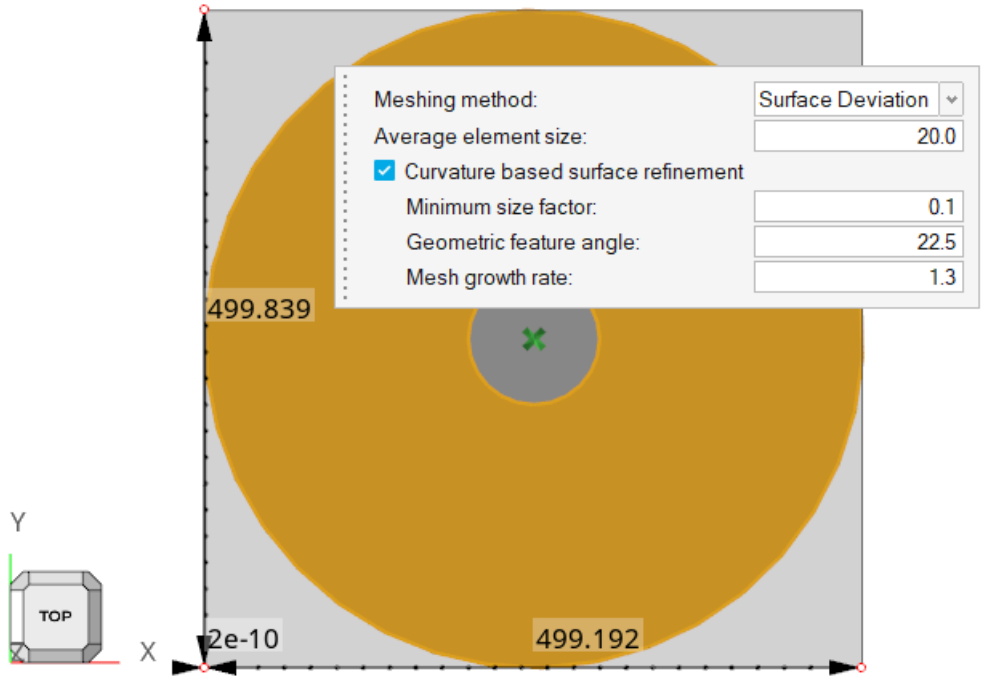


Figure 118:

4. Click ▸ on the guide bar.

5. Select the inner circular surface shown in the figure below.

6. In the microdialog, set the Average element size to `2` and the Minimum size factor to `0.001`. This ensures a smooth transition of mesh size of the airfoil.

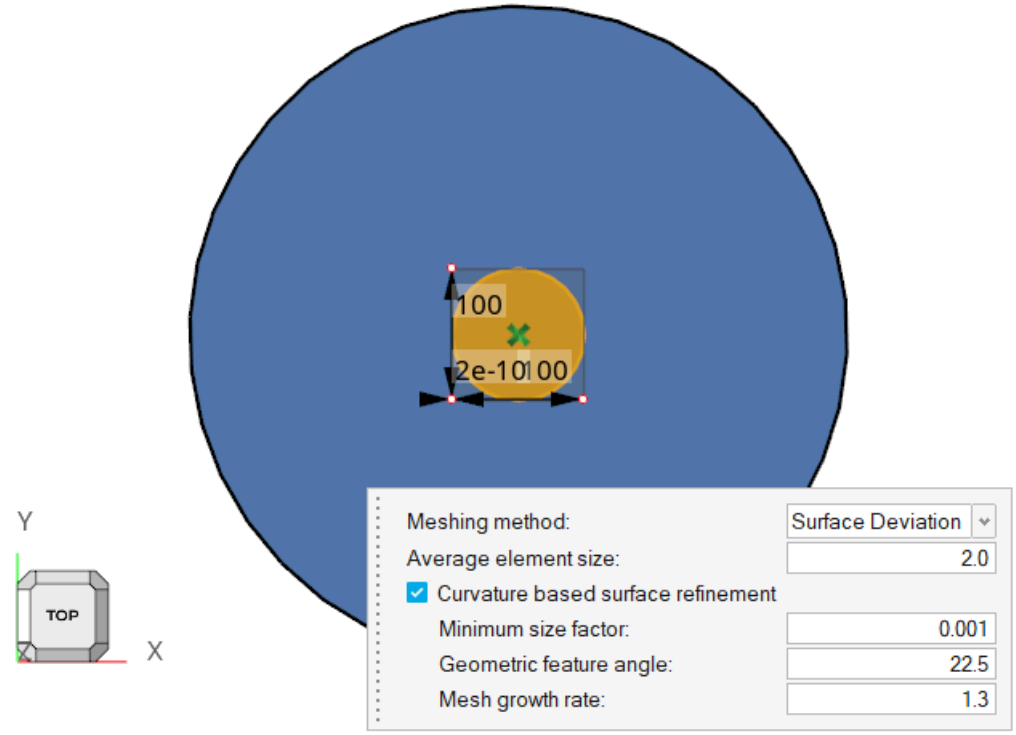


Figure 119:

7. Click ▸ on the guide bar.
The Surface Controls legend shows the status of the updated events.

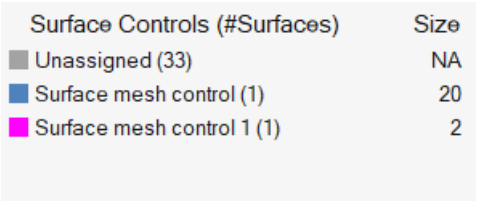


Figure 120:

8. On the guide bar, click ✓ to execute the command and exit the tool.

Set Edge Mesh Controls

1. Click the **Edge** tool.

Figure 121:

2. Zoom in on the airfoil.

Figure 122:

3. Right-click in the modeling window and select **Select** > **Advanced Selection** > **By Mesh Controls** > **Surface Mesh Controls 1**.
   This also selects the edge of the smaller circle, which will be unselected in the next step. The total number of lines selected is (31).
4. In the microdialog, enter a value of `0.002` for Average element size.

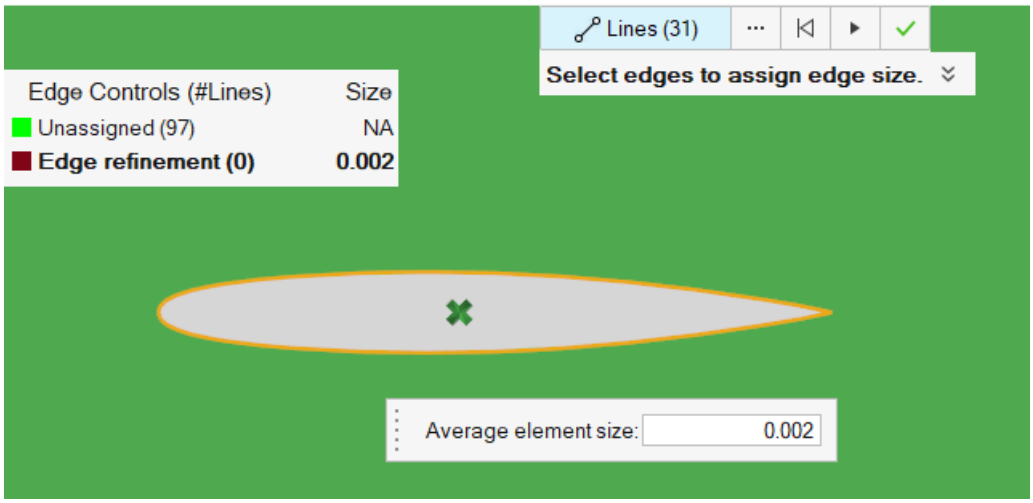


Figure 123:

5. Click ▸ on the guide bar.
6. Zoom out until the edge of the smaller circle appears.
7. Right-click **Edge refinement(31)** and select **Edit**. Keep the cursor on the highlighted edge of the smaller circular. Press Shift and left click to deselect the smaller circular edge so that the total lines selected becomes (30).

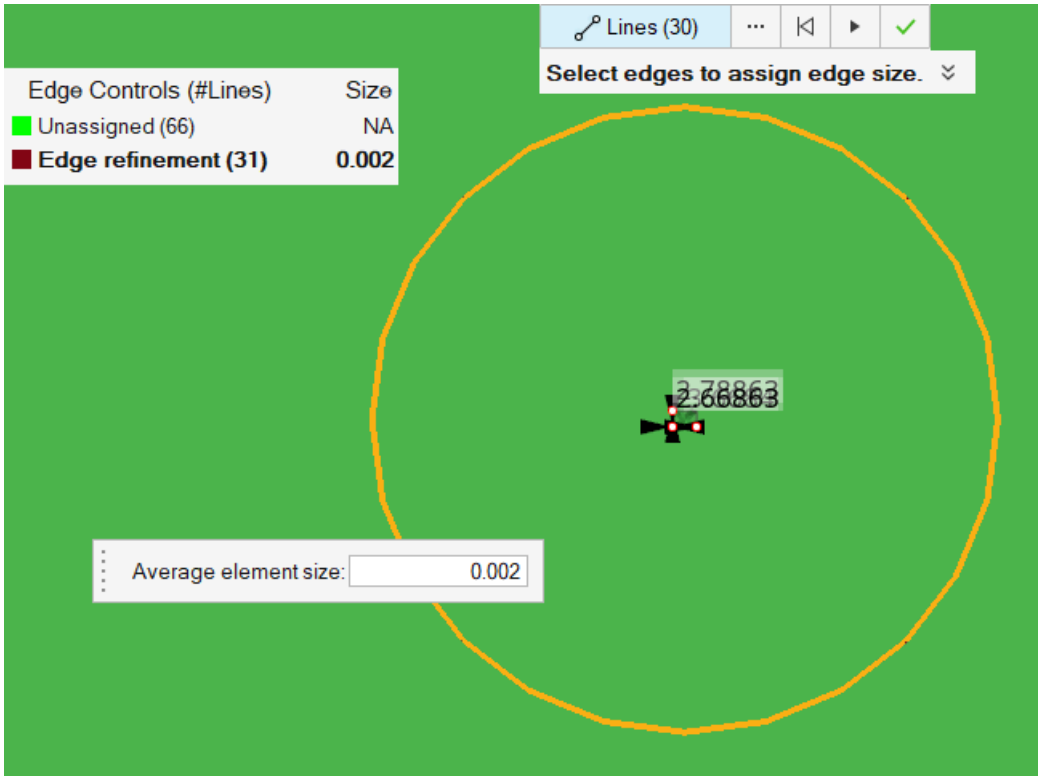


Figure 124:

8. Click ▸ on the guide bar.
9. Keep the mouse on the smaller circle edge line and click it. Enter the value of 2 for the Average element size.

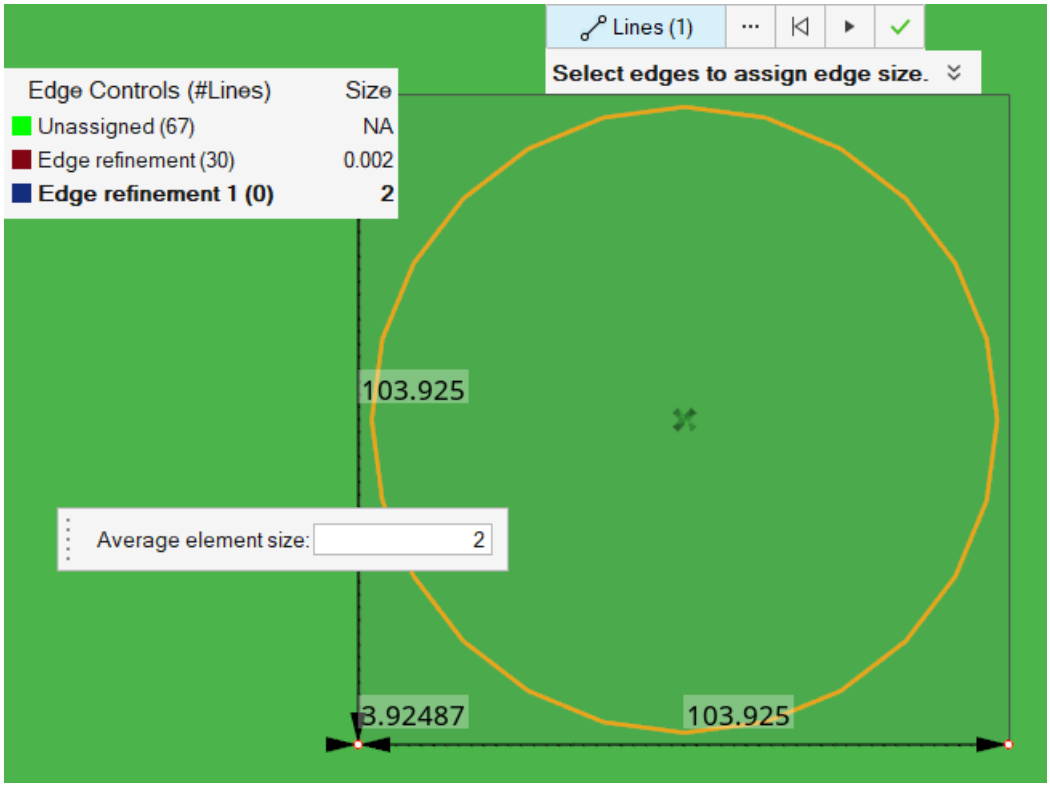


Figure 125:

10. Click ▸ on the guide bar.

11. Click the **Top** face of the View Cube.

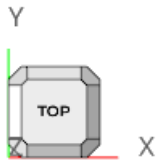


Figure 126:

12. Select the two outer edges of the outer circle and enter a value of `20` for the Average element size.

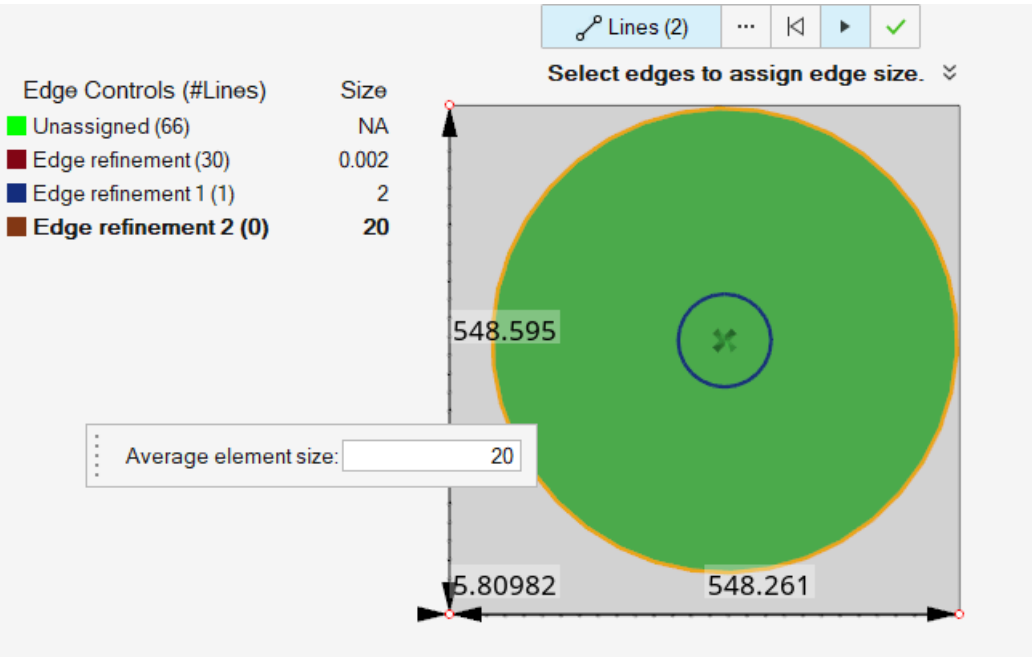


Figure 127:

13. Click ▸ on the guide bar.
The Edge Controls legend shows the status of the updated events.

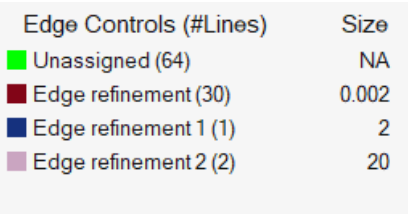


Figure 128:

14. On the guide bar, click ✓ to execute the command and exit the tool.

Set Edge Layer Controls

1. Zoom in on the airfoil.
2. Click the **Edge Layer** tool.

Figure 129:

3. On the guide bar, click ⋯ besides the Edges selector.

4. In the **Advanced Selection** dialog, select **By Mesh Controls** and enable **Edge refinement**.

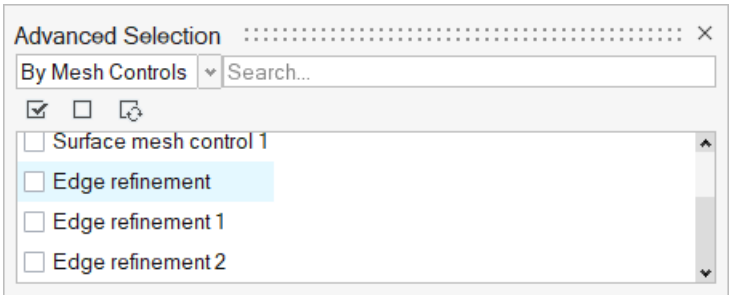


Figure 130:

5. In the microdialog, select **Constant** for First layer thickness definition. Enter a value of `0.0001` for First layer thickness and `20` for the Total number of layers for generating mesh boundary layers. Select **Constant** for Growth method, enter a value of `1.2` for Growth rate, and verify a value of `0.8` for Final Layer Height/Base Ratio. Accept all other default parameters.

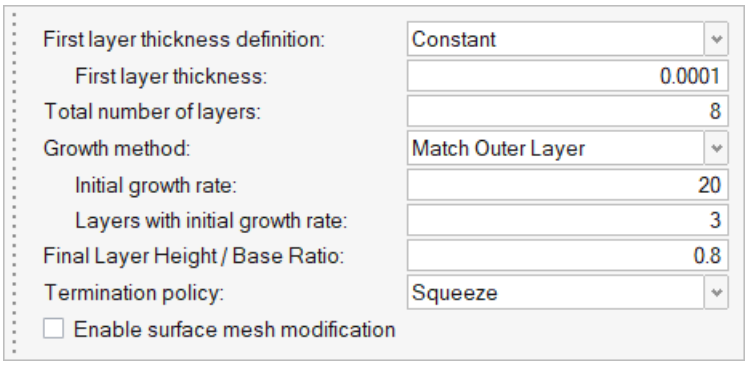


Figure 131:

6. Close the **Advanced Selection** dialog then click **Parent Surface** on the guide bar and select the smaller circular surface (around the airfoil) as shown below.

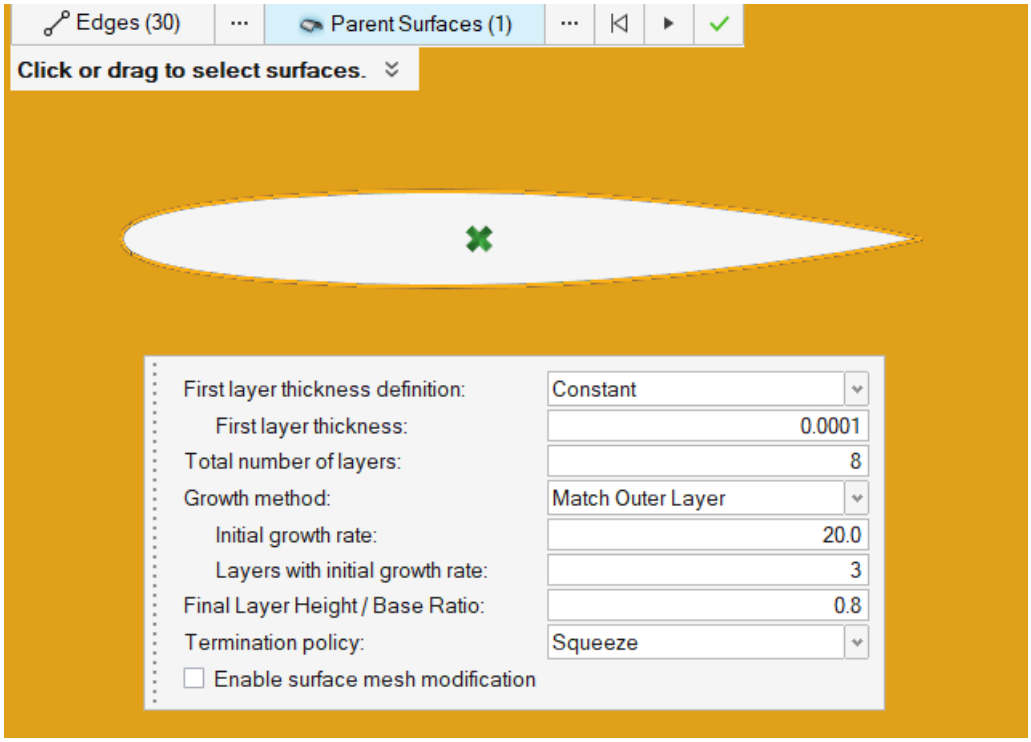


Figure 132:

7. Click ▸ on the guide bar.
   The 2D Boundary Layer Controls legend shows the status of the updated events.

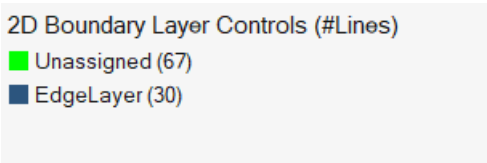


Figure 133:

8. On the guide bar, click ✓ to execute the command and exit the tool.
9. Click the **Top** face of the View Cube.

Set Extrusion Controls

1. Click the **Extrusion** tool.

Figure 134:

2. Click **Find** on the guide bar.
   This finds all the solids with mappable surfaces for extrusion.
3. Click **Solid (1)**.
   This selects all solids automatically.
4. Click **Source** then select the model surface.
5. In the microdialog, enter a value of `50` for Element Size Along Extrusion and `0.6` for Element size on the surfaces attached to targets. Accept all other default parameters.

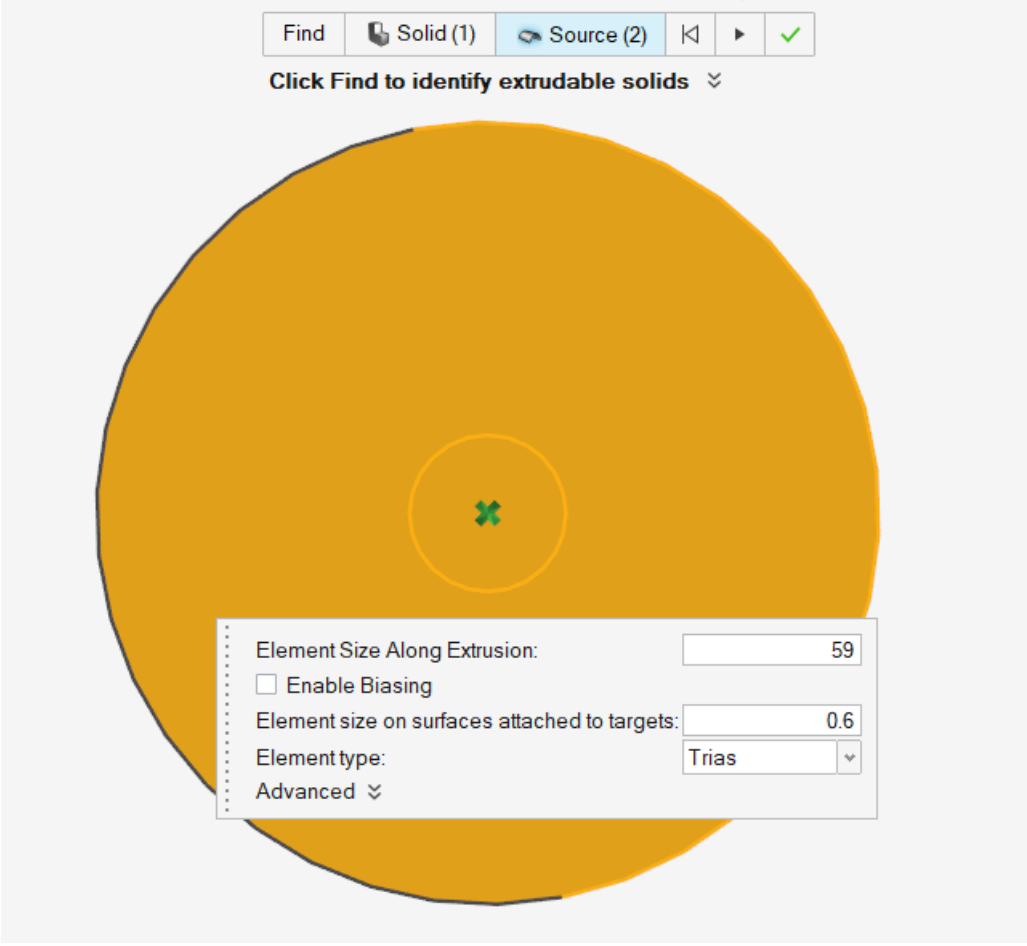


Figure 135:

6. Click ▸ on the guide bar, observe the change in the legend, then click ✓.

7. Save the database.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

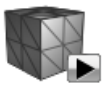


Figure 136:

The **Meshing Operations** dialog opens.

2. Check that the Average element size is 20. Accept all other default parameters.

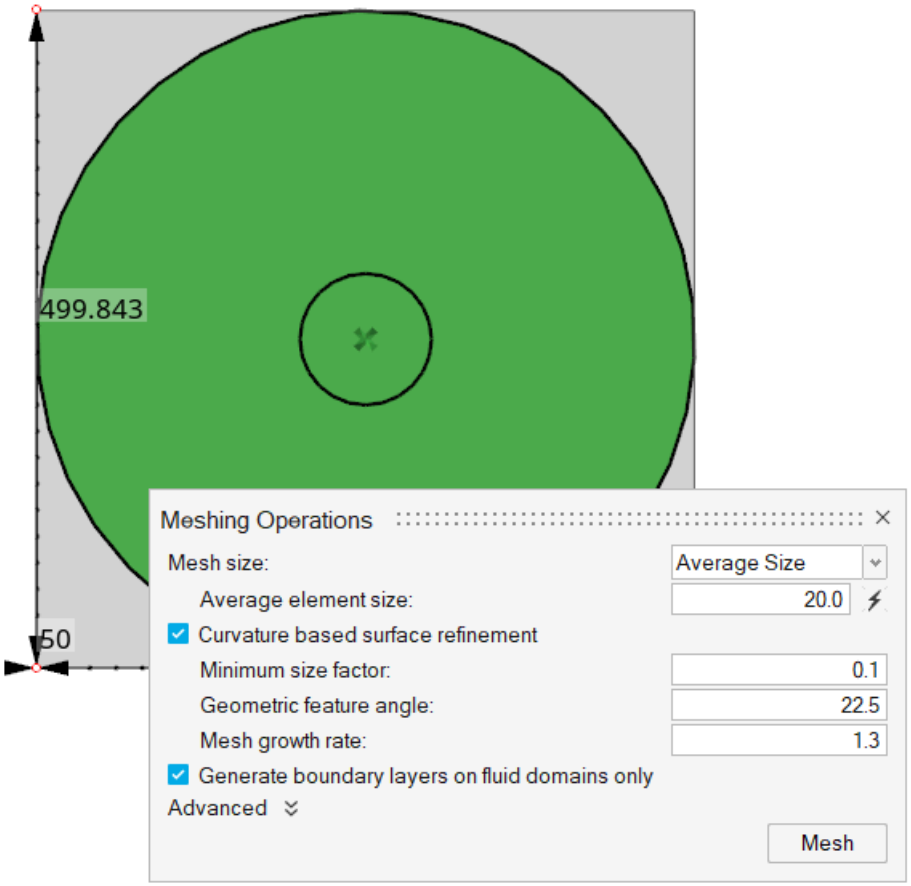


Figure 137:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Click **Mesh Only** on the View Controls toolbar.
   This is used to enable or disable the visualization of mesh.

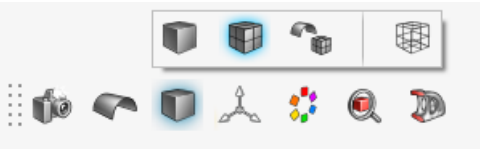


Figure 138:

5. Zoom in to check the mesh layers closer to the airfoil surface.

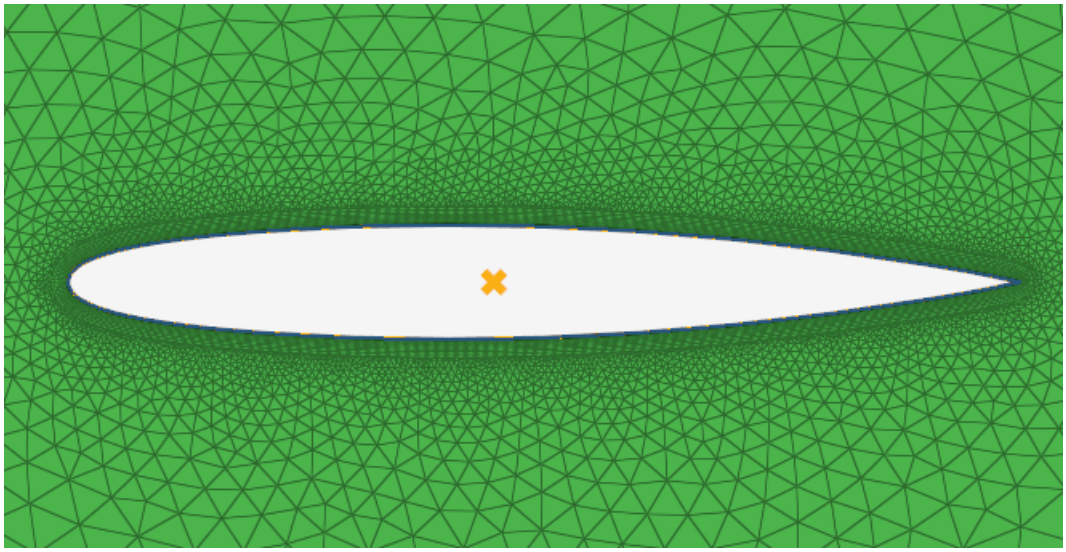


Figure 139:

6. Zoom towards the trailing edge of the airfoil to view the mesh.

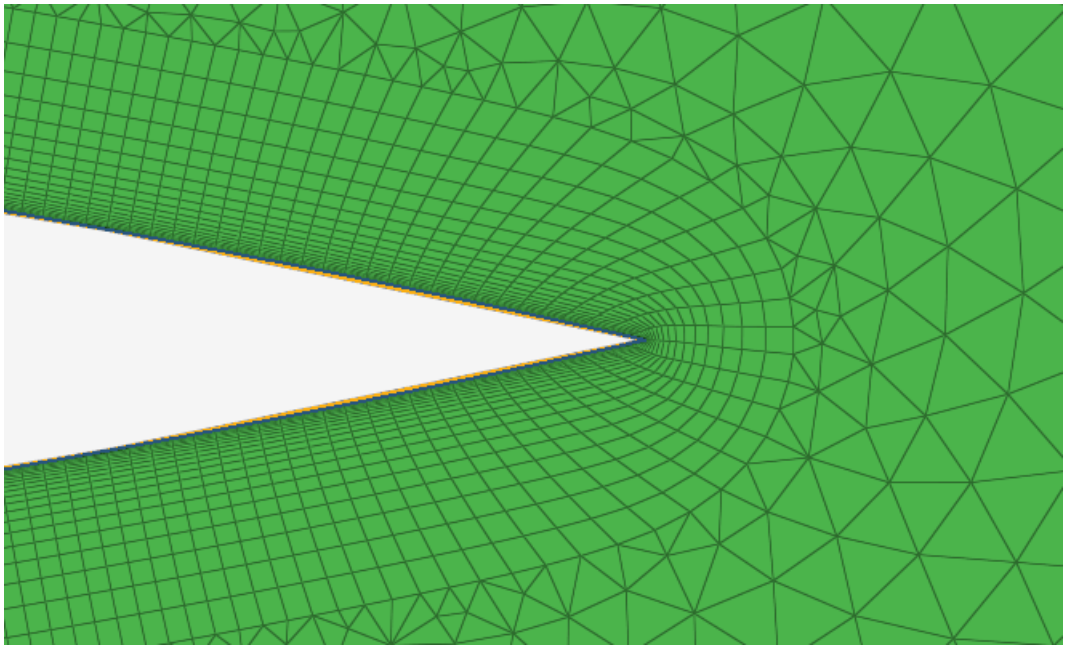


Figure 140:

7. From the Home tools, click the **Count** tool to get the node and element counts.

Figure 141:

Entity Count

Entity	Count
Points	66
Lines	97
Surfaces	35
Solids	1
Parts	8
Nodes	57568
2d Elements	73648
3d Elements	36280

Figure 142:

Note: The entity count may differ slightly every time a mesh is generated.

8. Close the dialog and save the database.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 143:

The **Launch AcuSolve** dialog opens.

2. Set the Problem name to `NACA0012`.
3. Set the Parallel processing option to **Intel MPI**.
4. Optional: Set the number of processors to **4** or **8** based on availability.
5. Deactivate the **Automatically define pressure reference** option.
6. Expand **Default initial conditions**, turn off **Pre-compute flow**, and set x-velocity yo `14.48345108` and y-velocity to `1.26713777823`. Enable **Pre-compute turbulence**.

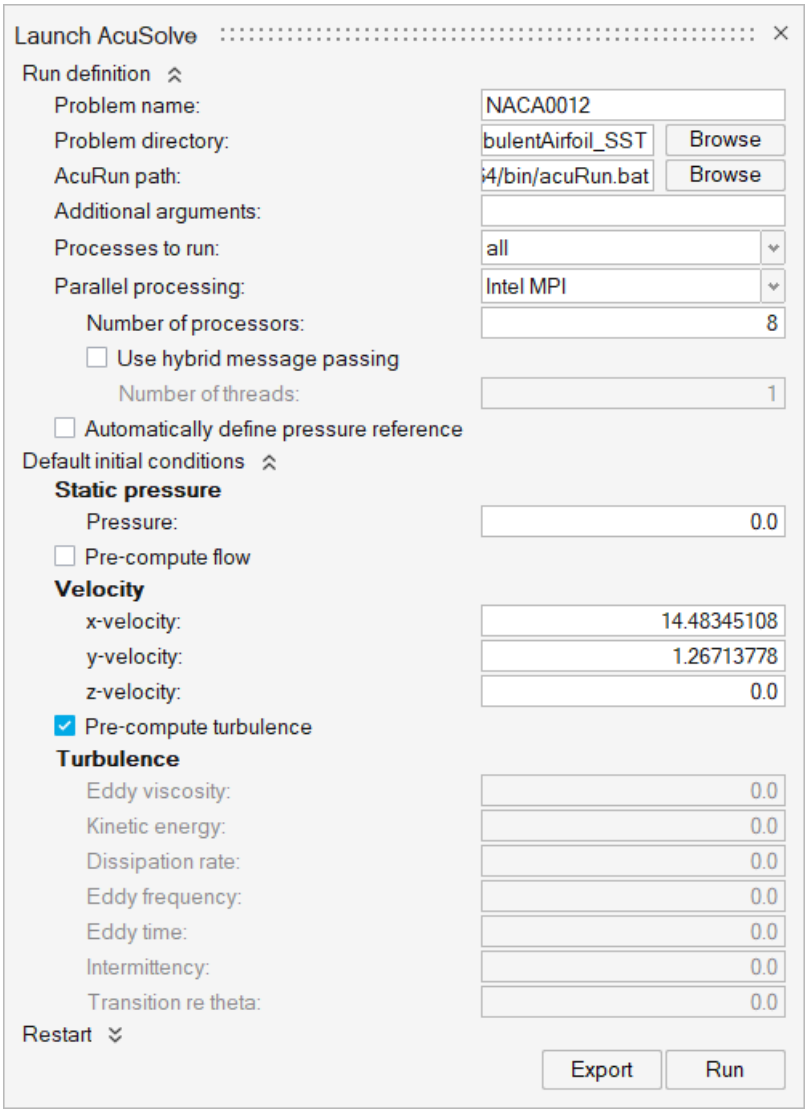


Figure 144:

7. Click **Run**.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, right-click on the AcuSolve run in the **Run Status** dialog and select **Plot time history**.

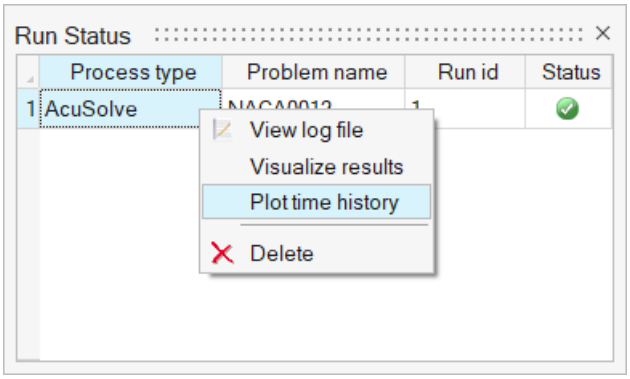


Figure 145:

The **Plot Manager** shows the Residual and Solution Ratios.

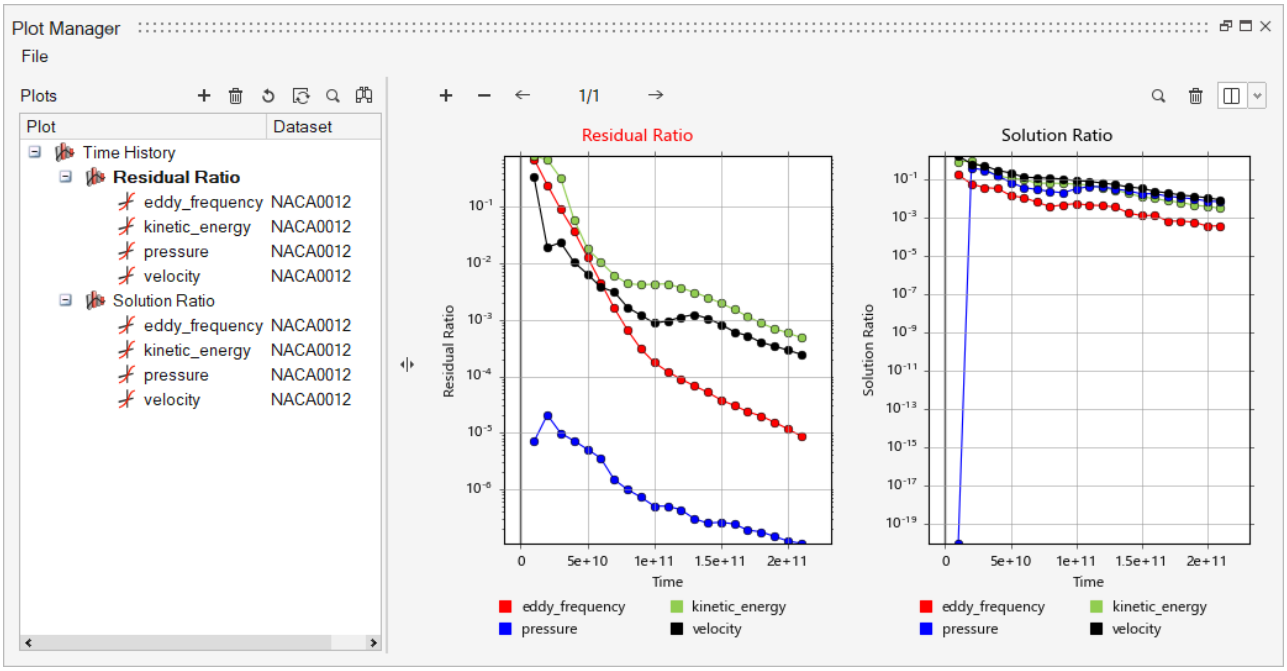


Figure 146:

2. Close the dialog.
3. Right-click on the AcuSolve run again and select **Visualize Results**.
   You are taken to the Post ribbon.
4. Click the **Slice Planes** tool.

Figure 147:

5. Select the x-y plane in the modeling window.

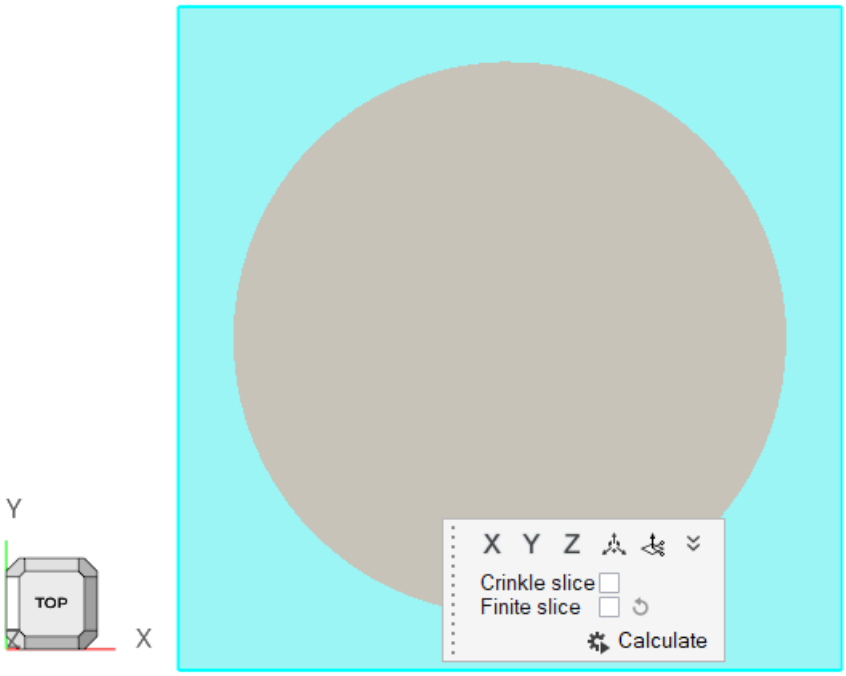


Figure 148:

6. Click to create the slice plane.
7. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
8. Click and set the Colormap Name to **Rainbow Uniform**.

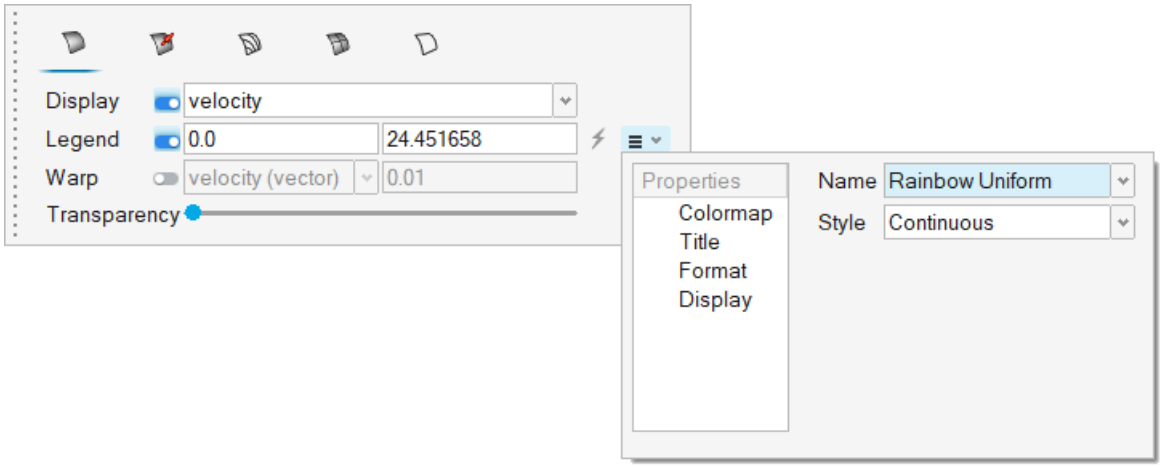


Figure 149:

9. Click ▸ on the guide bar to visualize the sqrt eddy period contour.

10. In the Post Browser, click the icon beside **Flow Boundaries** to turn off the display for all the flow boundaries.

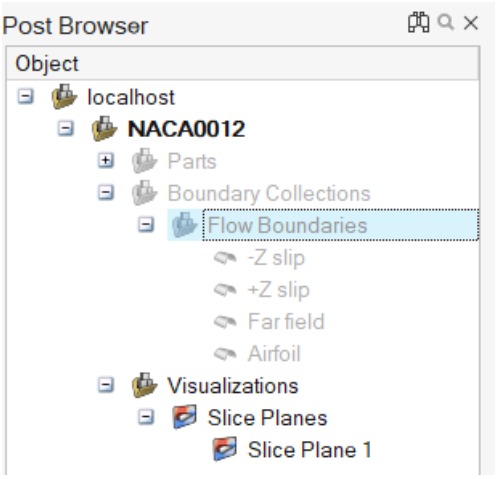


Figure 150:

11. On the guide bar, click ✓ to execute the command and exit the tool.

12. Zoom in on the airfoil to see the contour.

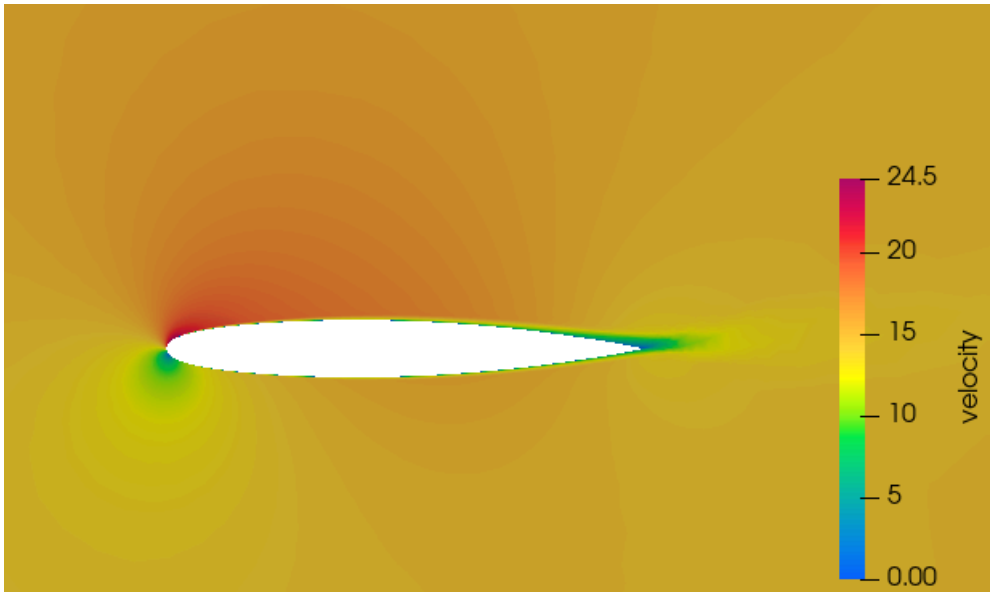


Figure 151:

Post-Process to Calculate Flow Coefficients

AcuSolve is shipped with a number of utility scripts to facilitate the pre and post-processing of a problem solved using the solver. You will be introduced to two of these scripts, AcuLiftDrag and AcuGetCpCf in this section, and their usage. These two scripts are focused on aerodynamic simulations as the ones solved in this tutorial.

Run AcuLiftGrag

AcuLiftDrag is a utility script used to calculate the lift and drag coefficients for an airfoil.

In the Problem Description section, it was described that the simulation is performed as 2D by including only a single layer of extruded elements in the airfoil span direction. When solving a problem in such a way, the span of the airfoil should be set equal to the thickness of the domain in the extrusion direction. When solving a 3D problem, the actual span of the airfoil should be used.

To execute the AcuLiftDrag script for this position, follow the steps below:

1. Start AcuSolve Command Prompt from the Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt**.
2. Change the directory to the present problem directory using the `cd` command.
3. Enter the following command at the prompt:

   ```
   acuLiftDrag -osis "Airfoil - Output" -aoa 5 -ref_vel 14.54 -chord 1 -ref_rho 1.225
   -span 50
   ```

   The output of the command should look like the image below:

```
acuLiftDrag:
acuLiftDrag:  Opening the AcuSolve solution data base
acuLiftDrag:  Problem <NACA0012> directory <ACUSIM.DIR> runId <0>
acuLiftDrag:
acuLiftDrag:  Opened run id <1>
acuLiftDrag:
acuLiftDrag:  Processing surface output variable:  traction
acuLiftDrag:    Processing surface output:  Airfoil - Output
acuLiftDrag:    Writing file: NACA0012.liftDrag.dat
acuLiftDrag:
acuLiftDrag:    C_l - final: 0.476861907157
acuLiftDrag:      C_l - min: 0.283576968887
acuLiftDrag:      C_l - max: 0.476861907157
acuLiftDrag:      C_l - ave: 0.400990531205
acuLiftDrag:      C_l'- rms: 0.0556047469165
acuLiftDrag:
acuLiftDrag:    C_d - final: 0.014310610452
acuLiftDrag:      C_d - min: 0.0142556317488
acuLiftDrag:      C_d - max: 0.156371719106
acuLiftDrag:      C_d - ave: 0.029874435835
acuLiftDrag:      C_d'- rms: 0.0330518280759
acuLiftDrag:
acuLiftDrag:  Total elapsed time <0.0180> seconds
acuLiftDrag:
```

Figure 152:

The value C_l corresponds to the lift coefficient, and C_d corresponds to the drag coefficient. The final keyword refers to the respective coefficient values at the last time step of the simulation. The rest of the values provide the basic statistics for these coefficients over all the time steps of the simulation. These statistics are more meaningful if the simulation is transient.

The script also creates the file `NACA0012.liftDrag.dat` in the problem directory. The file contains the lift and drag data for all the available time steps in a determined tabular arrangement. The first column is the time step, the second column is the lift coefficient, and the third column is the drag coefficient.

AcuGetCpCf

AcuGetCpCf is another utility script used to calculate the coefficient of pressure (Cp) and coefficient of friction (Cf) for the airfoil.

To execute the AcuGetCpCf script for this problem, follow the steps below:

1. Start AcuSolve Command Prompt from the Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt**.
2. Change the directory to the present problem directory using the `cd` command.
3. Enter the following command at the prompt to extract Cp and save it to the file with the `.dat` extension

   ```
   acuGetCpCf -osis "Airfoil - Output" -type cp -ref_vel 14.54 -ref_rho 1.225 -no_nc -ref_pres 0 -cut_dir z -rad_locs 0.0
   ```

   The *`cut_dir`* option describes the cut plane axis to calculate the Cp and Cf values. *`rad_locs`* describes the location of the cut plane on the *`cut_dir`* axis. The script prints the minimum and maximum values for the pressure coefficient. It also creates a file, `NACA0012.cp.1.dat`, in the problem directory. The file contains the pressure coefficient data along the chord of the airfoil. The first column is the x-coordinate along the chord, and the second column is the pressure coefficient

The output of the command should look like the image below:

```
acuGetCpCf:
acuGetCpCf:  Opening the AcuSolve solution data base
acuGetCpCf:  Problem <NACA0012> directory <ACUSIM.DIR> runId <0>
acuGetCpCf:
acuGetCpCf:  Opened run id <1>
acuGetCpCf:
acuGetCpCf:  Extracting <pressure> field from step <21>
acuGetCpCf:
acuGetCpCf:  Projecting a circle of radius: 1.10e+000
acuGetCpCf:
acuGetCpCf:  Cut plane: x-y
acuGetCpCf:
acuGetCpCf:  Building projection object
acuGetCpCf:
acuGetCpCf:  Processing location:  1
acuGetCpCf:     Cut coordinate:             z = 0.0
acuGetCpCf:     Local chord:                1.000
acuGetCpCf:     Rotational velocity:        0.0
acuGetCpCf:     Total velocity:             14.54
acuGetCpCf:     Reference dynamic pressure: 1.29e+002
acuGetCpCf:     Cp min/max:                 -1.89249/1.00552
acuGetCpCf:     Chord Reynolds number:      1.00e+006
acuGetCpCf:     Writing file:               NACA0012.cp.1.dat
acuGetCpCf:
acuGetCpCf:
```

Figure 153:

4. Enter the following command at the prompt to plot the variation of Cp along the airfoil surface, or you can use an external plotting utility to the plot the data.

```
acuPlotData -files NACA0012.cp.1.dat -x_col 1 -y_cols 2 -no_pts -ptf -lfmt "-r" -
legend AcuSolve -title Cp_variation_along_the_chord -x_label x_along_the_chord -
y_label Cp
```

```
acuPlotData:  Plotting frame <1>
acuPlotData:
acuPlotData:  Evaluating argument <NACA0012.cp.1.dat>
acuPlotData:  Found <1> files
acuPlotData:  Processing file <NACA0012.cp.1.dat>
acuPlotData:  Plotting <940> points
acuPlotData:
acuPlotData:  Printing plot to file <NACA0012.pdf>
acuPlotData:
```

Figure 154:

The plotting result is shown below:

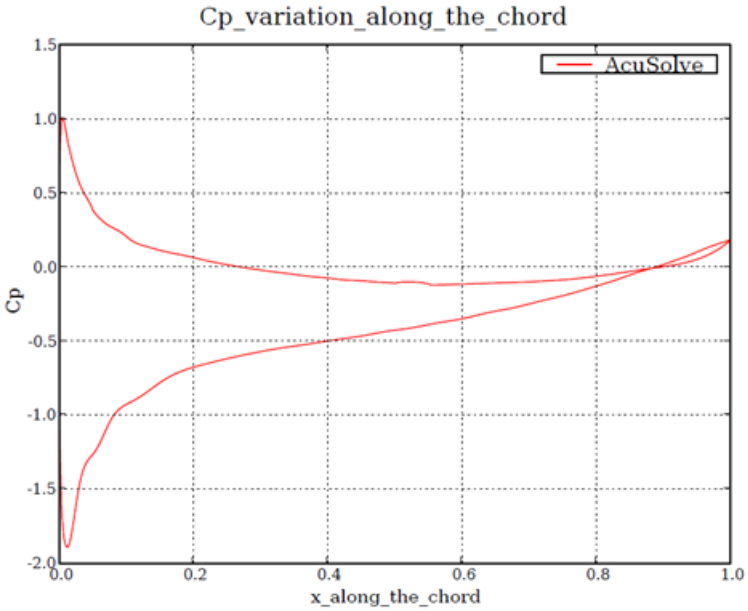


Figure 155:

Summary

In this tutorial, you learned how to set up and run a NACA0012 airfoil simulation using HyperMesh CFD and AcuSolve. You started by importing the input CAD file. Next, you assigned the flow boundary conditions and generated the mesh using extrusion. Once the solution was computed, you post-processed the results using HyperMesh CFD Post. Finally, you used a utility script of AcuLiftDrag to calculate the lift and drag coefficients of the airfoil and a utility script of AcuGetCpCfto calculate coefficient of pressure (Cp) and coefficient of friction (Cf). The variation of pressure coefficient along the airfoil surface was plotted using a plotting utility acuPlotData.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/rmitifskkt

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T2300_Building.hm`

Since the HyperMesh CFD database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 156. As an example, this problem shows the capability of Atmospheric Boundary Layer modelling in AcuSolve.

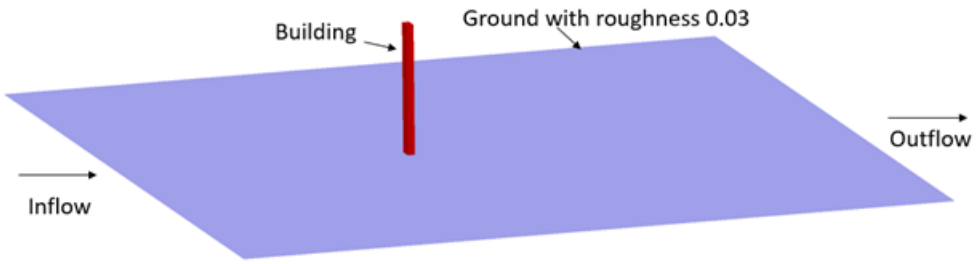


Figure 156:

In this tutorial, you will simulate the air flow over a building with a ground roughness of 0.03 m. In this case, User Defined Atmospheric Roughness Type is considered.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 157:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T2300_Building.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Building_ABL` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Building` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 158:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

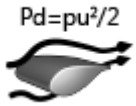


Figure 159:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that the **Incompressible** option is selected under Single phase flow.
   b) Set the Time marching to **Steady**.
   c) Select **Spalart-Allmaras** as the Turbulence model.

      The robustness and accuracy of the Spalart Allmaras turbulence model makes it an excellent choice for simulation of steady state flows.

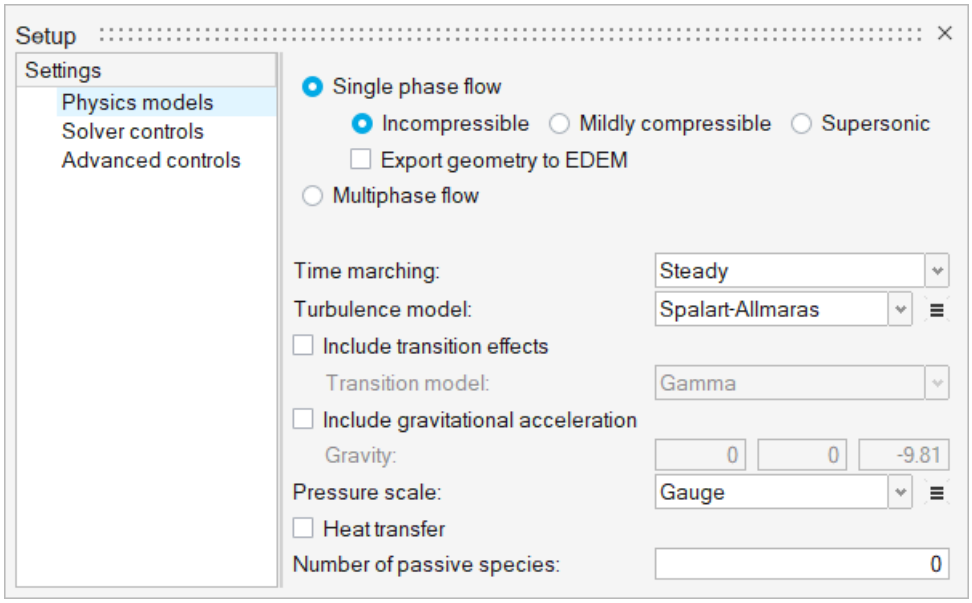


Figure 160:

3. Click the **Solver controls** setting.
4. Set the Steady update factor to `0.6` and the Steady maximum steps to `20`.

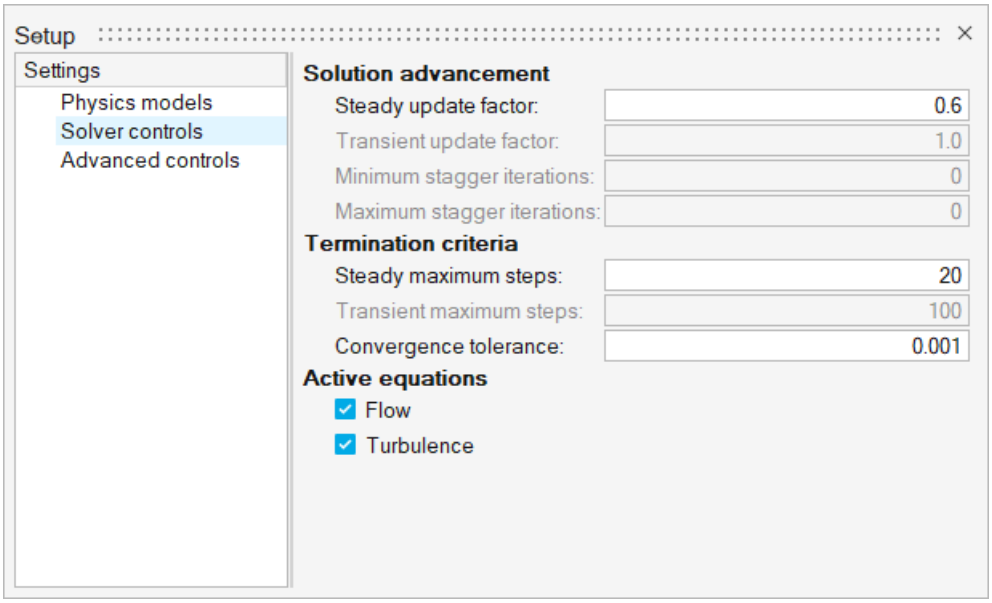


Figure 161:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 162:

2. Select the solid.
3. Select the **Air** material in the microdialog.
4. Click ✓ on the guide bar.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Atmospheric** tool.

Figure 163:

2. Click the inlet face highlighted in the figure below.

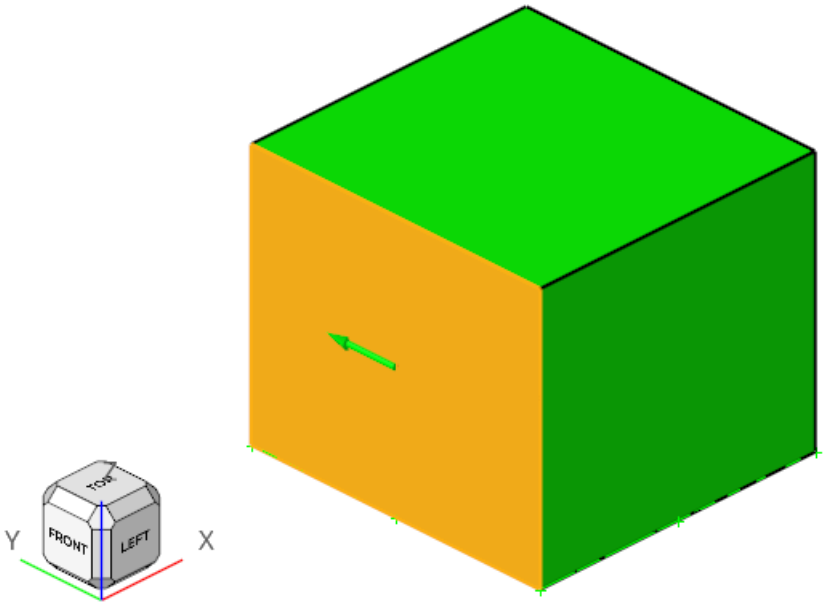


Figure 164:

3. In the microdialog, select **User Value** for the Ground roughness type.
4. Set the Friction Velocity to `0.106` m/s.

   The atmospheric boundary layer inflow should have a roughness of 0.001 m if the standard turbulence wall function has a roughness of 0.03 m on the ground because the relationship between inflow roughness and standard wall roughness has an empirical ratio of 30.
5. Click the **Move** tool icon .

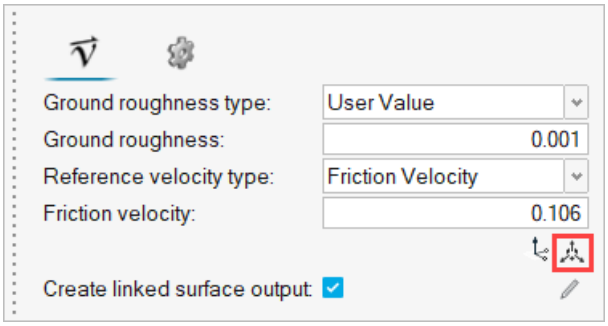


Figure 165:

6. Select the **z rotation** arrow, set it to `-90` degrees, then press **Enter**.

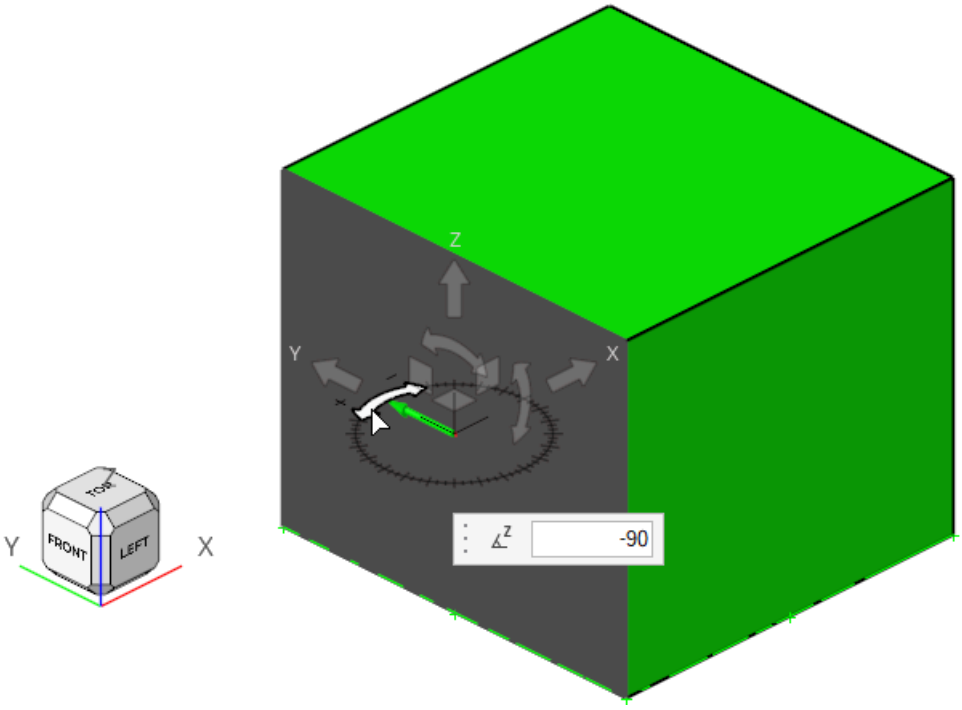


Figure 166:

7. Verify the atmospheric flow direction (y) is aligned to the global coordinate X.

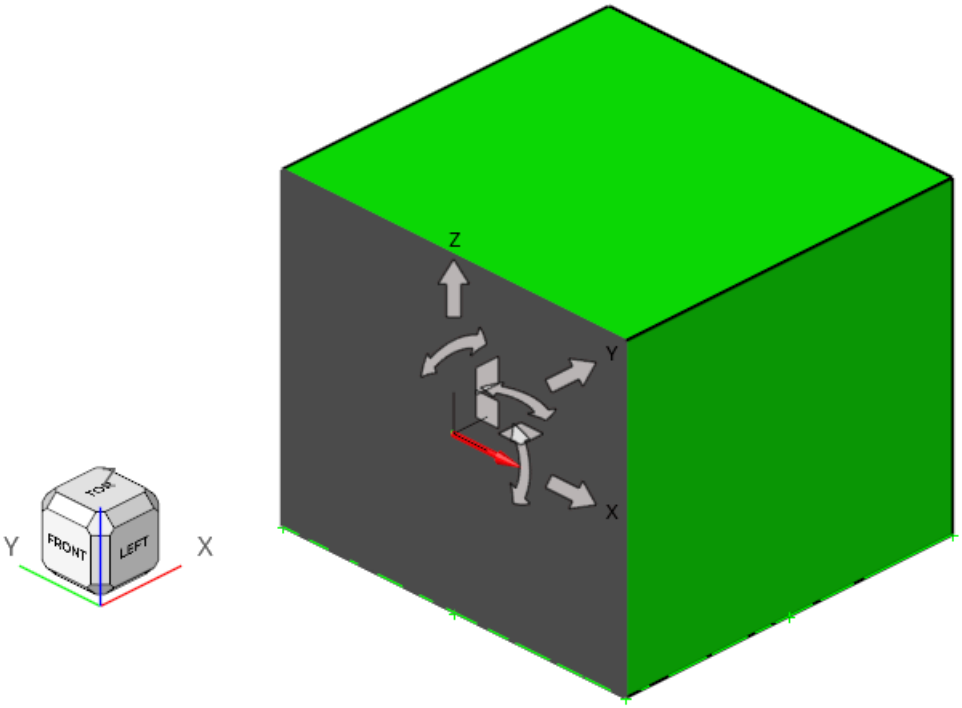


Figure 167:

8. Select the **y rotation** arrow, set it to `-90` degrees, then press **Enter**.

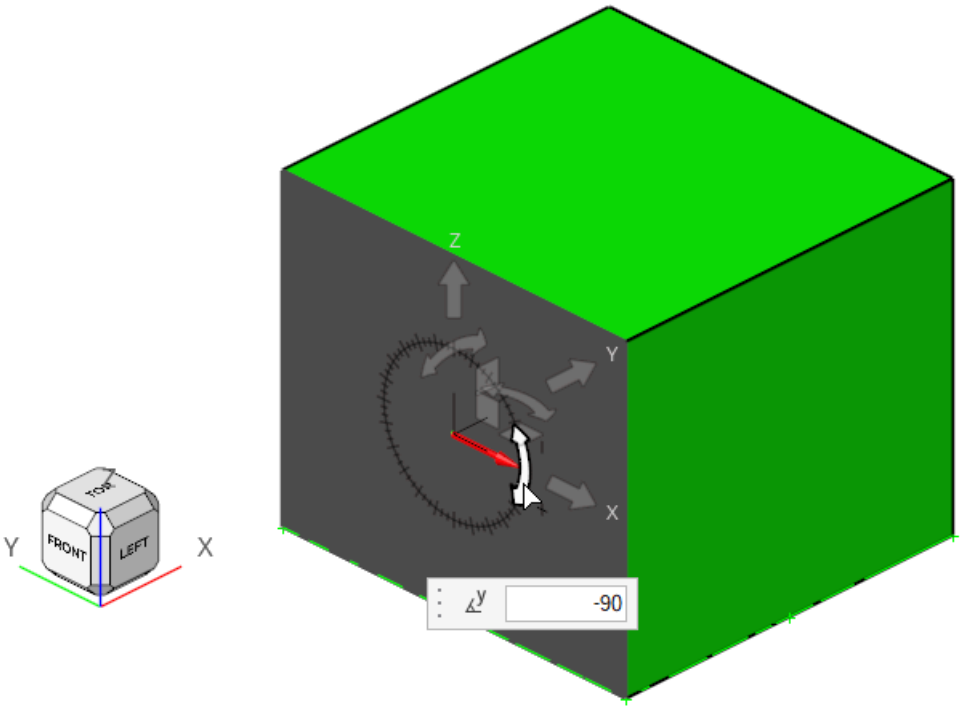


Figure 168:

9. Verify the ground normal direction (x) align to the global coordinate Z.

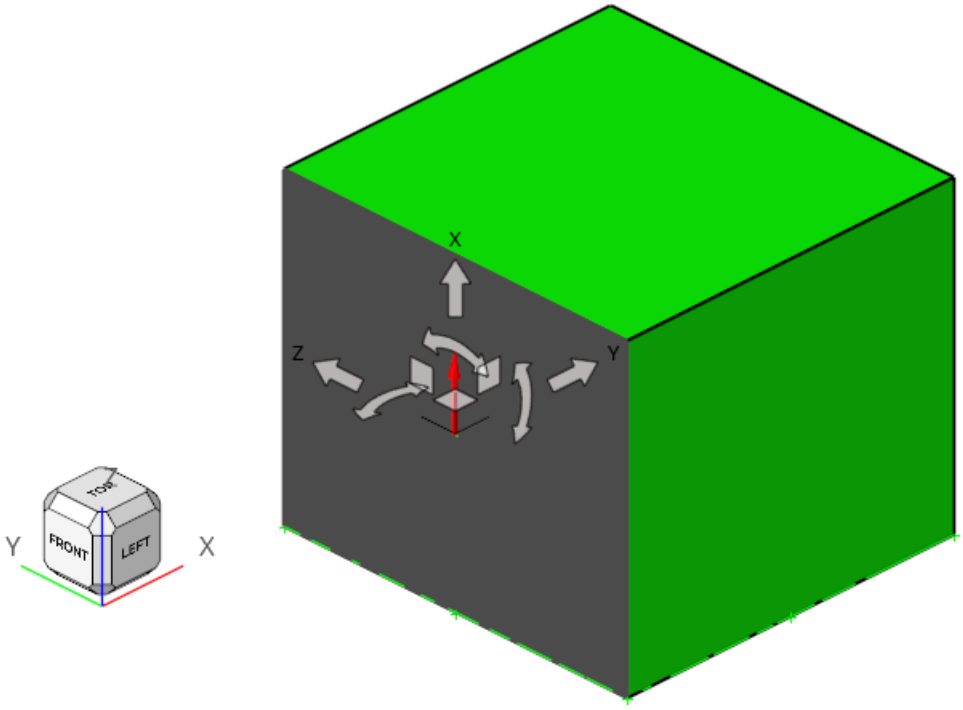


Figure 169:

10. Click the origin of the coordinate (marked by the arrow), set the atmospheric ground origin to `(0, 0, 0)`, and press **Enter**.

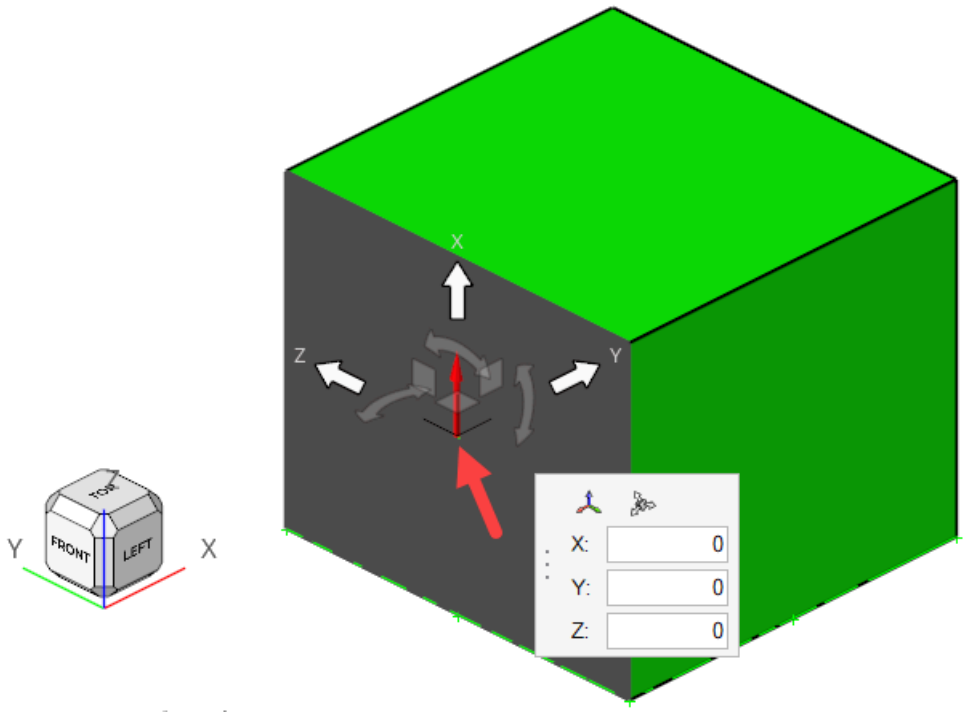


Figure 170:

11. Press **Esc** to return from the Move tool.

12. On the guide bar, click ✓ to execute the command and exit the tool.

13. Click the **Outlet** tool.

Figure 171:

14. Select the face highlighted in the figure below, verify that both the static pressure (Pa) and the pressure loss factor are `0` in the microdialog, then click ✓ on the guide bar.

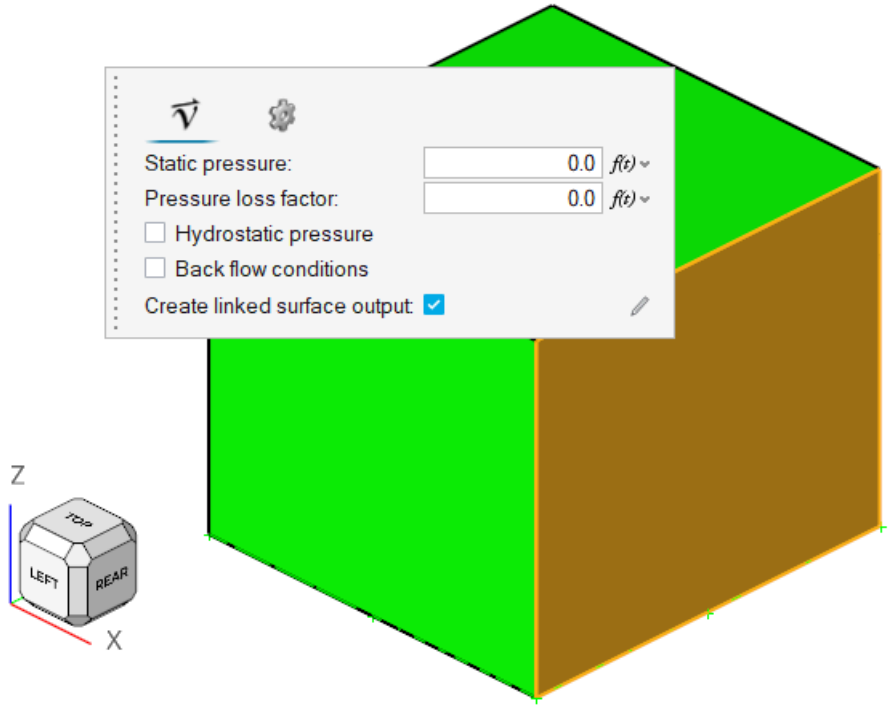


Figure 172:

15. Click the **Slip** tool.

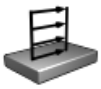


Figure 173:

16. Hide the **Atmospheric boundary** and the **Outlet** using the Boundaries legend.

17. Select the top surface and the two sides as shown in the figure below then click ✓ on the guide bar.

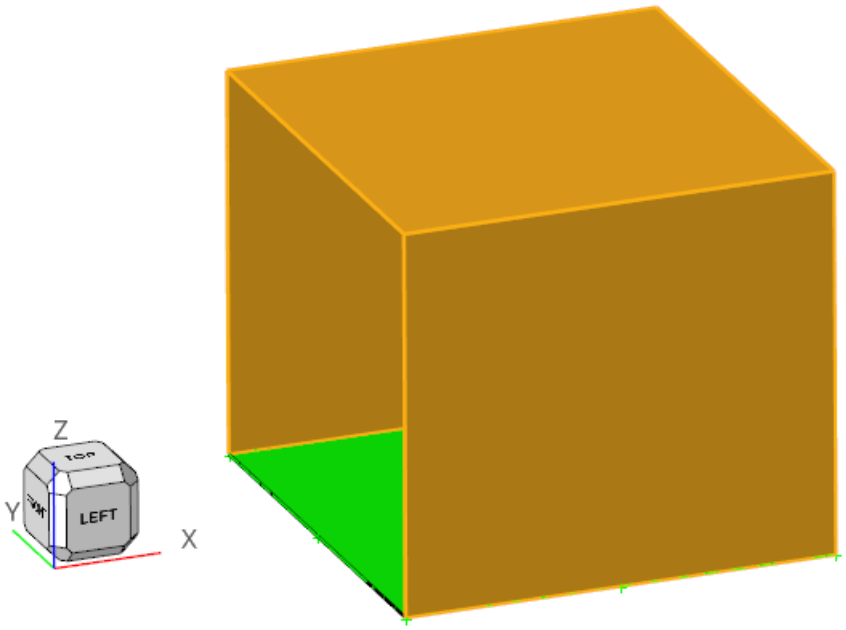


Figure 174:

18. Click the **No Slip** tool.

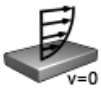


Figure 175:

19. Hide the **Slip** surfaces using the Boundaries legend.

20. Select the bottom surface as shown in the figure below.

21. Set the Roughness height to `0.03` m.

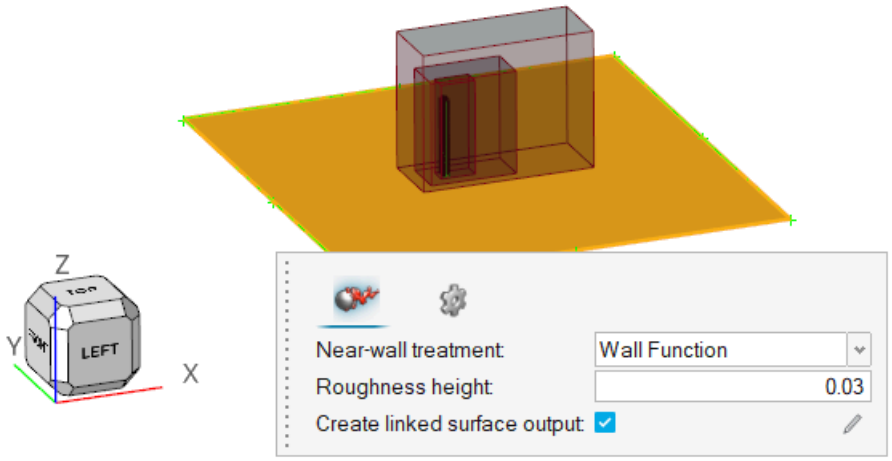


Figure 176:

22. On the guide bar, click ✓ to execute the command and exit the tool.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 177:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Uncheck the **Automatically define pressure reference** option.
5. Expand **Default initial conditions** and uncheck the **Pre-compute flow** and **Pre-compute turbulence** options.
6. Set all the velocity components to `0` m/s and the eddy viscosity to `0.0001` m^2/s.
7. Leave the remaining options as default and click **Run** to launch AcuSolve.

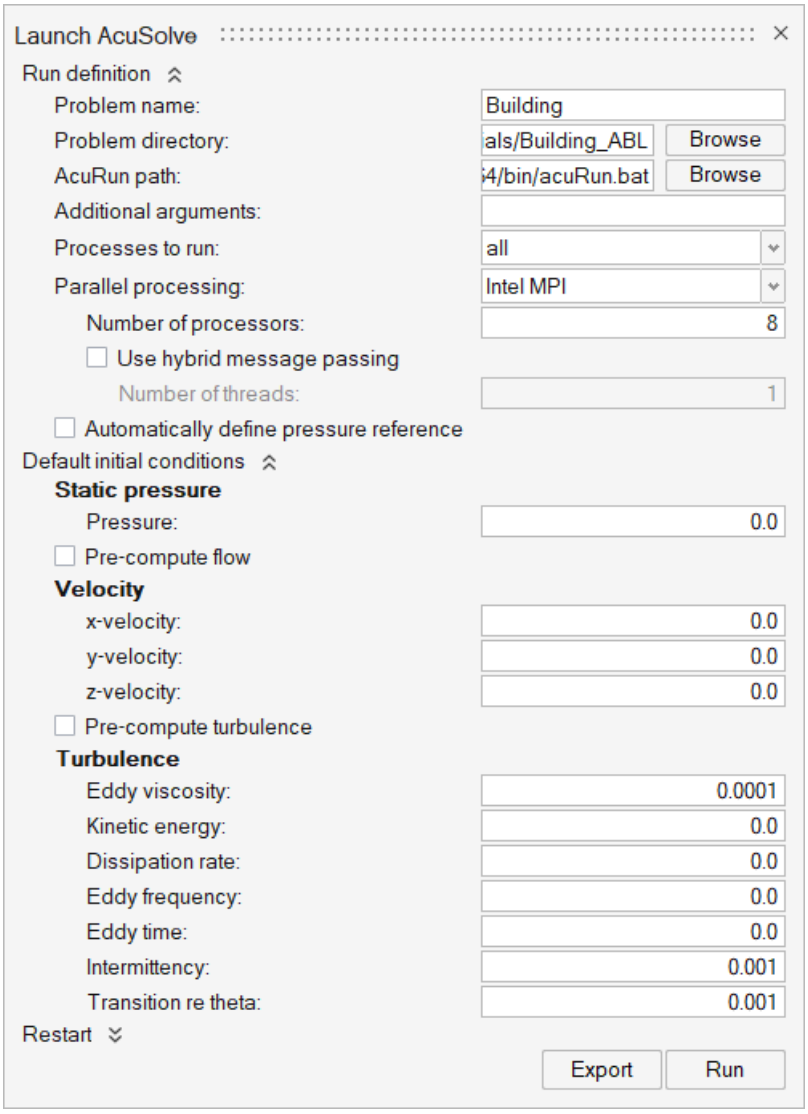


Figure 178:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

4. Click the **Slice Planes** tool.

Figure 179:

5. Select the slice plane shown below.

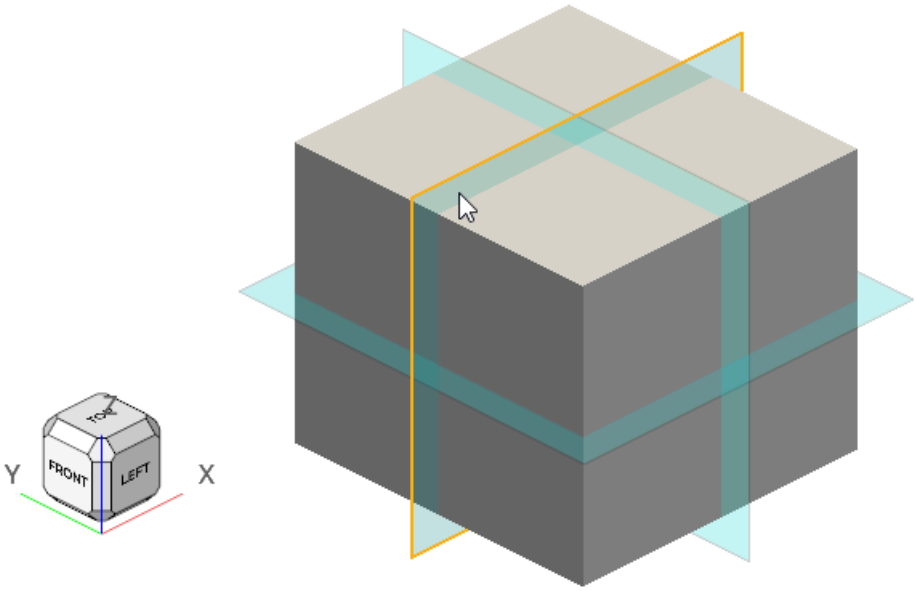


Figure 180:

6. In the slice plane microdialog, click to create the slice plane.
7. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
8. Click and set the Colormap name to **Rainbow Uniform**.

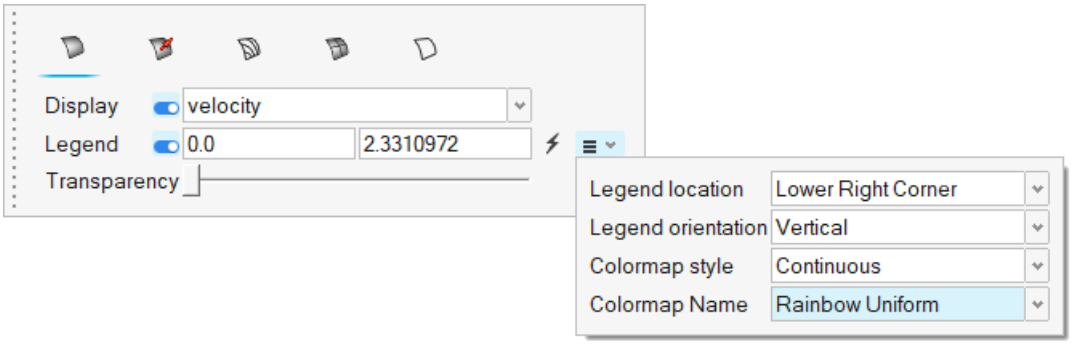


Figure 181:

9. Isolate **Slice Plane 1**.
10. Click the **Left** face on the View Cube to orient the slice plane.
11. Click ✓ on the guide bar to show the velocity (m/s) contours.

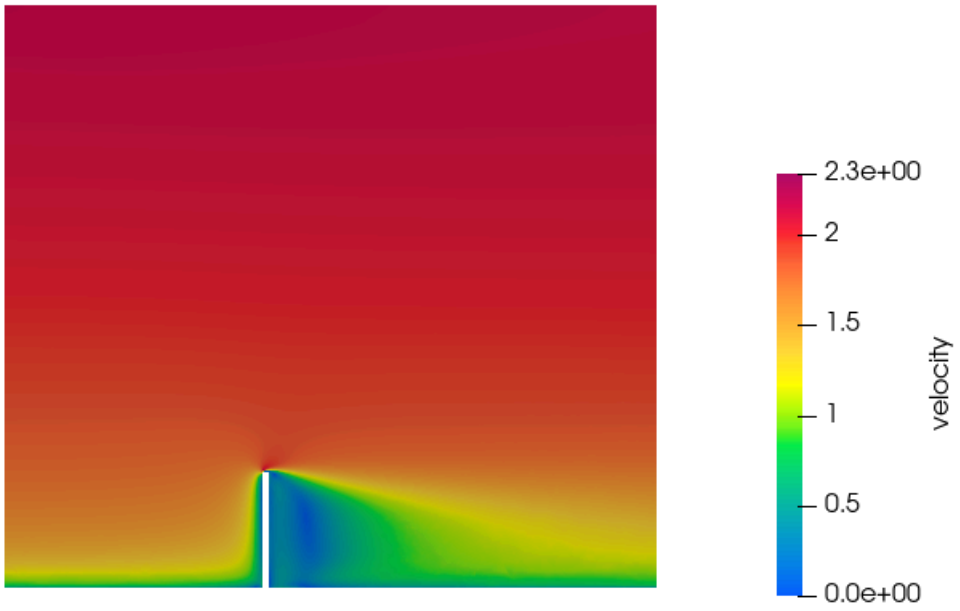


Figure 182:

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving an atmospheric boundary condition using HyperMesh CFD. You started by opening the HyperMesh CFD input file with the geometry and then defined the simulation parameters, fluid material, and boundary conditions. Once the solution was computed, you visualized the results of the velocity magnitude on a plane.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/3wjr645fc8

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 2400 Supersonic Flow in a Converging-Diverging Nozzle

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T2400_CD_Nozzle.hm`

Note: This tutorial does not cover steps related to geometry cleanup.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 183. It is based on the popular de Laval or converging-diverging nozzle. It consists of a tube that is pinched in the middle making an axisymmetric hourglass shape. Air at high pressure enters the inlets and the flow accelerates as the area of the nozzle decreases. The flow reaches sonic speed at the throat and becomes choked. A region of supersonic flow forms just downstream of the throat. Unlike a subsonic flow, the supersonic flow accelerates as the area gets bigger. This region of supersonic acceleration is terminated by a normal shock wave. The shock wave produces a near-instantaneous deceleration of the flow to subsonic speed. This subsonic flow then decelerates through the remainder of the diverging section and exhausts as a subsonic jet.

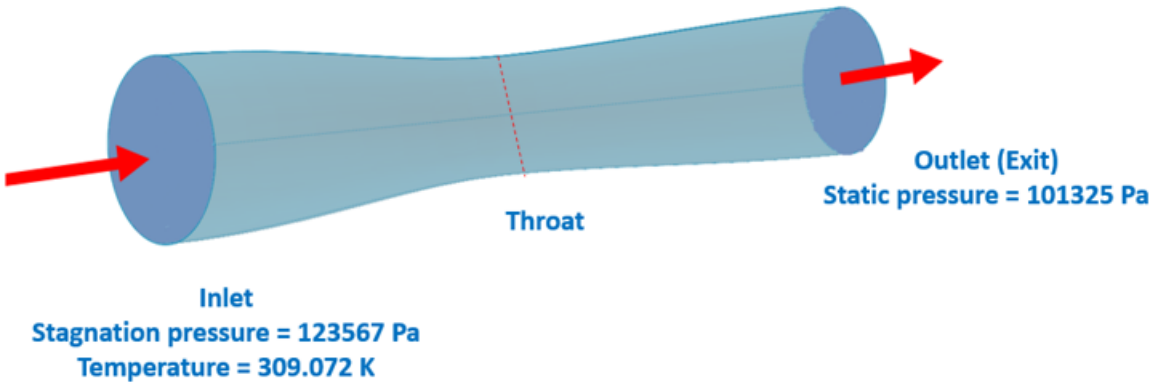


Figure 183:

The exit to throat area ratio of the nozzle is 1.5. The radius at the inlet is 0.892 m, while the radius at the outlet is 0.691 m. The inlet stagnation pressure and the inlet temperature are 123,567 Pa and 309.072 K, respectively. The static pressure at the outlet is set to 101,325 Pa. The fluid in this problem is air, where the flow is assumed inviscid (viscosity and conductivity are zero) and density is based on the ideal gas model.

The problem is rotationally periodic about the longitudinal axis, and it is assumed that the resulting flow is also rotationally periodic, allowing for modeling with the use of a wedge-shaped section. For this tutorial, a 60° section of the geometry is modeled, as shown in the figure. Modeling a portion of a rotationally periodic geometry leads to reduced computation time while still providing an accurate solution.

Figure 184:

The AcuSolve simulation will be set up to model a transient supersonic flow where the flow variables reach an asymptotic state to determine the stable flow solution.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 185:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T2400_CD_Nozzle.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `CD_Nozzle` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `CD_Nozzle` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 186:

Set Up Flow

Define Material Properties

1. From the Flow ribbon, click the **Material Library** tool.

Figure 187:

The **Material Library** dialog opens.

2. Under Settings, click **Ideal Gas**, and then click the **My Material** tab.
3. Click **+** to add a new ideal gas model.
4. Keep the default values for the Gas constant and the Specific heat and set the Viscosity and Conductivity values to `0`.
5. Rename the model to `Air Ideal Inviscid`.

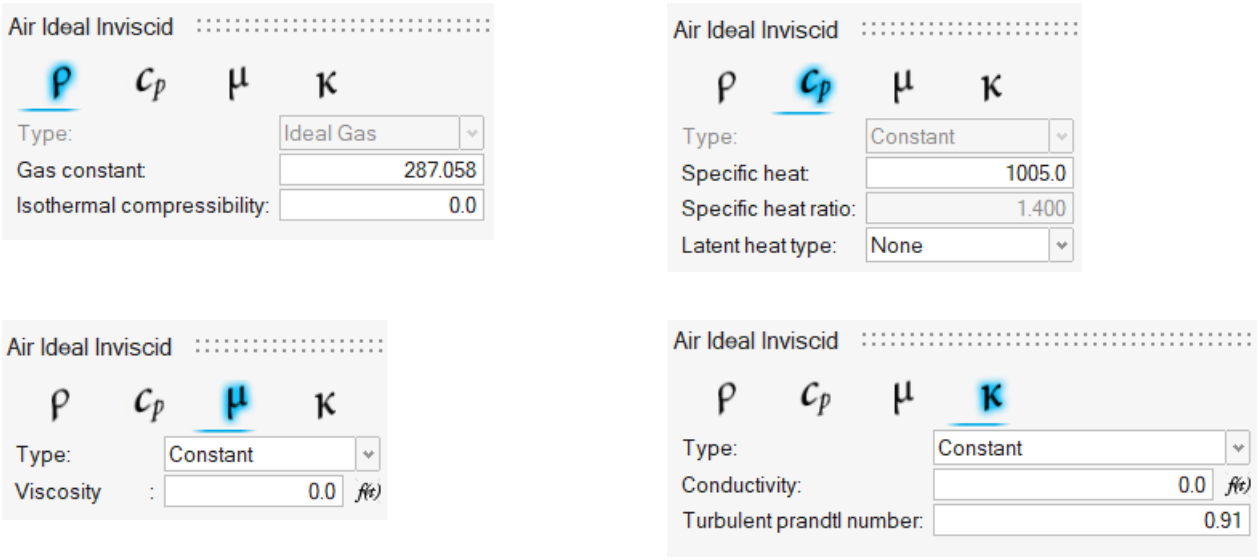


Figure 188:

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

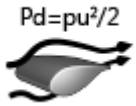


Figure 189:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Select the **Supersonic** radio button under Single phase flow.
   b) Set the Ideal gas model to **Air Ideal Inviscid**.
   c) Set the Time step size to `0.000125`.
   d) Set the Final time to `0.3125`.

e) Verify that the Turbulence model is set to **Laminar**.

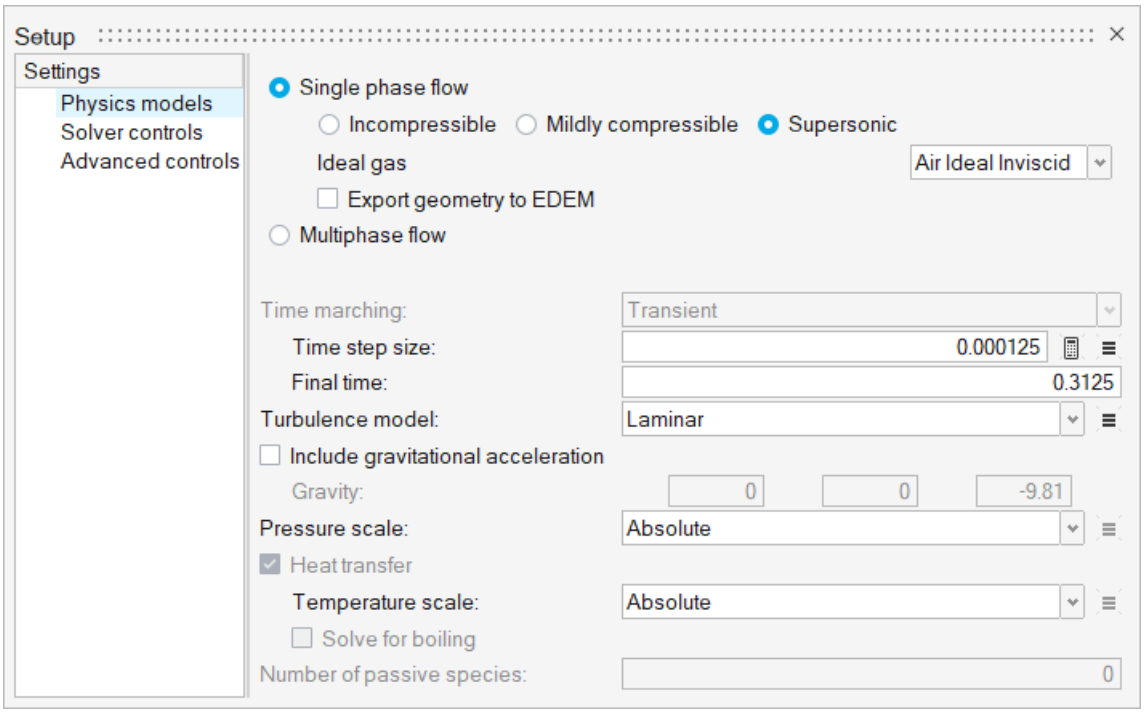


Figure 190:

3. Click the **Solver controls** setting.
4. Set the Transient update factor to `0.5`.

> **Note:** Since we are interested in a solution that reaches a steady state, the transient update factor can be set to a non-zero value without affecting the solution accuracy.

5. Set the Maximum stagger iterations to `5`.

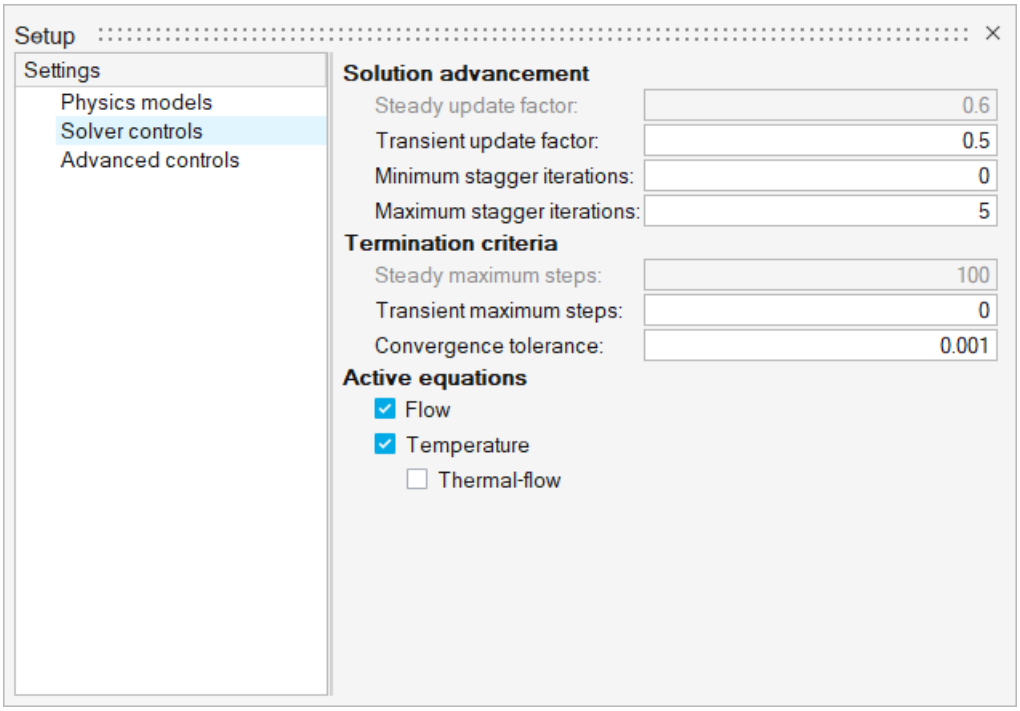


Figure 191:

6. Close the dialog and save the model.

Verify the Material Selection

1. From the Flow ribbon, click the **Material** tool.

Figure 192:

2. Verify that the Air Ideal Inviscid material has been assigned to the volume domain.
3. Click ✓ on the guide bar.

Define Flow Boundary Conditions

1. From the Flow ribbon, Pressure tool group, click the **Stagnation Pressure** tool.

Figure 193:

2. Click on the inlet face highlighted in the figure below.

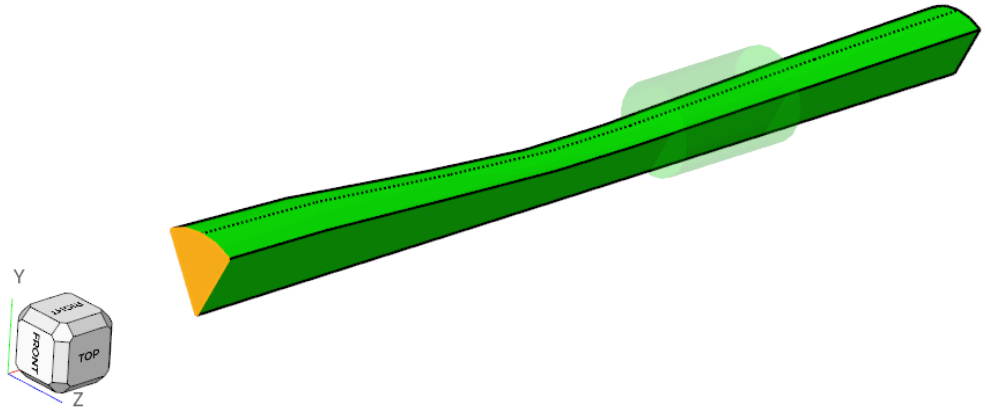


Figure 194:

3. In the microdialog, enter the following values for the stagnation pressure and temperature.

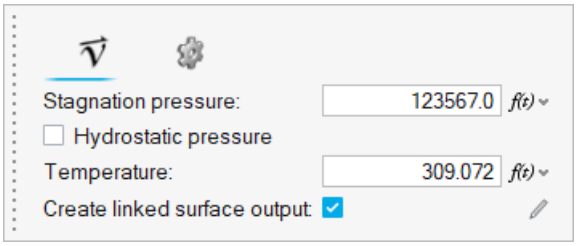


Figure 195:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 196:

6. Select the face highlighted in the figure below, enter the `101325` as the static pressure value, and then click ✓ on the guide bar.

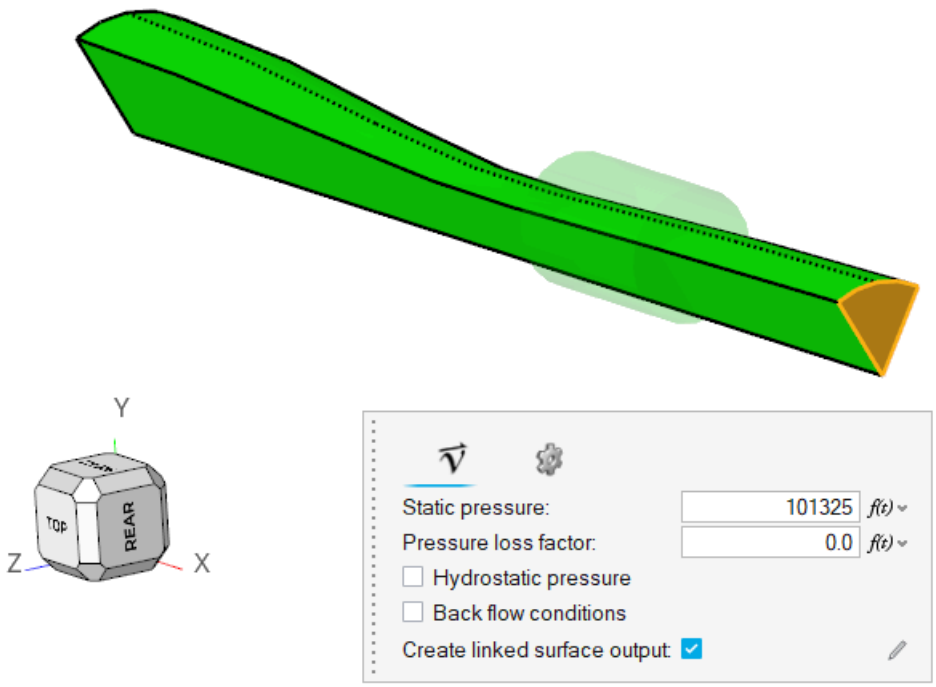


Figure 197:

7. Click the **Slip** tool.

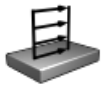


Figure 198:

8. Select the face highlighted in the figure below and then click ✓ on the guide bar.

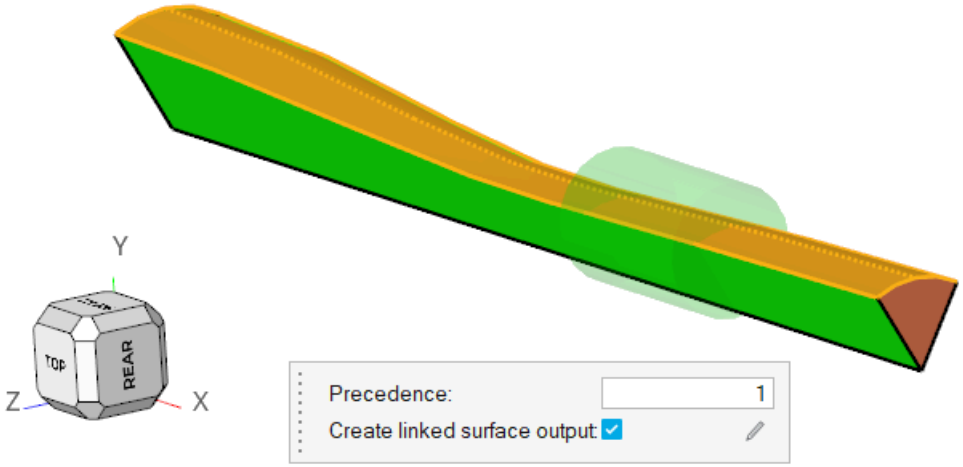


Figure 199:

9. Click the **Periodic** > **Rotation** tool.

Figure 200:

10. Select the face highlighted below as the Source.

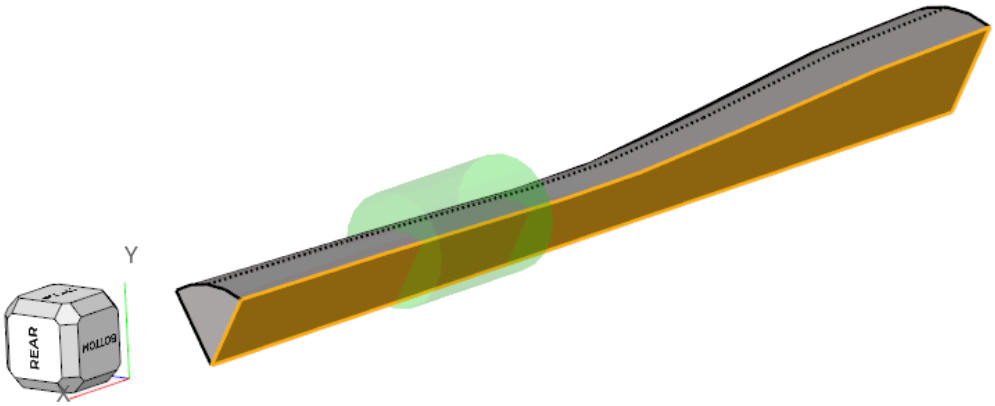


Figure 201:

11. Click **Target** on the guide bar then select the opposite face.

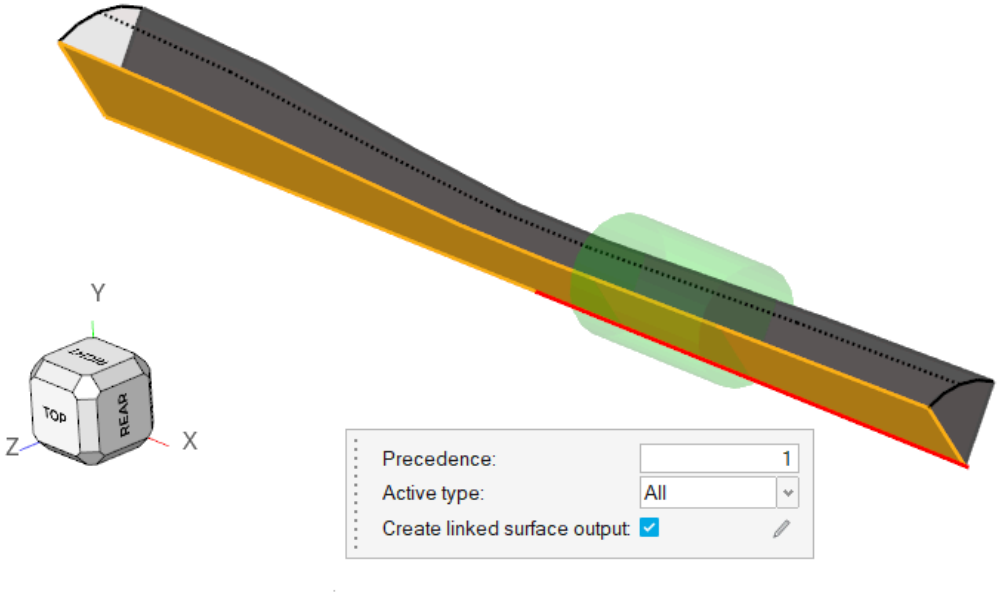


Figure 202:

12. Click ✓ on the guide bar.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

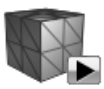


Figure 203:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.1 and the Mesh growth rate is 1.3.

3. Accept all other default parameters.

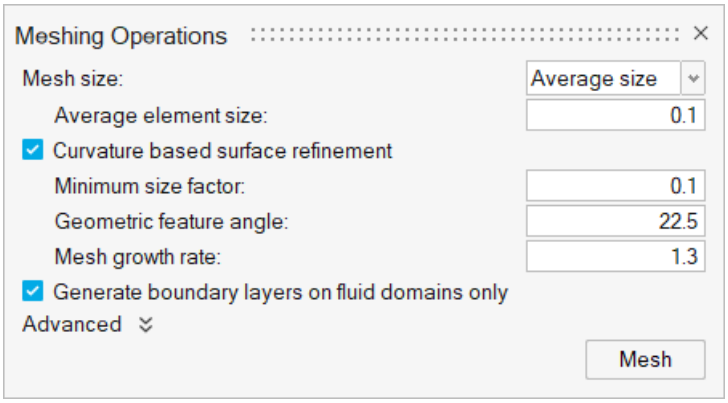


Figure 204:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 205:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial conditions** and enter the values as shown below.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

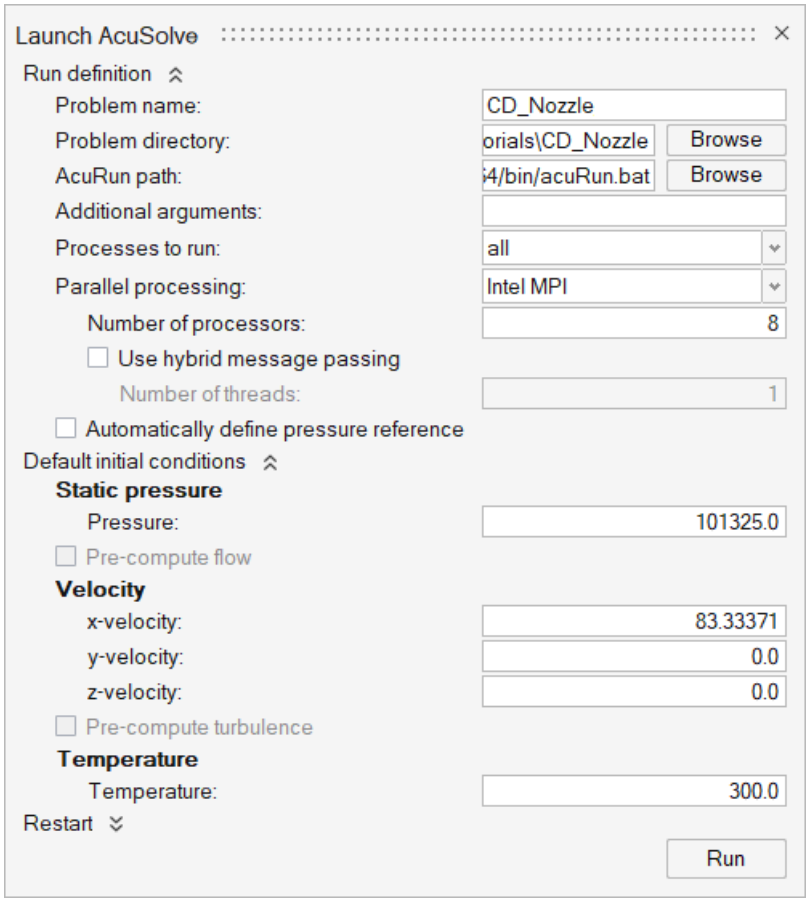


Figure 206:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

In this step, you will check the contours of pressure on a mid slice plane.

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

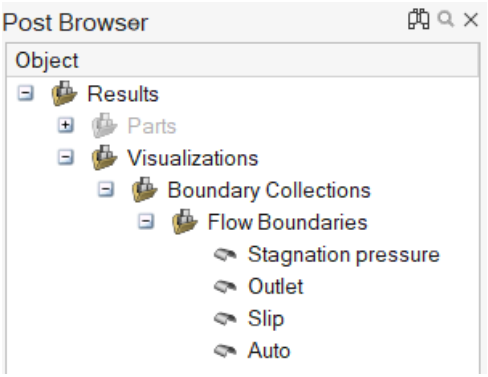


Figure 207:

4. Move the animation slider to the end to load the last time step data.

Figure 208:

5. Click the **Slice Planes** tool.

Figure 209:

6. Select the x-y plane in the modeling window.
7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, set the display to **pressure** and activate the **Legend** toggle.
9. Click and set the Legend location to **Upper Center**, the Legend Orientation to **Horizontal**, and the Colormap name to **Rainbow Uniform**.

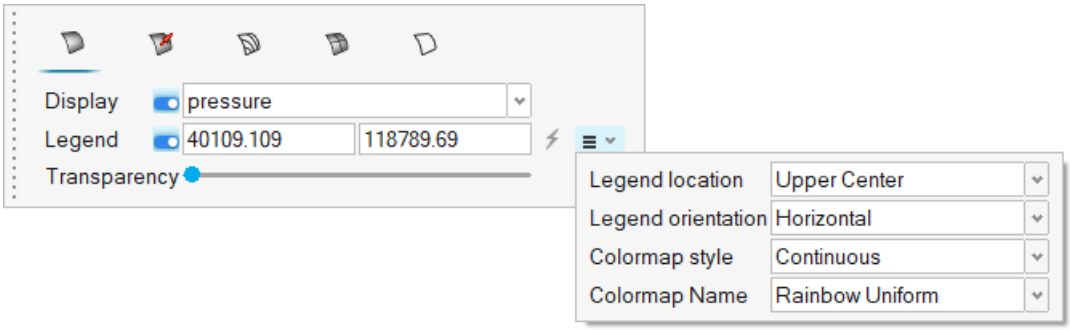


Figure 210:

10. Click ✓ on the guide bar.

11. In the Post Browser, hide the **Flow Boundaries** surfaces to display the pressure contours on the slice plane.

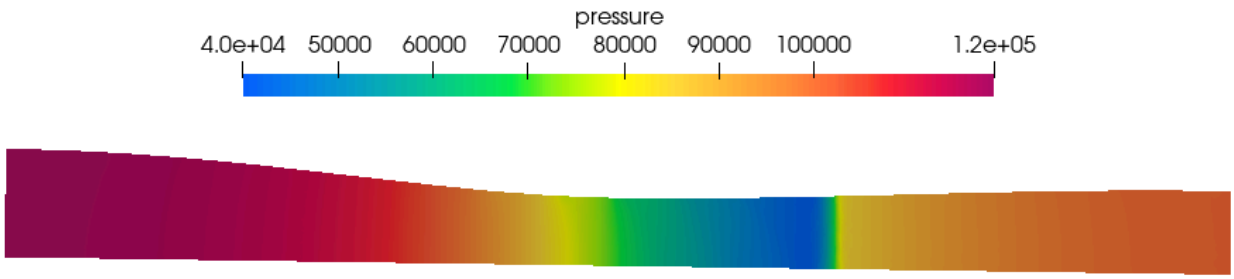


Figure 211:

Similarly, you can display the Mach number contour.

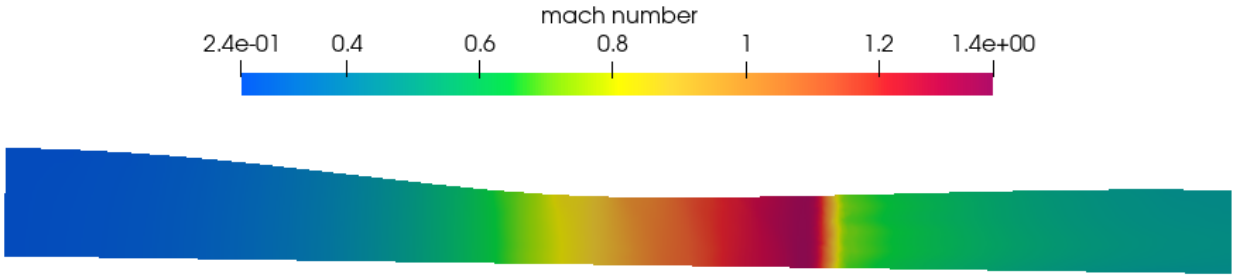


Figure 212:

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/n865kl6pox

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 2500 Floating Surface Monitor

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/empk3dg2zq

Additional Resources

For more tutorials, visit the SimLab Learning Center.

Heat Transfer

3

This chapter covers the following:

- ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection (p. 129)
- ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow (p. 130)
- ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow (p. 149)
- ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping (p. 164)
- ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model (p. 189)
- ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial (p. 213)
- ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model (p. 242)
- ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model (p. 257)
- ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model (p. 272)
- ACU-T: 3300 Modeling of a Heat Exchanger Component (p. 305)
- ACU-T: 3310 Single Phase Nucleate Boiling (p. 321)
- ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model (p. 335)
- ACU-T: 3400 AcuSolve -Flux Integration (p. 352)
- ACU-T:3401 Electric Motor Analysis using AcuSolve - Flux One-Way Coupling (p. 353)
- ACU-T: 3500 Electric Potential - Automotive Fuse (p. 354)
- ACU-T: 3510 Battery Thermo – Electric: 2S2P Module (p. 355)
- ACU-T: 3511 Battery Thermo-Electric Pouch Cell (p. 356)
- ACU-T: 3512 Battery Thermal Runaway (p. 357)
- ACU-T: 3513 ARC-Based Thermal Runaway: 1S1P Module (p. 358)
- ACU-T: 3600 Melting of Diesel Exhaust Additive within an Enclosed Tank (p. 359)

ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/mli527qi2s

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3100_MixingElbowHeatTransfer.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 213. It consists of a mixing elbow made of stainless steel with water entering through two inlets with different velocities and at different temperatures. The geometry is symmetric about the XY midplane of the pipe, as shown in the figure.

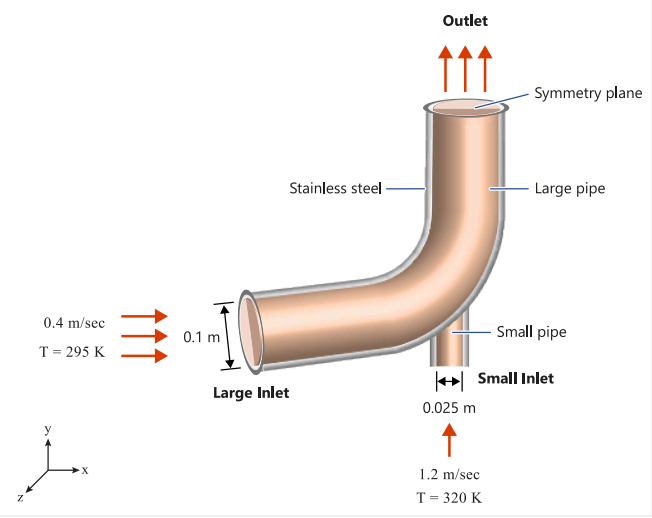


Figure 213: Schematic of Mixing Elbow with Stainless-steel Walls

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 214:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3100_MixingElbowHeatTransfer.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `MixingElbow_HeatTransfer` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `MixingElbow_HeatTransfer` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 215:

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

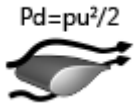


Figure 216:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.
      The robustness and accuracy of the Spalart Allmaras turbulence model makes it an excellent choice for simulation of steady state flows.
   c) Activate the **Heat transfer** checkbox.

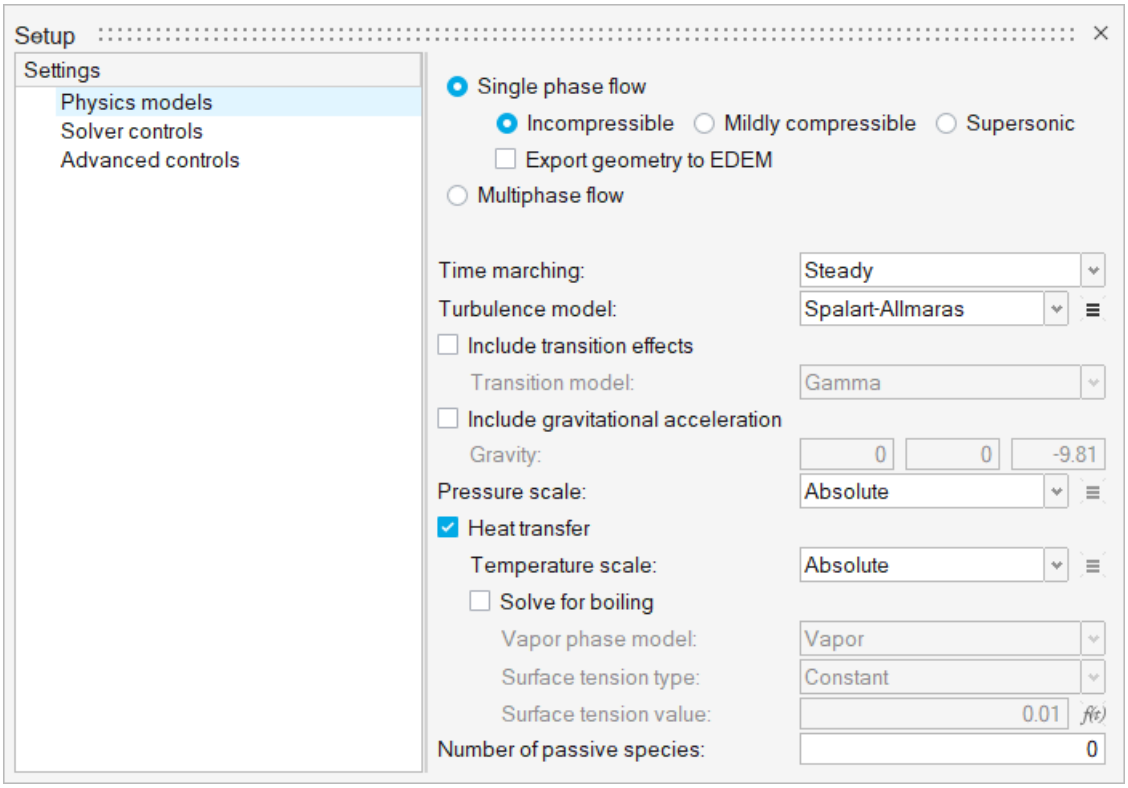


Figure 217:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

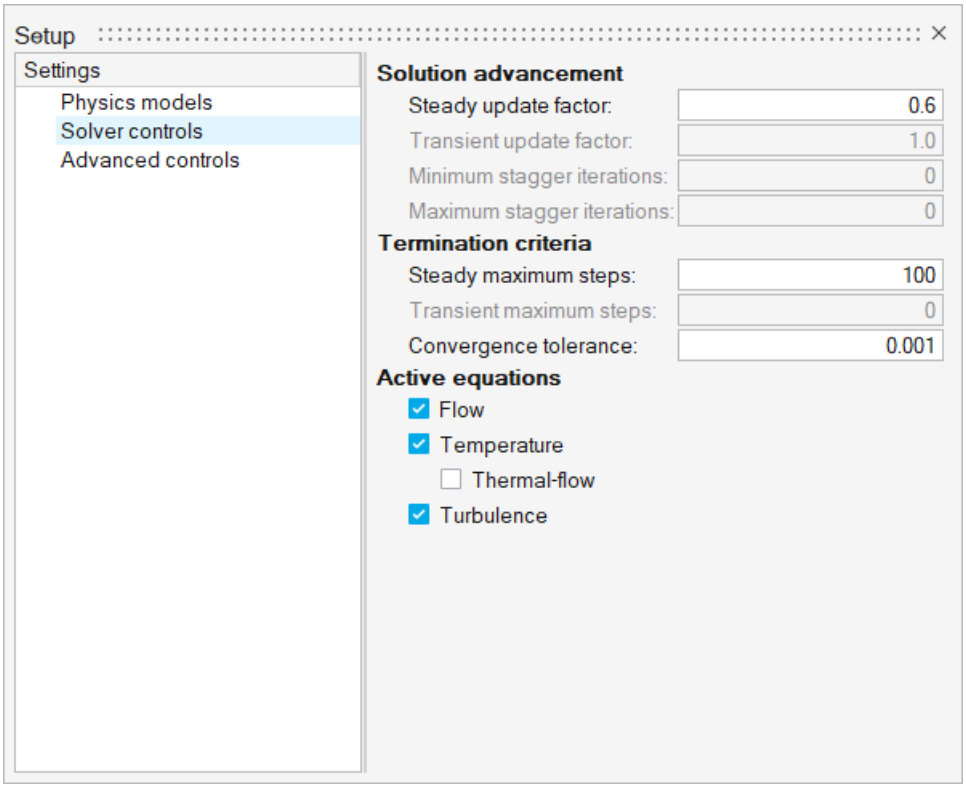


Figure 218:

4. Close the dialog and save the model.

Create a New Material Model

1. From the Flow ribbon, click the **Material Library** tool.

Figure 219:

The **Material Library** opens.

2. Under Settings, click **Solid**.
3. Click the **My Materials** tab then click + to create a new solid.
A material creation dialog opens.
4. Rename the material.
   a) In the top-left of the dialog, click **Solid** to edit the name.
   b) Type `Steel` and press **Enter**.
5. Change the Density value to `8030`.

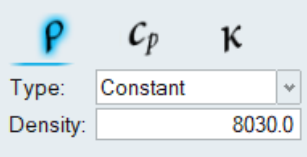


Figure 220:

6. Click the **Specific Heat** tab.
7. Change the Specific heat value to `500`.

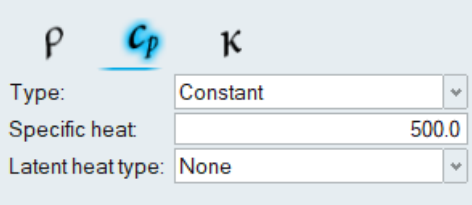


Figure 221:

8. Click the **Conductivity** tab.
9. Change the Conductivity value to `16.2`.

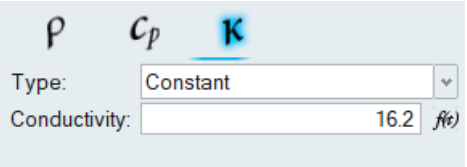


Figure 222:

10. Exit the material creation dialog.

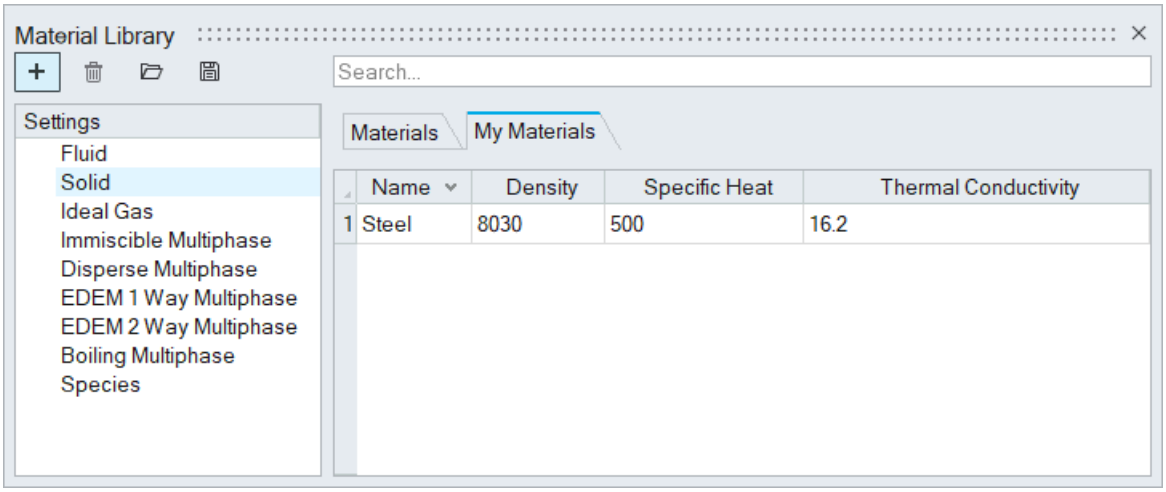


Figure 223:

11. Exit the **Material Library**.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 224:

2. Select the outer pipe solid.

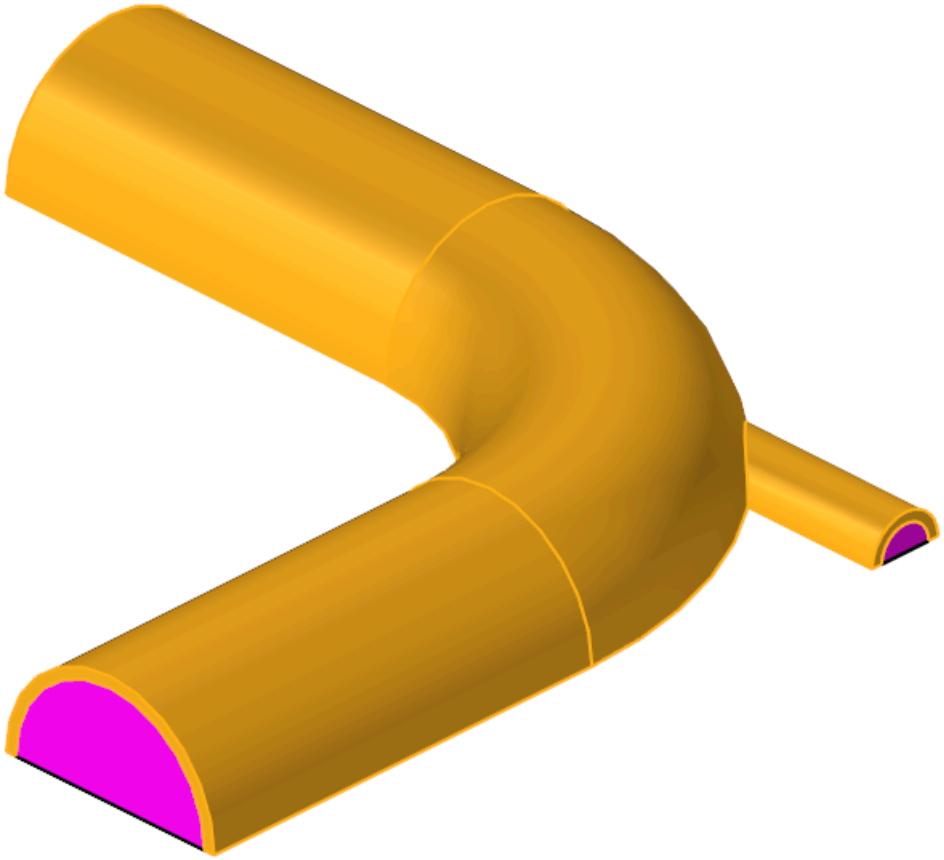


Figure 225:

3. In the microdialog, click the drop-down menu next to Material and select **Steel**.
4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Next, select the inner pipe solid.

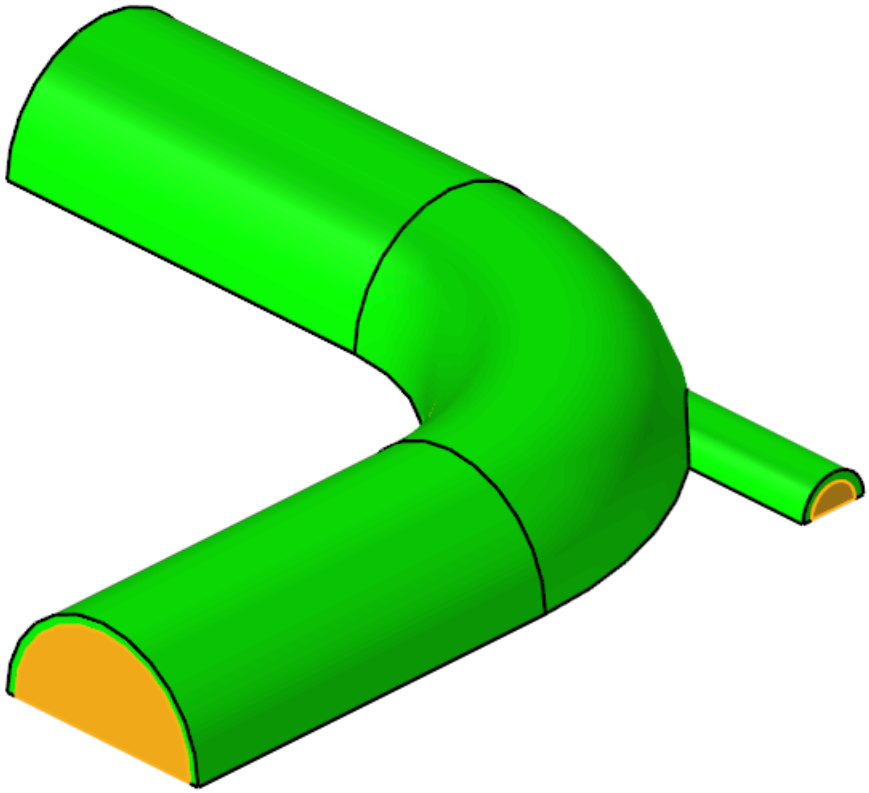


Figure 226:

6. In the microdialog, click the drop-down menu next to Material and select **Water**.
7. On the guide bar, click ✓ to execute the command and exit the tool.

Assign Flow and Thermal Boundary Conditions

Set Boundary Conditions for the Large Inlet

The large inlet of the mixing elbow will be defined as a profiled inlet based on average velocity.

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 227:

2. Click the face of the large inlet.

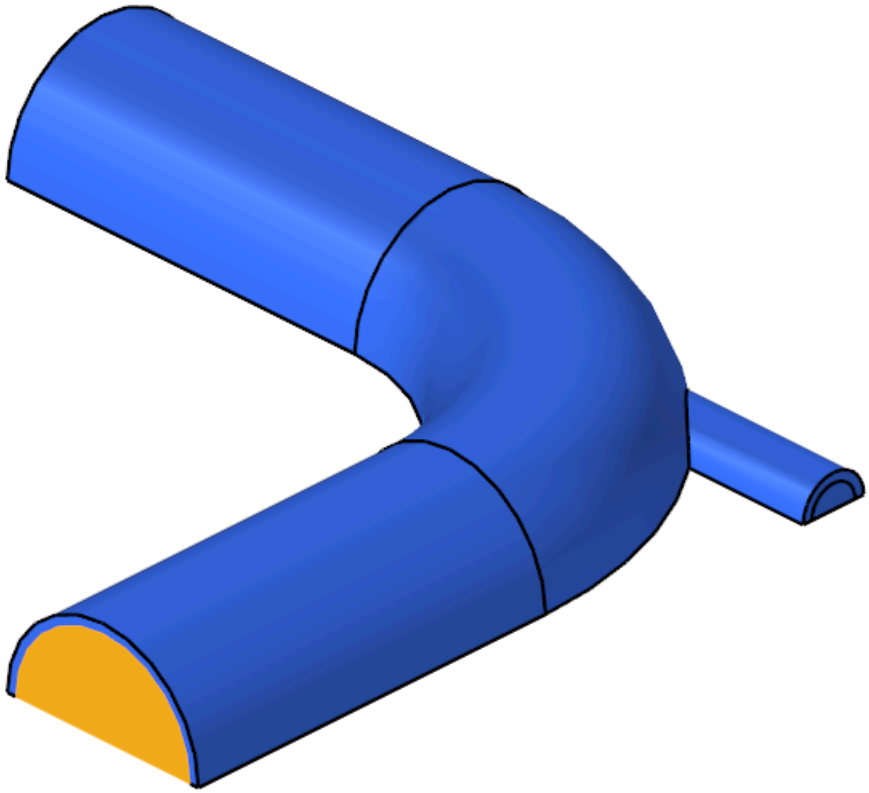


Figure 228:

3. In the microdialog, enter a value of `0.4` for Average velocity and `295` for Temperature.
4. Rename the inlet.
   a) From the legend on the left side of the modeling window, double-click on **Inlet**
   b) Type `Large_Inlet` and press **Enter**.
5. On the guide bar, click ▸ to execute the command and remain in the tool.

> **Note:** The number of inlets created appears in parenthesis on the top-right of the Profiled tool icon.

Set Boundary Conditions for the Small Inlet

The small inlet of the mixing elbow will also be defined using average velocity. The guide bar for the Profiled tool should still be open from the last step.

1. Click the face of the small inlet.

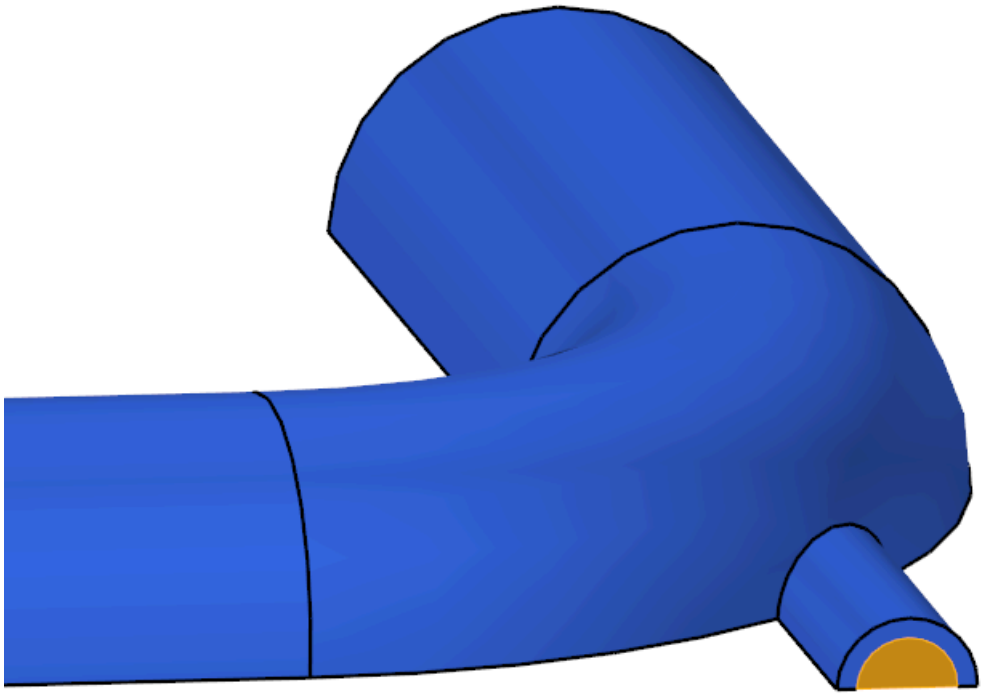


Figure 229:

2. In the microdialog, enter a value of `1.2` for Average velocity and `320` for Temperature.
3. Rename the inlet.
   a) From the legend on the left side of the modeling window, double-click on **Inlet**
   b) Type `Small_Inlet` and press **Enter**.
4. On the guide bar, click ✓ to execute the command and exit the tool.

Set Boundary Conditions for the Outlet

1. From the Flow ribbon, click the **Outlet** tool.

Figure 230:

2. Click the face of the outlet.

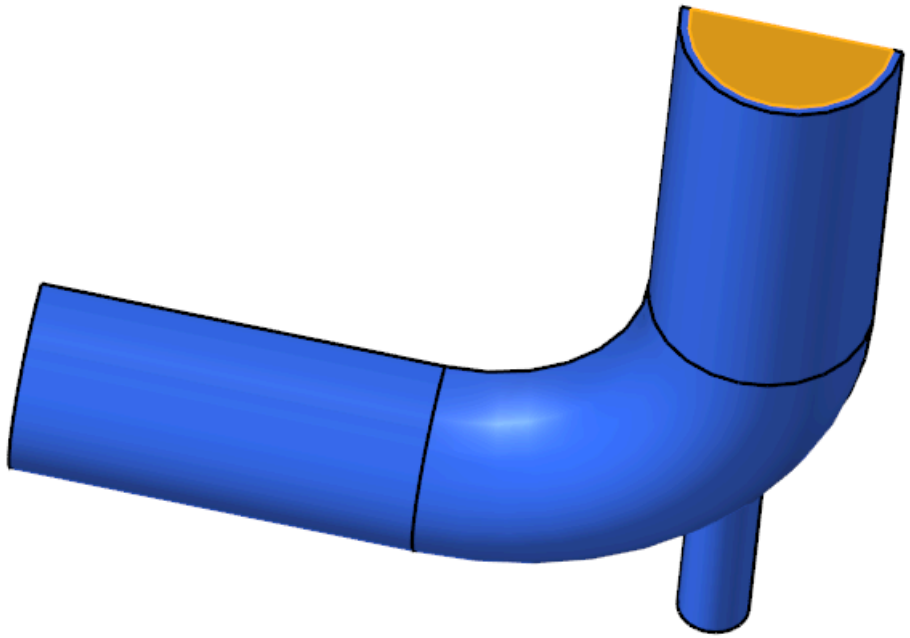


Figure 231:

3. In the microdialog, make sure both Static pressure and Pressure loss factor are 0.

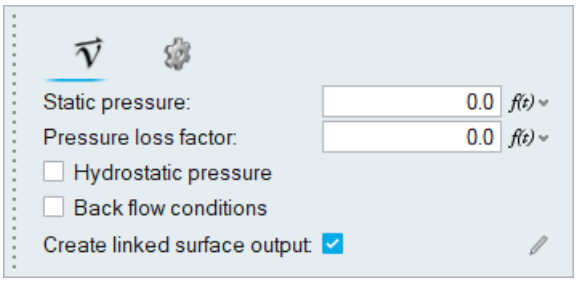


Figure 232:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Set Boundary Conditions for the Symmetry Planes

This geometry is symmetric about the XY midplane, and can therefore be modeled with half of the geometry. In order to take advantage of this, the midplane needs to be identified as a symmetry plane. The symmetry boundary condition enforces constraints such that the flow field from one side of the plane is a mirror image of that on the other side.

1. From the Flow ribbon, click the **Symmetry** tool.

Figure 233:

2. Click the face of the symmetry plane.

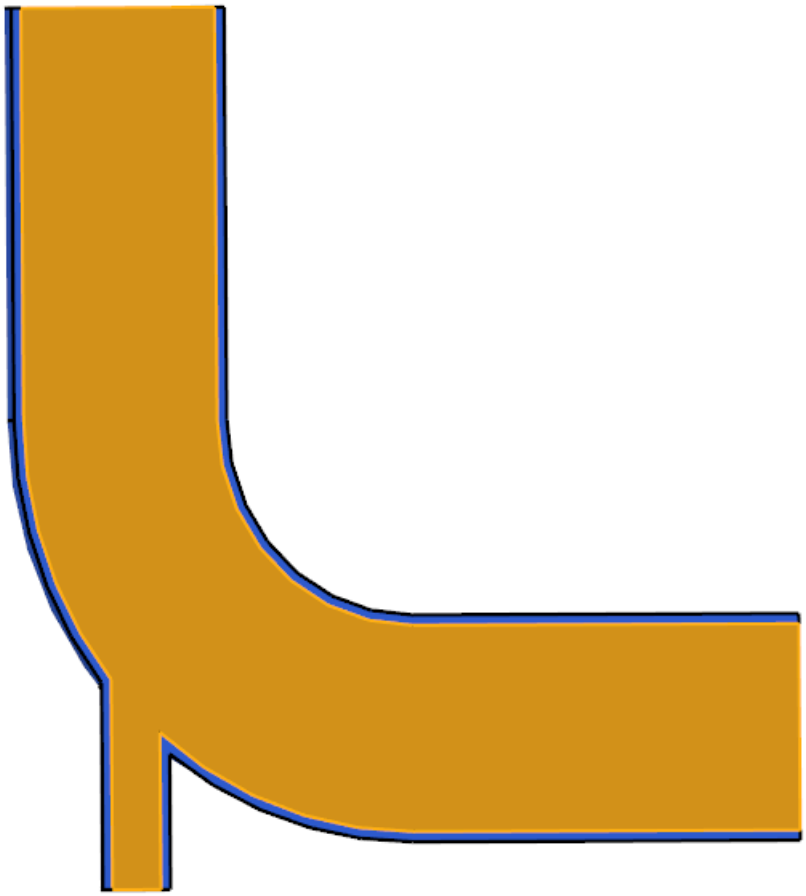


Figure 234:

3. In the microdialog, accept the default symmetry conditions.

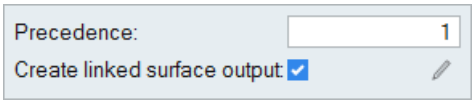


Figure 235:

4. On the guide bar, click ▸ to execute the command and remain in the tool.

5. Next, select the faces for the pipe wall symmetry.

Figure 236:

6. In the microdialog, accept the default symmetry conditions.
7. Rename the boundary.
   a) From the legend on the left side of the modeling window, double-click on **Symmetry 1**
   b) Type `Pipe_Symmetry` and press **Enter**.
8. On the guide bar, click ✓ to execute the command and exit the tool.

Set Boundary Conditions for the Outer Pipe Walls

1. From the Flow ribbon, click the **No Slip** tool.

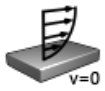


Figure 237:

2. Click the faces of the outer pipe walls.

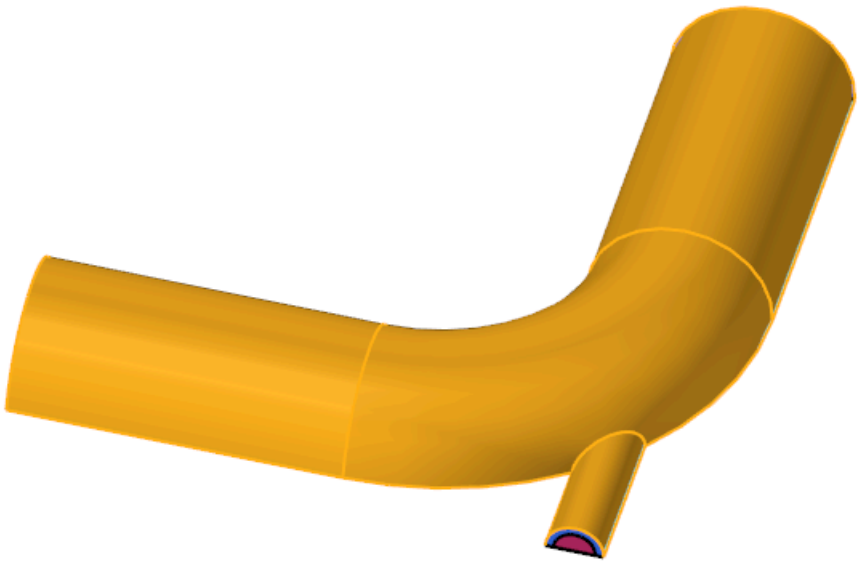


Figure 238:

3. In the microdialog, click to open the temperature tab.
4. Change the Convective heat coefficient value to `100`.
5. Change the Convective heat resistance temperature value to `302.594`.

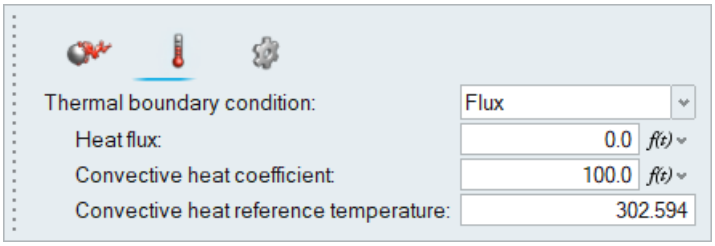


Figure 239:

6. Rename the boundary.
   a) From the legend on the left side of the modeling window, double-click on **Wall**
   b) Type `Pipe_OuterWalls` and press **Enter**.
7. On the guide bar, click ✓ to execute the command and exit the tool.

8. Save the database.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

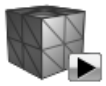


Figure 240:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.03606.
3. Accept all other default parameters.

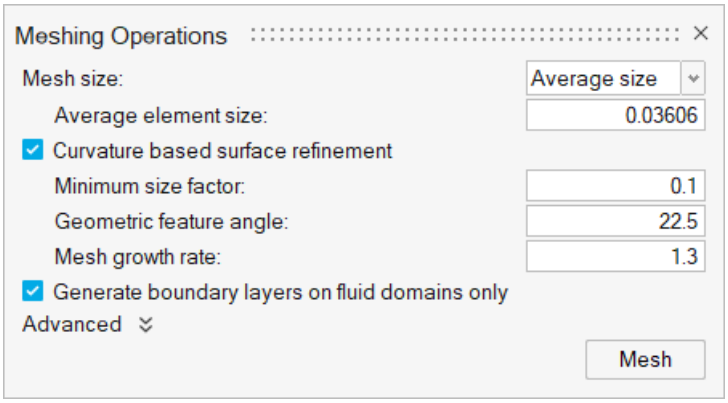


Figure 241:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

5. Observe the refined mesh around the pipe walls.

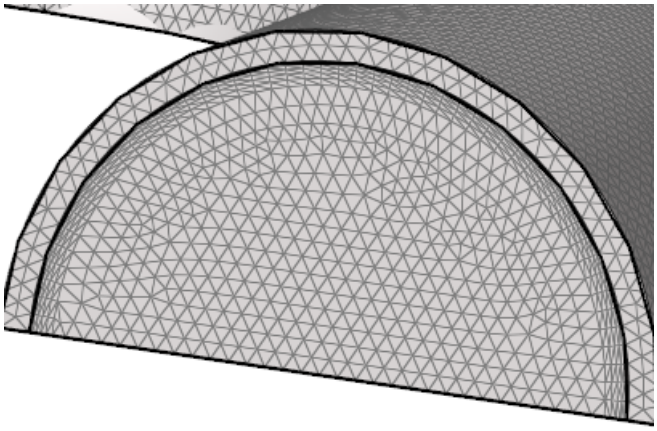


Figure 242:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 243:

   The **Launch AcuSolve** dialog opens.
2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

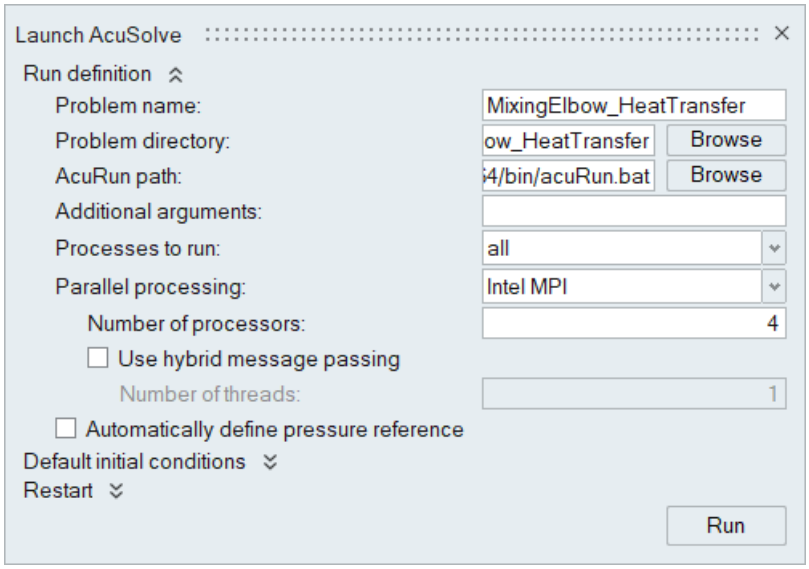


Figure 244:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Isolate the **Symmetry** flow boundary in the Post Browser.
5. Right-click on the **Symmetry** and select **Edit**.
6. In the display properties microdialog, set the display to **temperature** and activate the **Legend** toggle.
7. Click ≡ ˅ and set the Colormap Name to **Rainbow Uniform**.

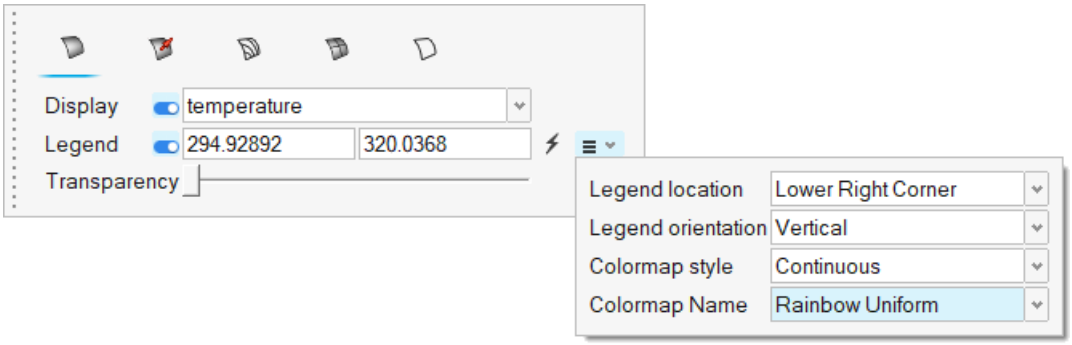


Figure 245:

8. Click ✓ on the guide bar.

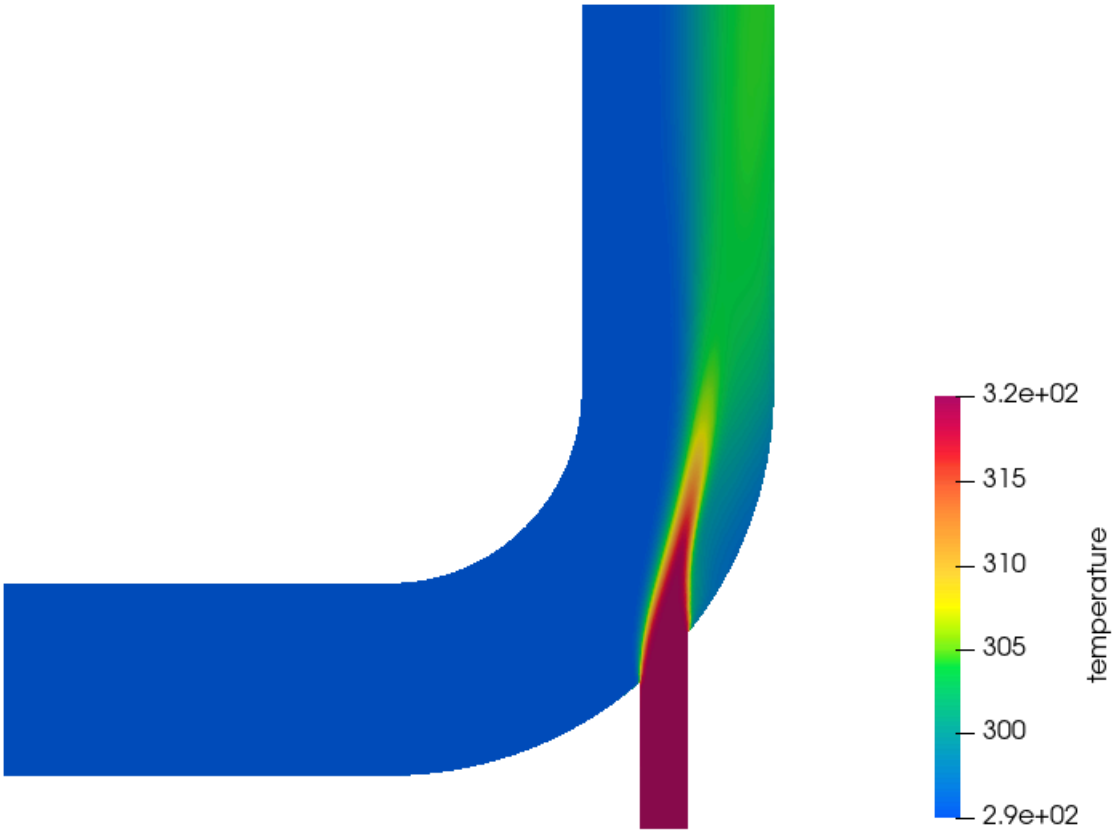


Figure 246:

9. Isolate the **Outlet** flow boundary in the Post Browser.

10. Click the **Right** face on the View Cube to align the model.

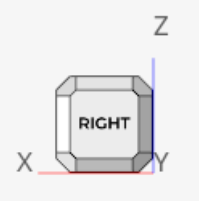


Figure 247:

> **Tip:** If the orientation is not the standard, click the face again to re-align the model back to the standard orientation.

11. Right-click on **Outlet** and select **Edit**.
12. Display the **temperature**.
13. Click ⚡ to reset the range.

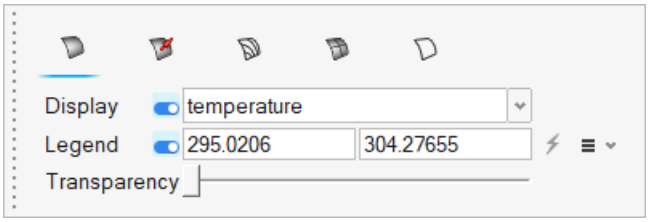


Figure 248:

14. Click ✓ on the guide bar.

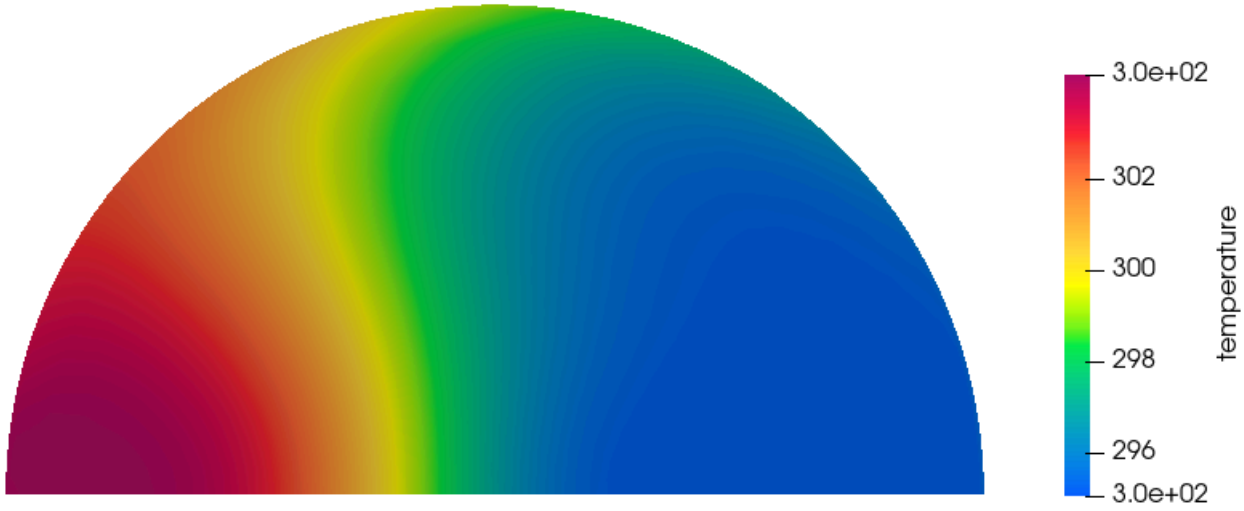


Figure 249:

Summary

In this tutorial, you learned how to set up a conjugate heat transfer simulation using HyperMesh CFD and how to create a new material model. You launched AcuSolve directly from HyperMesh CFD to compute the solution and then post-processed the results using the Post ribbon.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/ltzckmm4e6

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial provides you instructions for running a transient simulation of a 3D turbulent flow with conjugate heat transfer in a mixing elbow using HyperMesh CFD and AcuSolve. You should have already run through the ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow tutorial and have a basic understanding of HyperMesh CFD and AcuSolve. The introductory tutorial, ACU-T: 1000 UI Introduction, gives a basic introduction to AcuSolve and HyperView.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3101_MixingElbowTransient.hm`

Problem Description

This problem is divided into two components, a steady state solution and a transient solution. The schematic of the steady state component is shown below.

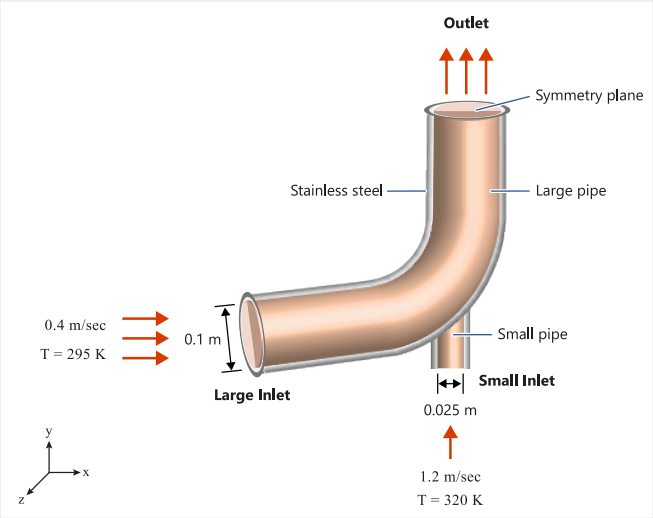


Figure 250:

The diameter of the large inlet is 0.1 m, the inlet velocity (v) is 0.4 m/s and the temperature (T) of the fluid entering the large inlet is 295 K. The diameter of the small inlet is .025 m, the velocity is 1.2 m/s, and the temperature of the fluid entering the small inlet is 320 K. The pipe wall has a thickness of 0.005 m. The fluid in this problem is water and the pipe walls are made of stainless steel with a density of 8030 kg/m3, a conductivity of 16.2 W/m-K, and a specific heat of 500 J/kg-K.

The model file for the steady state part of the problem is provided as the input file. Once the steady state solution is computed, it is projected on to the mesh and used as the initial state for the transient simulation. The starting point for the transient portion of the problem is shown schematically in the figure below.

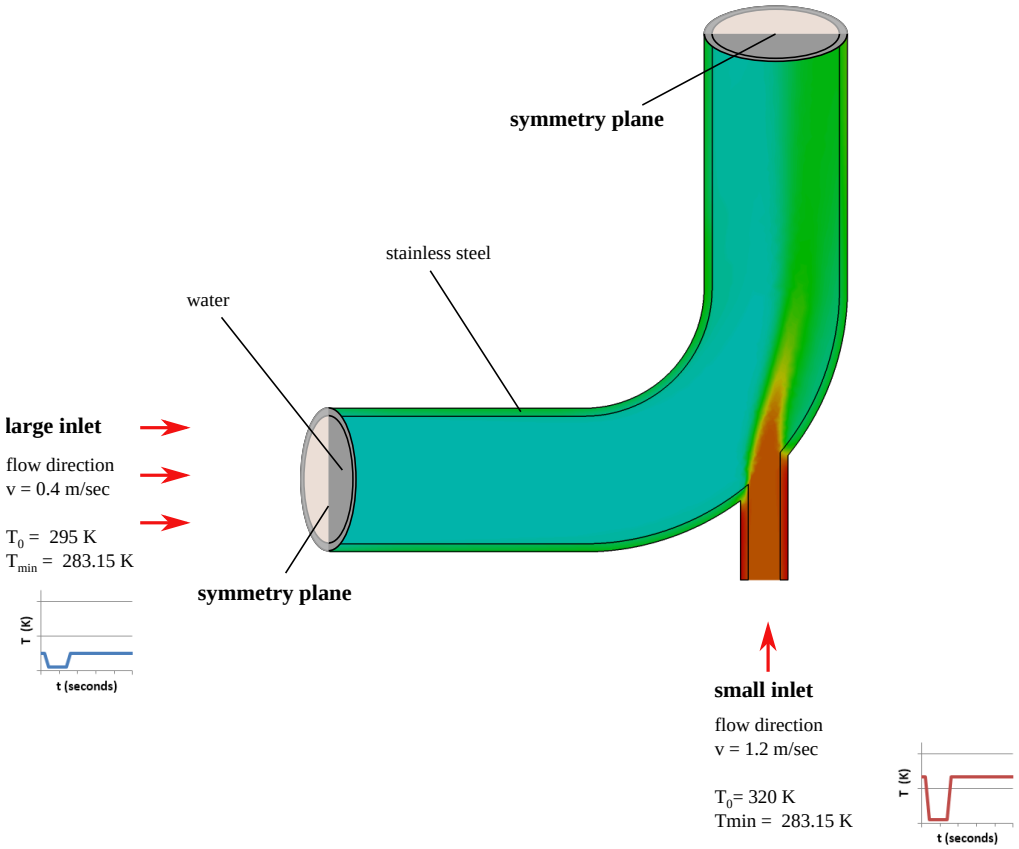


Figure 251:

At 0.2s into the simulation, a cold slug of water is injected at both the inlets and the temperature is ramped down to 283.15 K starting from 0.2 s to 0.4 s. Then it is maintained constant at 283.15 K for 1 sec and then ramped up to initial states from 1.4s to 1.6s. Given a flow path of 0.6356 m, the transit time for the slug is approximately 1.6s. Therefore, our simulation time should be at least 3.2 s to factor in the duration of the slug and transit time. The total simulation time will be 4.5s to allow time for the thermal conditions to return to a steady state.

The temperature change at the large inlet is from 295 K to 283.15 K. At the small inlet, the temperature changes from 320 K to 283.15 K. The ratio of the cold slug temperature to the initial temperature of the large inlet flow is 0.9598. The ratio of the cold slug temperature to the initial temperature of the small inlet flow is 0.8848. These values will be used in creating multiplier functions to model the transient temperatures at the inlets.

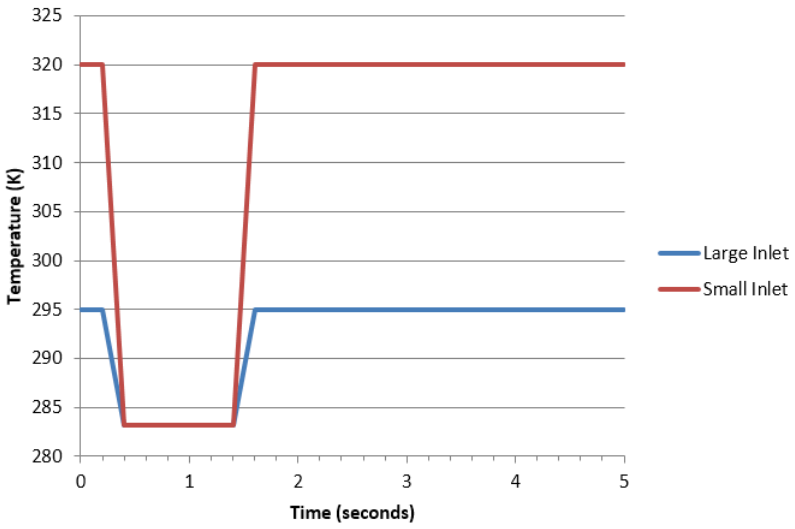


Figure 252:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 253:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3101_MixingElbowTransient.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `MixingElbow_Transient` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `MixingElbow_Transient_Transient` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Run the Steady State Simulation

In this step, you will run the steady state simulation with the input file provided.

1. From the Solution ribbon, click the **Run** tool.

Figure 254:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand the **Default initial conditions** drop-down and deactivate the **Pre-compute flow** option.
6. Set the x-velocity to `0` and the Temperature to `300`.
7. Click **Run** to launch AcuSolve.

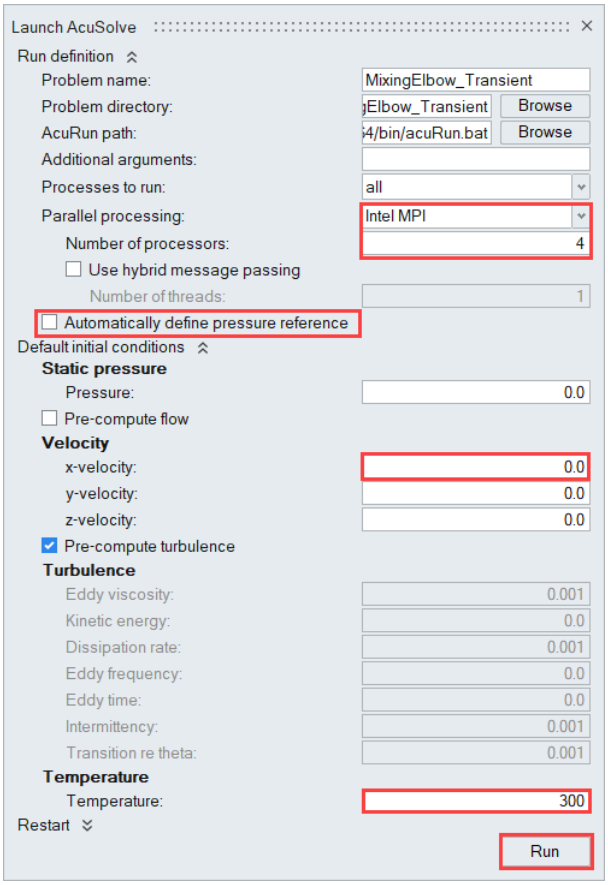


Figure 255:

8. In the **Run Status** dialog, right-click on the AcuSolve run and select **View log file**.
9. Once the solver run is complete, close the **Run Status** dialog.

```
acuSolve: ---------- End Time Step ----------
acuSolve: Input      CPU/Elapse time= 7.609000e+000 1.289000e+001 Sec
acuSolve: Output     CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Exchange   CPU/Elapse time= 6.362690e+002 2.562190e+002 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =        19        19
acuSolve:   Elem.Form CPU/Elapse time= 1.721060e+002 4.984000e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 2.650215e+003 1.078415e+003 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =        19        19
acuSolve:   Elem.Form CPU/Elapse time= 1.078110e+002 3.318800e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 3.132510e+002 1.242760e+002 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =        19        19
acuSolve:   Elem.Form CPU/Elapse time= 1.042040e+002 3.420600e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 9.920600e+002 3.892120e+002 Sec
acuSolve: Pressure proj. Elapse  time= 2.732790e+002 Sec
acuSolve: Linear solver  Elapse  time= 1.298730e+003 Sec
acuSolve: Total CPU/Elapse time      = 4.957765e+003 1.924029e+003 Sec
acuSolve: Total Memory Usage         = 5.002344e+002 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
```

Figure 256:

Set the Transient Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

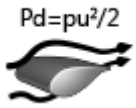


Figure 257:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.053` and the Final time to `4.5`.

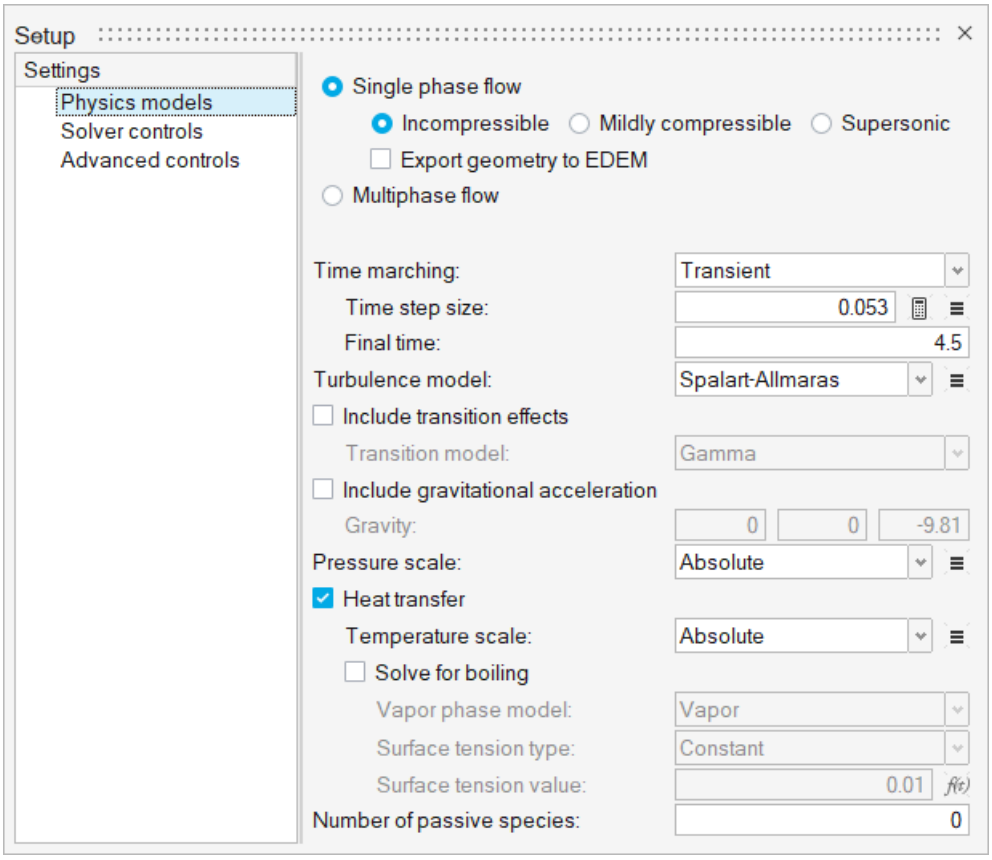


Figure 258:

3. Click the **Solver controls** setting.
4. Set the Minimum and Maximum stagger iterations to `2` and `5`, respectively.
5. De-activate the **Flow** and **Turbulence** equations.

   By turning these options off, AcuSolve will not update the solution to these equations. Instead, the current flow and turbulence values (generated from the steady state solution for this tutorial) will be used throughout the simulation, and AcuSolve will only solve for the temperature field.

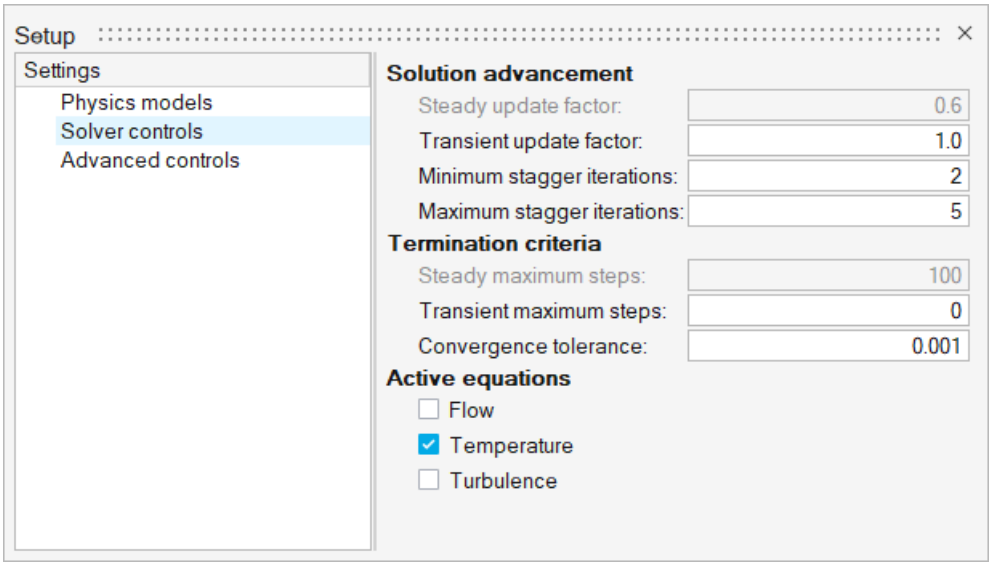


Figure 259:

6. Close the dialog and save the model.

Specify the Transient Inflow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 260:

2. In the modeling window Boundaries legend, right-click **Large Inlet** and select **Edit** from the context menu.

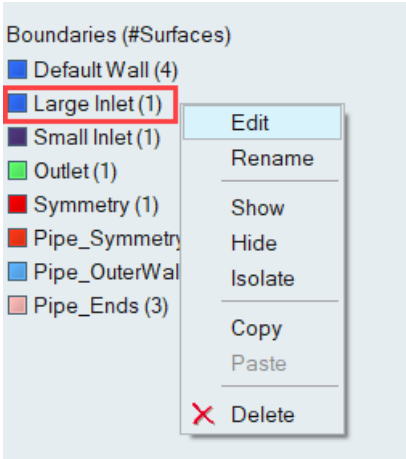


Figure 261:

3. In the microdialog, click multiplier function drop-down menu next to Temperature and select **Create new**.

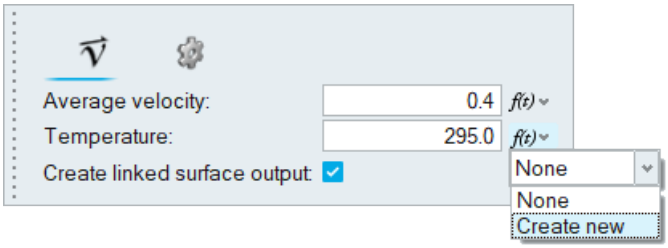


Figure 262:

4. In the dialog that appears, edit the name of the multiplier function by clicking in the top-left corner. Set the name to `Large Inlet`.
5. Set the Type to **Piecewise Linear**.
6. Verify that the Variable is set to **Time** and Evaluation is set to **Time Step**.
7. Click four times to add four new rows to the bottom of the table.
8. Enter the table values for the multiplier function as shown in the image below.

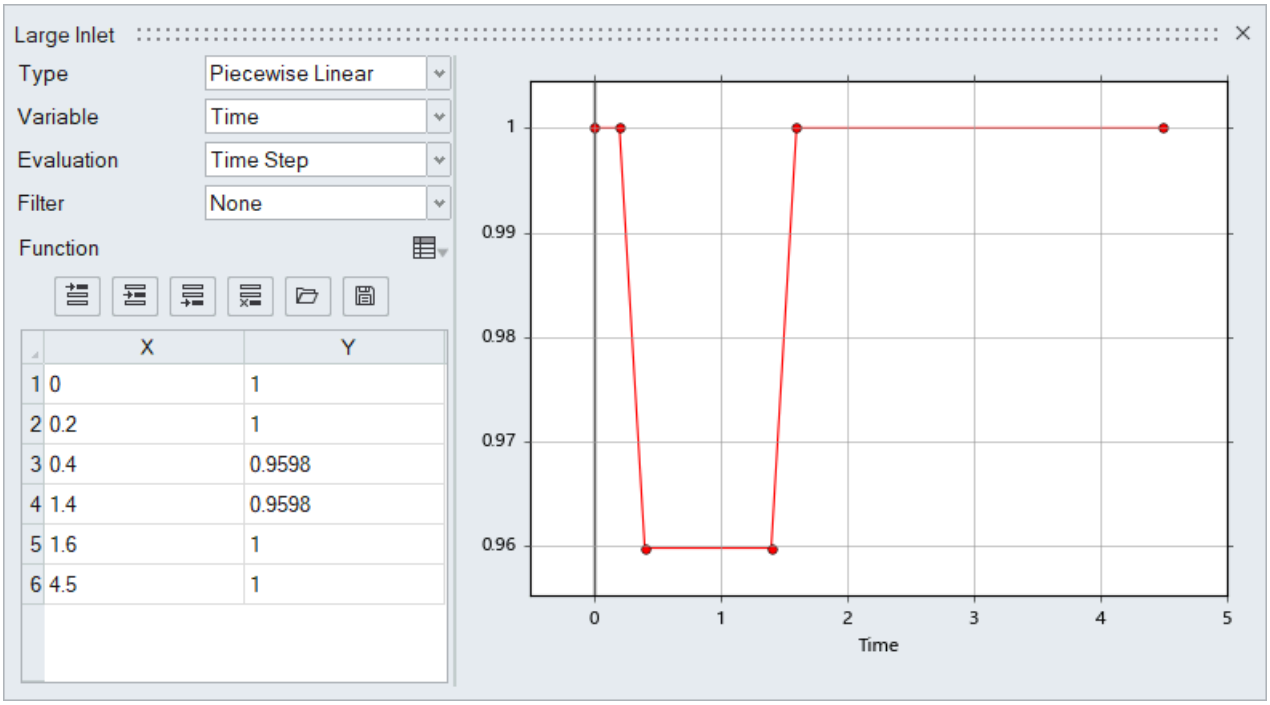


Figure 263:

9. Close the dialog.
10. On the guide bar, click ▸ to execute the command and remain in the tool.
11. In the Boundaries legend, right-click **Small Inlet** and select **Edit** from the context menu.
12. In the microdialog, click multiplier function drop-down menu next to Temperature and select **Create new**.
13. In the dialog that appears, edit the name of the multiplier function by clicking ✎ in the top-left corner. Set the name to `Small Inlet`.
14. Set the Type to **Piecewise Linear**.
15. Verify that the Variable is set to **Time** and Evaluation is set to **Time Step**.
16. Click ⊞ four times to add four new rows to the bottom of the table.
17. Enter the table values for the multiplier function as shown in the image below.

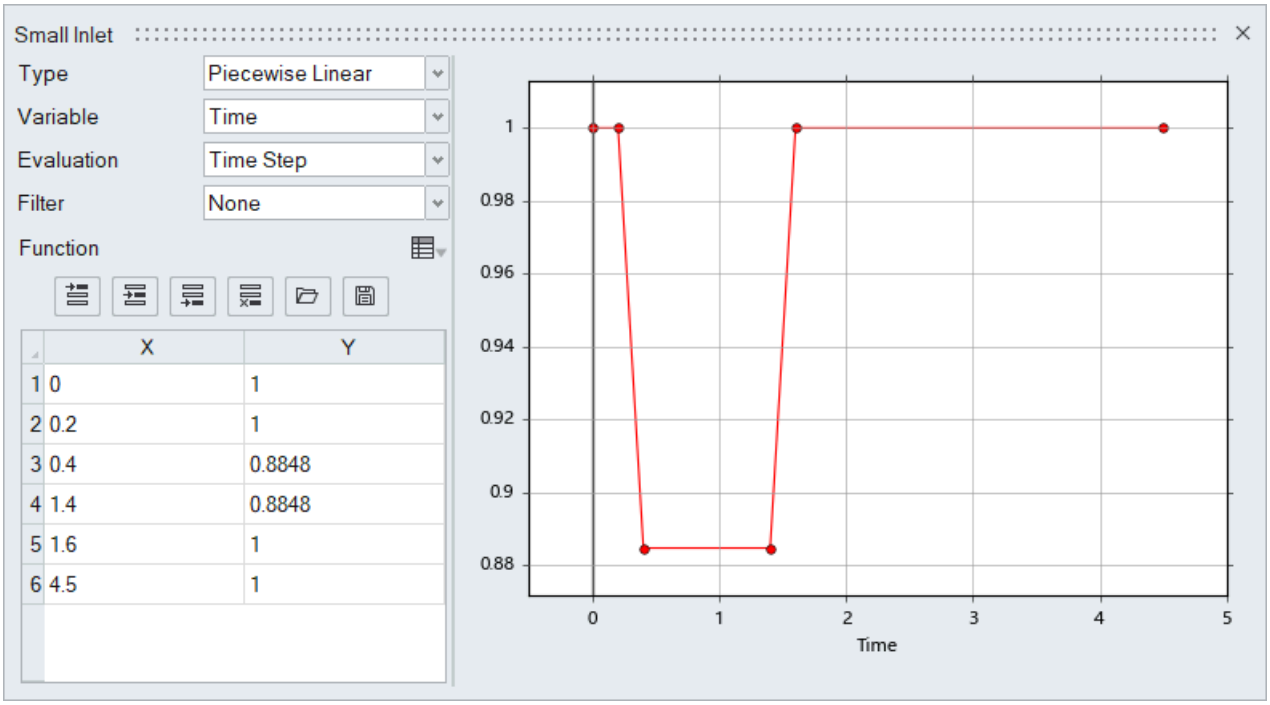


Figure 264:

18. Close the dialog.
19. On the guide bar, click ✓ to execute the command and exit the tool.
20. Save the model.

Compute the Solution

Define Nodal Outputs

1. From the Solution ribbon, click the **Field** tool.

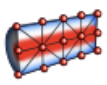


Figure 265:

The **Field Output** dialog opens.

2. Activate the **Write initial conditions** option and set the Time step interval to 3 for the Solution variables.

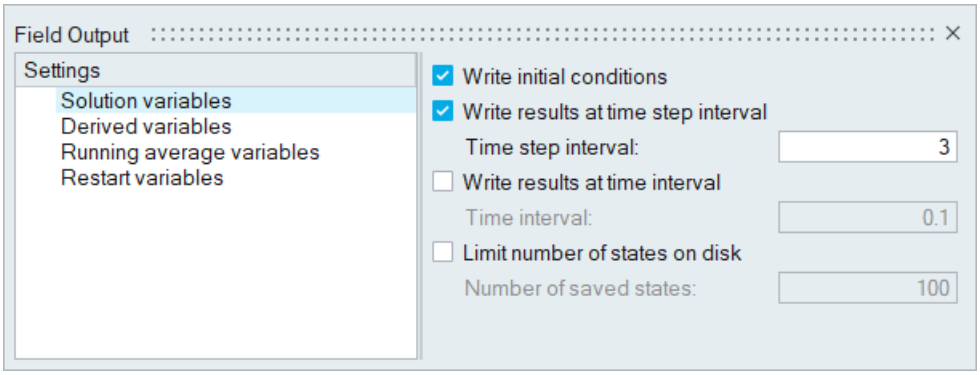


Figure 266:

Launch AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 267:

   The **Launch AcuSolve** dialog opens.
2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand the **Restart** menu and activate the **Restart from previous solution** option.
6. Set the Problem name to **MixingElbow_Transient** (if not set already).
7. Verify that the Run number is set to 1 and the Reset time step option is activated.
8. Click **Run** to start the transient run.

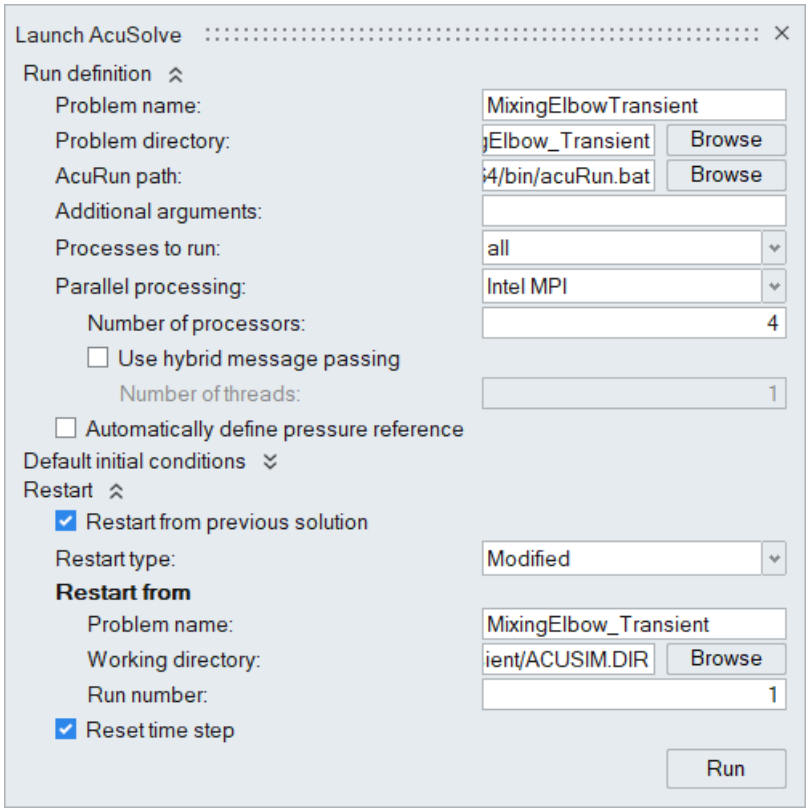


Figure 268:

9. In the **Run Status** dialog, right-click on the second AcuSolve run and select **View log file**.
10. Once the solver run is complete, close the **Run Status** dialog.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Isolate the **Pipe_Symmetry** and **Symmetry** flow boundaries in the Post Browser.

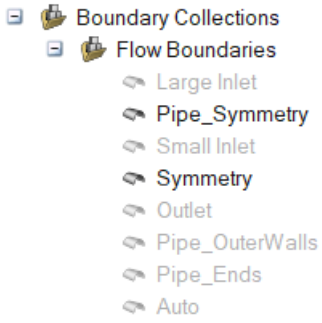


Figure 269:

5. Click the **Boundary Groups** tool.

Figure 270:

6. Select all visible surfaces in the modeling window.
7. In the display properties microdialog, set the display to **temperature**.
8. Activate the **Legend** toggle and click ⚡ to reset the range.
9. Click ≡ ˅ and set the Colormap Name to **Rainbow Uniform**.

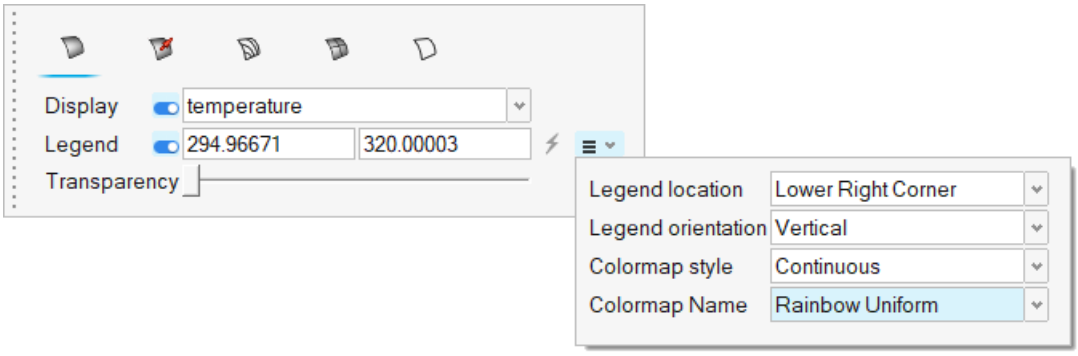


Figure 271:

10. Click ✓ on the guide bar.

11. Click ▶ at the bottom of the modeling window to view an animation of the transient flow with respect to the temperature contour.

Figure 272:

12. Save the animation.

a) Go to **File** > **Screen Capture** > **Advanced Capture**.
b) Click ≡ on the toolbar.
c) Uncheck **Include mouse cursor**.
d) Set the frame rate to `50`.
e) Click ⌖ on the toolbar then drag over the area you want to record.
f) Click ◉ to start recording and the same button to stop recording.
g) Name the file and save it.

Summary

In this tutorial, you learned how to set up and run a transient conjugate heat transfer simulation using HyperMesh CFD and AcuSolve. You started by importing the input file, which had the conjugate heat transfer setup for the steady state run. Once the steady state solution was computed, you set the transient simulation parameters and applied the transient conditions at the inlets. Once the transient solution was computed, you post-processed the results using the Post ribbon.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve.

This tutorial demonstrates how to set up a transient conjugate heat transfer simulation and visualize the results. Before you begin, download the SimLab database file.

You will need an internet connection to watch the video tutorials and download the files.

Tutorial Video

https://fast.wistia.net/embed/iframe/apz6dnwnst

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3110_acuOptiStruct.hm`

Since the HyperMesh CFD database (.hm file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 273. It consists of an exhaust manifold with four inlets and one outlet. The inlets have flanges with holes for steel bolts to attach the manifold. The body of the manifold is made up of stainless steel.

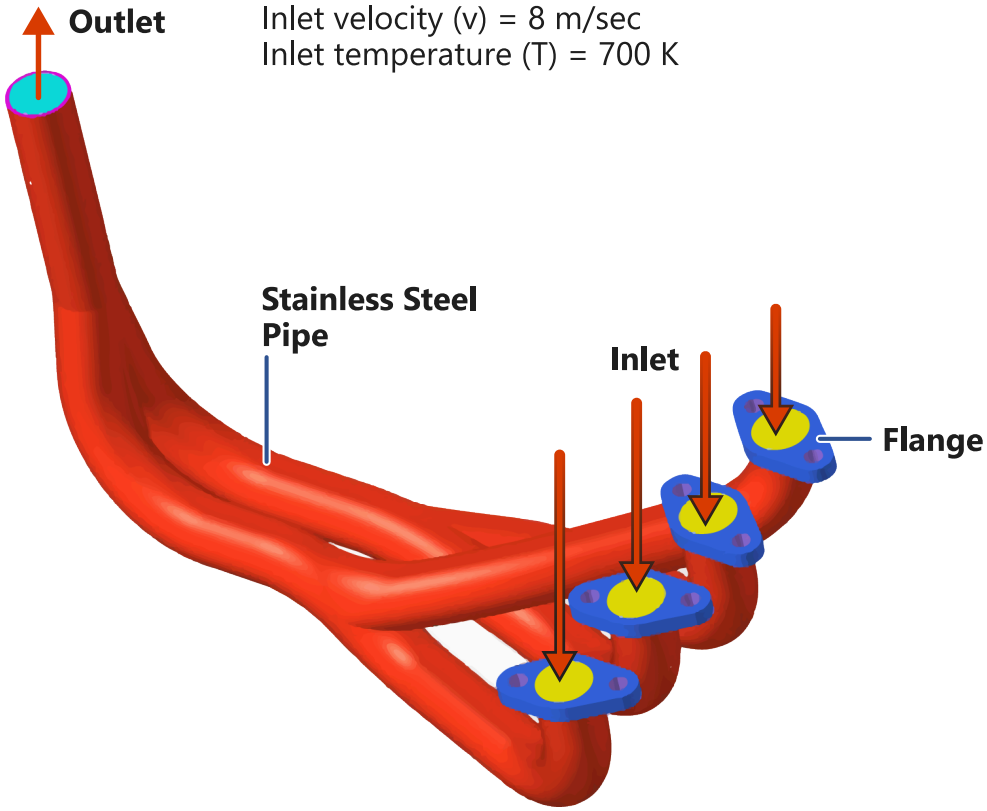


Figure 273: Schematic of Exhaust Manifold

The diameter of the inlets is 0.036 m; the inlet velocity (v) is 8.0 m/s; and the temperature (T) of the fluid entering the inlets is 700 K. The diameter of the outlet is 0.036 m. The pipe wall has a thickness of 0.003 m and the flanges have a thickness of 0.01 m.

The combustion mixture enters the inlets and heat is transferred through conduction inside the manifold. The heat transfer causes deformations and stress in the manifold body which can be simulated using OptiStruct.

The fluid in this problem is air, which has the following material properties:

Density (ρ)	1.225 kg/m^3
Viscosity (μ)	$1.781 * 10^{-5}$ kg/m-s
Specific Heat (C_p)	1005 J/kg-K
Conductivity (k)	0.0251 W/m-K

The exhaust manifold is designed as Steel which has following material properties:

Density (ρ)	8000 kg/m^3
Specific Heat (C_p)	500 J/kg-K
Conductivity (k)	16.2 W/m-K

For the AcuSolve simulation, the variation in material properties of air with temperature is ignored.

The AcuSolve simulation will be set up to model steady state heat transfer to determine the temperature and pressure distribution on the walls of the manifold.

The nodal surface output needs to be activated for all the surfaces in order to create the OptiStruct input deck from the acuOptiStruct command.

The temperature distribution and forces on the wetted surfaces are used by OptiStruct to calculate the deformations and stress in the solid body.

The OptiStruct input deck is generated through a utility acuOptiStruct which can be used for a one-way coupled simulation. The following input commands are of importance for this simulation:

`-solids`	Input name for the solid body/bodies where conduction heat transfer would take place.
`-den`	Density values for the solid body/bodies.
`-spcsurfs`	List of surfaces where boundary condition constraints need to be specified.
`-spcsurfsdof`	List of degrees of freedom for the surfaces.
`-spcsurfsdofvals`	List of degrees of freedom values for the surfaces which is zero by default.
`-type`	Stress analysis type for the OptiStruct solver.

For this simulation, the constrained surfaces are the flange bolts and Outlet end of the manifold. These surfaces will be constrained in all six degrees of freedom (Translation and Rotation). The default value of zero is used.

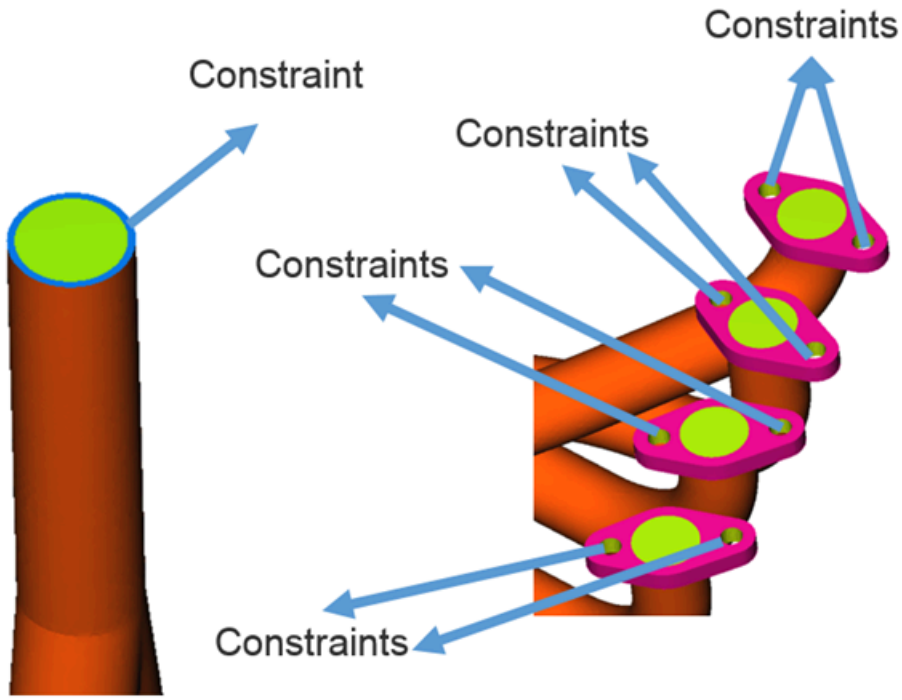


Figure 274:

The stress analysis type is selected as steady linear where the deformations are in the elastic range; that is, the stresses, σ, are assumed to be linear functions of the strains, ε, Hooke's law can be used to calculate the stresses.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 275:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3110_acuOptiStruct.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Manifold_TFSI` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.

6. Enter `Manifold_TFSI` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 276:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

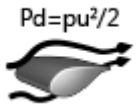


Figure 277:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.
   c) Activate the **Include gravitational acceleration** and **Heat transfer** options.

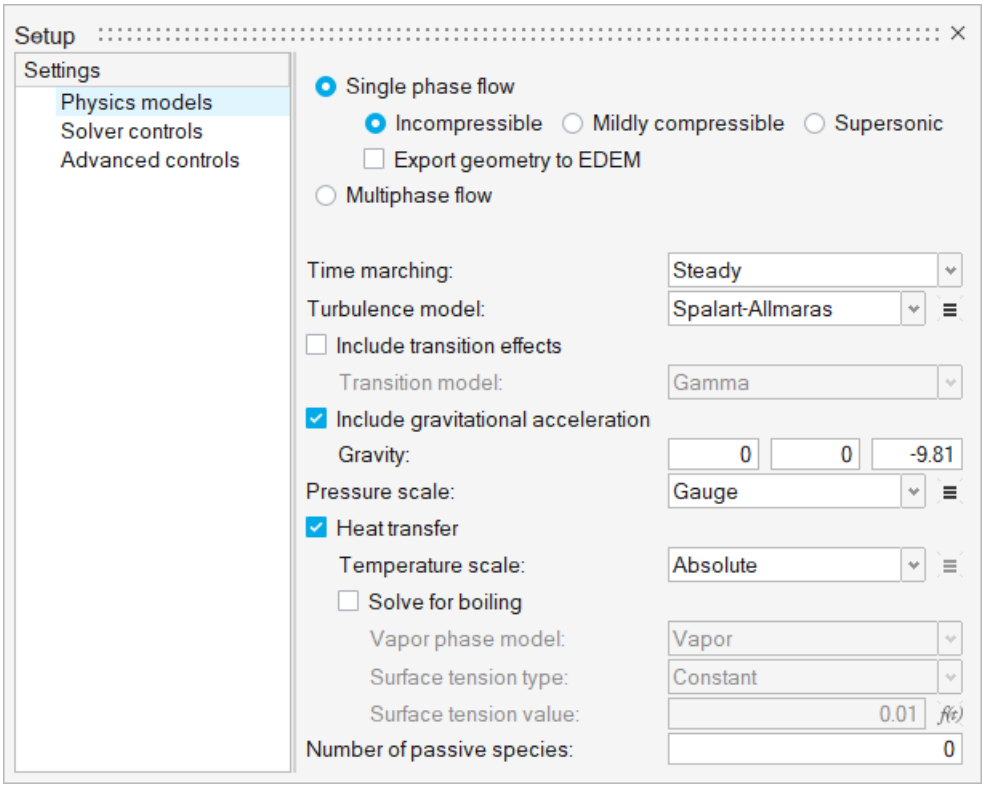


Figure 278:

3. Click the **Solver controls** setting.
4. Verify the following parameters.

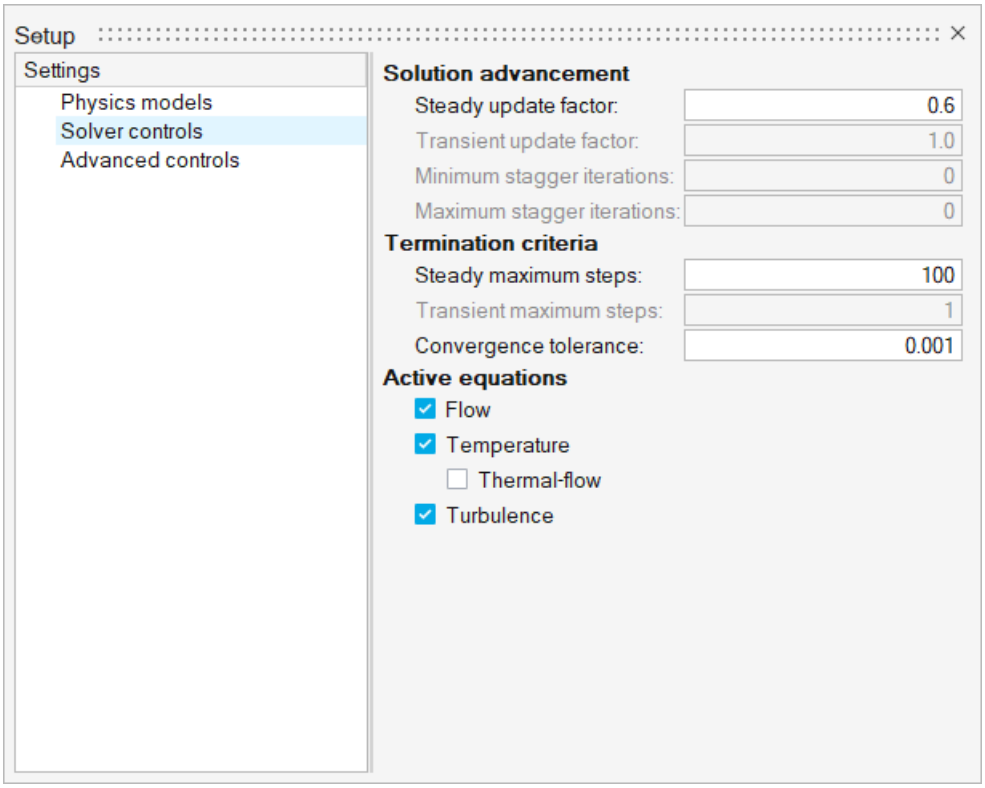


Figure 279:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 280:

2. Select the exhaust volume.
3. In the microdialog, select **Stainless steel (304)** from the drop-down.

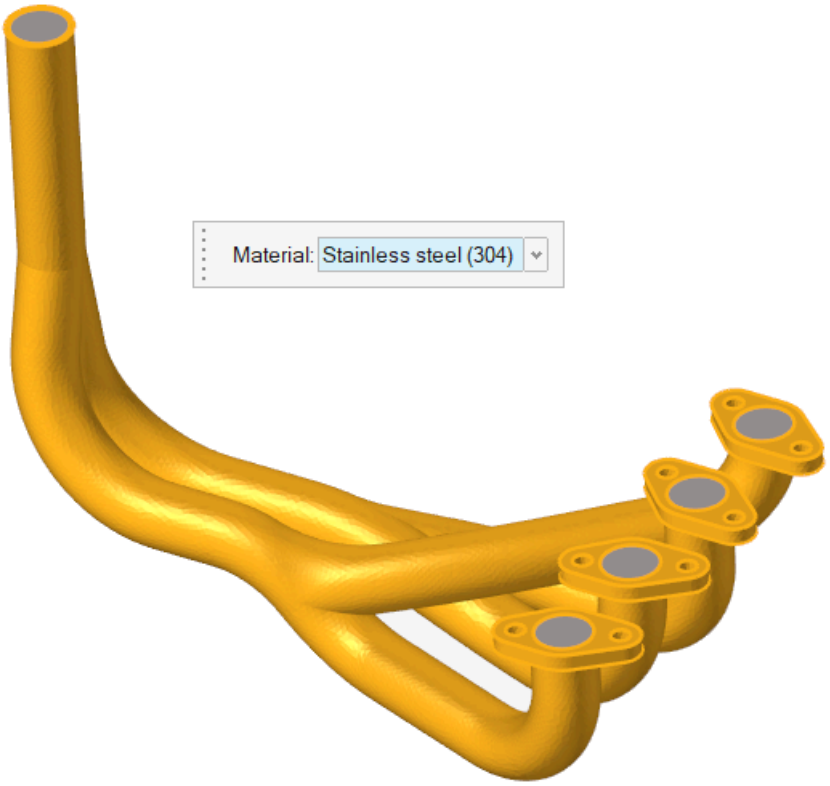


Figure 281:

4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Select the air volume.
6. In the microdialog, select **Air** from the drop-down.

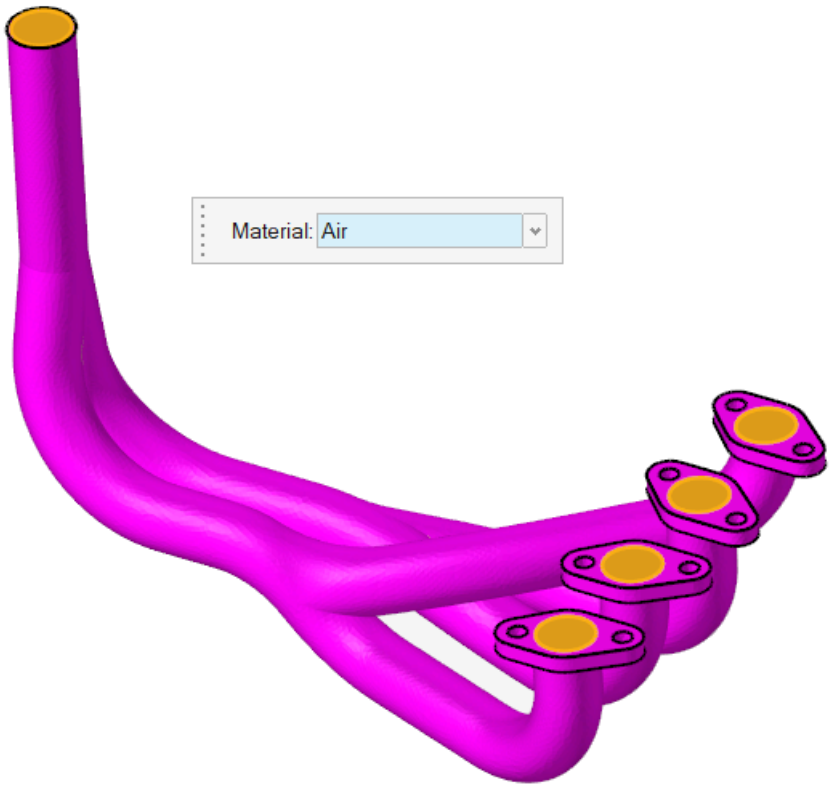


Figure 282:

7. On the guide bar, click ✓ to execute the command and exit the tool.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 283:

2. Select the manifold inlets highlighted in the figure below.

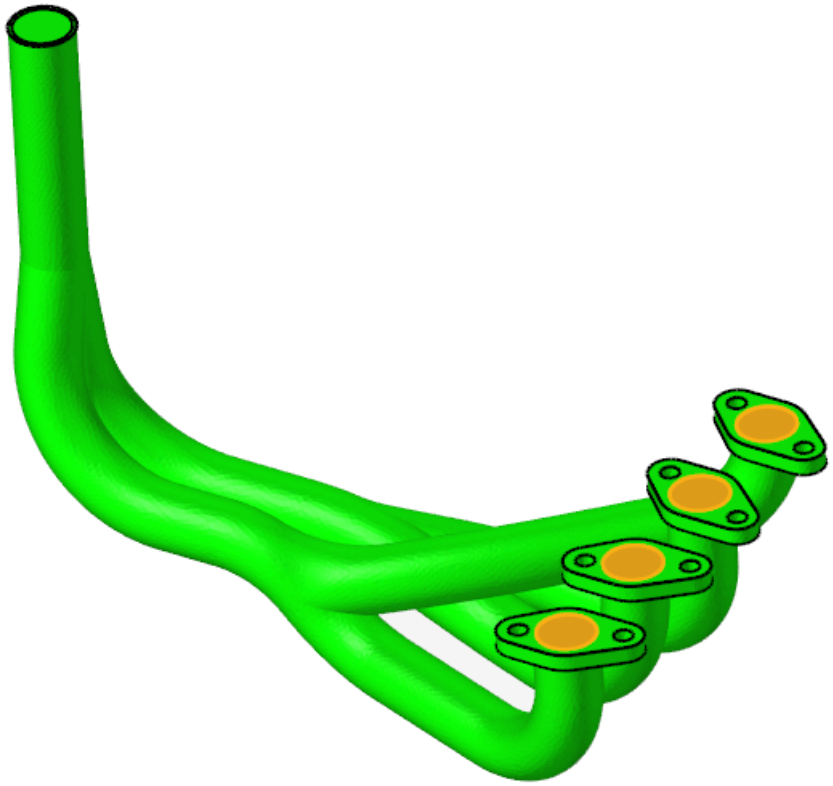


Figure 284:

3. In the microdialog, set the Inflow velocity type to **Normal**, the Normal velocity to `8.0` and the Temperature to `700`.

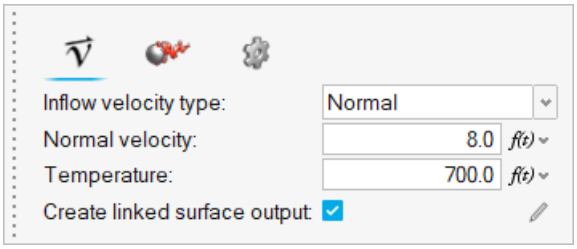


Figure 285:

4. Click the **Turbulence** tab in the microdialog. Set the input type to **Viscosity Ratio** and the viscosity ratio value to `40`.

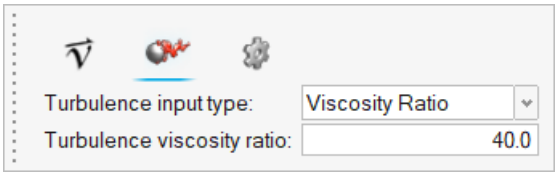


Figure 286:

5. On the guide bar, click ✓ to execute the command and exit the tool.
6. Click the **Outlet** tool.

Figure 287:

7. Select the face highlighted in the figure below and verify that both the static pressure and the pressure loss factor are set to `0`.

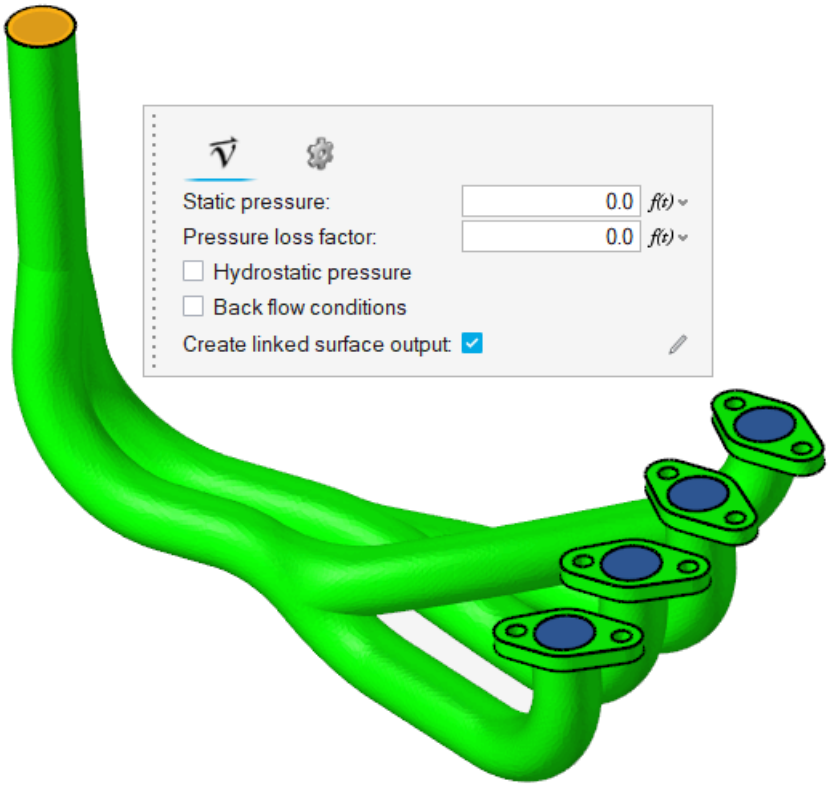


Figure 288:

8. Click ✓ on the guide bar.
9. Click the **No Slip** tool.

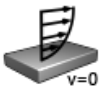


Figure 289:

10. Select the wall flanges.

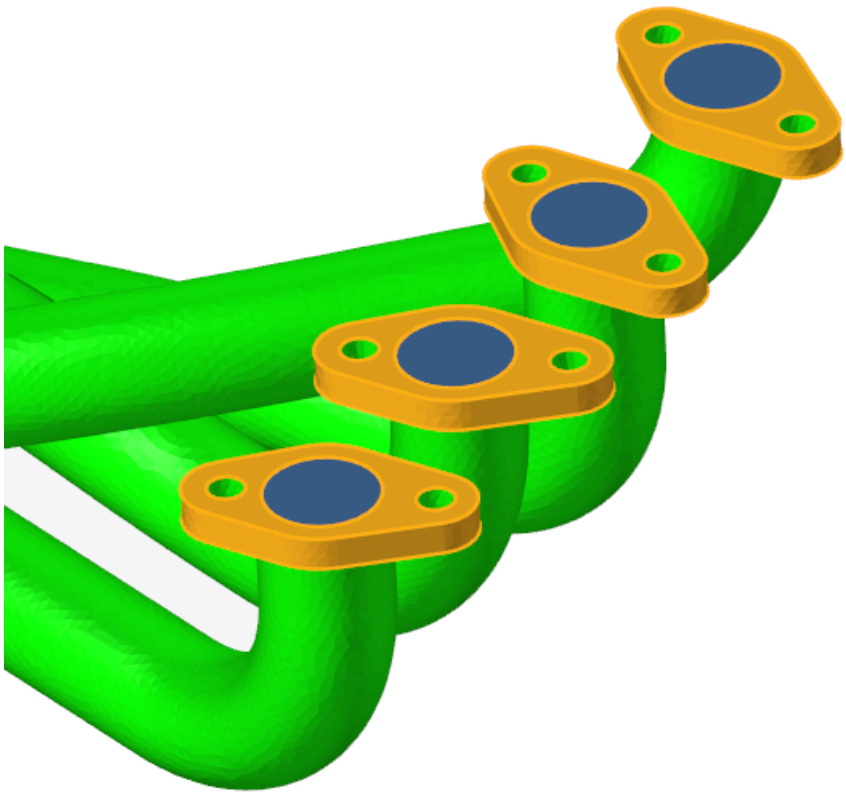


Figure 290:

> **Note:** Make sure you select the bottom and side surfaces of the flanges. In total, 12 surfaces should be selected.

11. In the microdialog, click the **Temperature** tab, set the Convective heat coefficient to `100`, and the Convective heat reference temperature to `303`.

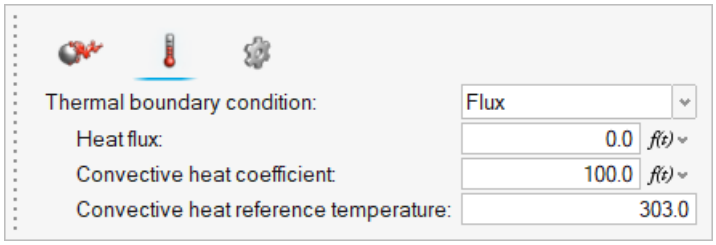


Figure 291:

12. Click ▸ on the guide bar.

13. Using the Boundaries legend, rename the surface group to `Flanges`.

14. Select the flange bolts.

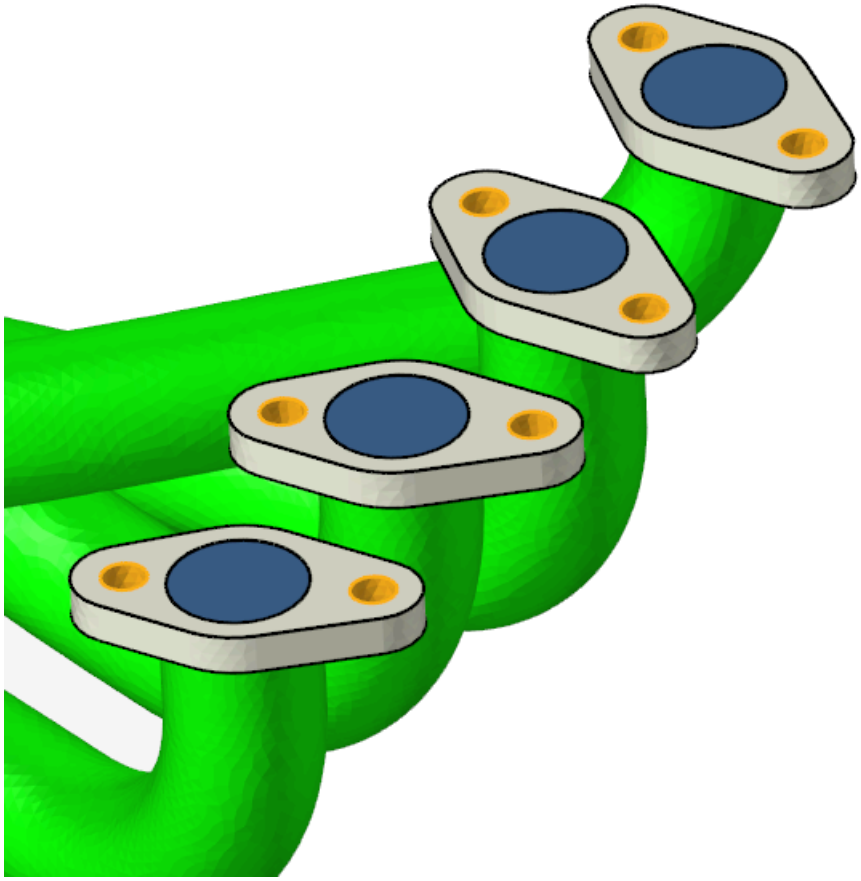


Figure 292:

15. In the microdialog, click the **Temperature** tab and assign the same parameters as the wall flanges.
16. Click ▸ on the guide bar.
17. Using the Boundaries legend, rename the surface group to `Flange_Bolts`.
18. Select the outlet end.

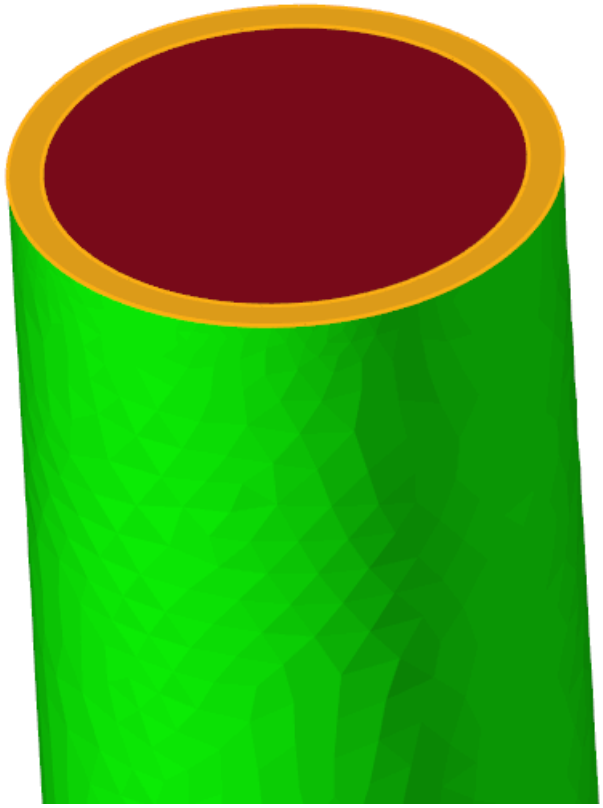


Figure 293:

19. In the microdialog, click the **Temperature** tab and assign the same parameters as the wall flanges and bolts.

20. Click ▸ on the guide bar.

21. Using the Boundaries legend, rename the surface group to `Outlet_End`.

22. Select the outer solid walls.

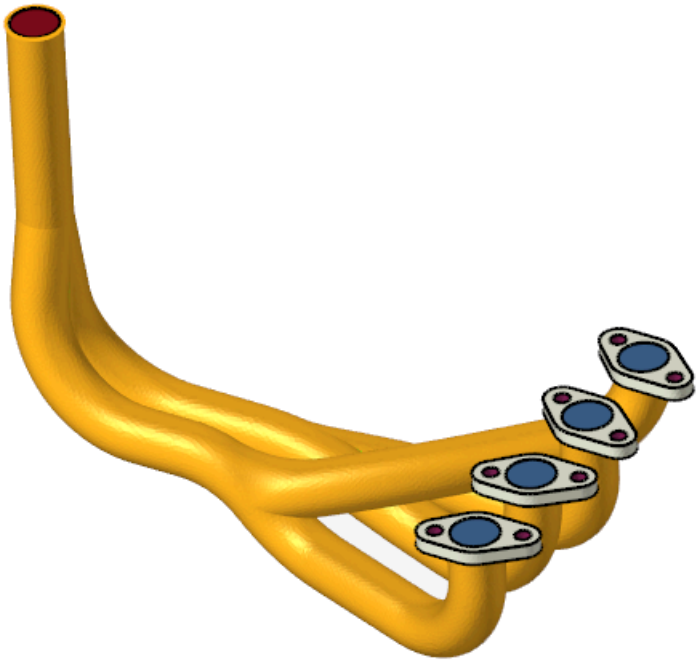


Figure 294:

23. In the microdialog, click the **Temperature** tab and assign the same parameters as the other no slip surfaces.

24. Click ▸ on the guide bar.

25. Using the Boundaries legend, rename the surface group to `Outer_Solid_Walls`.

26. Hide all previously assigned surfaces then select the remaining surface.

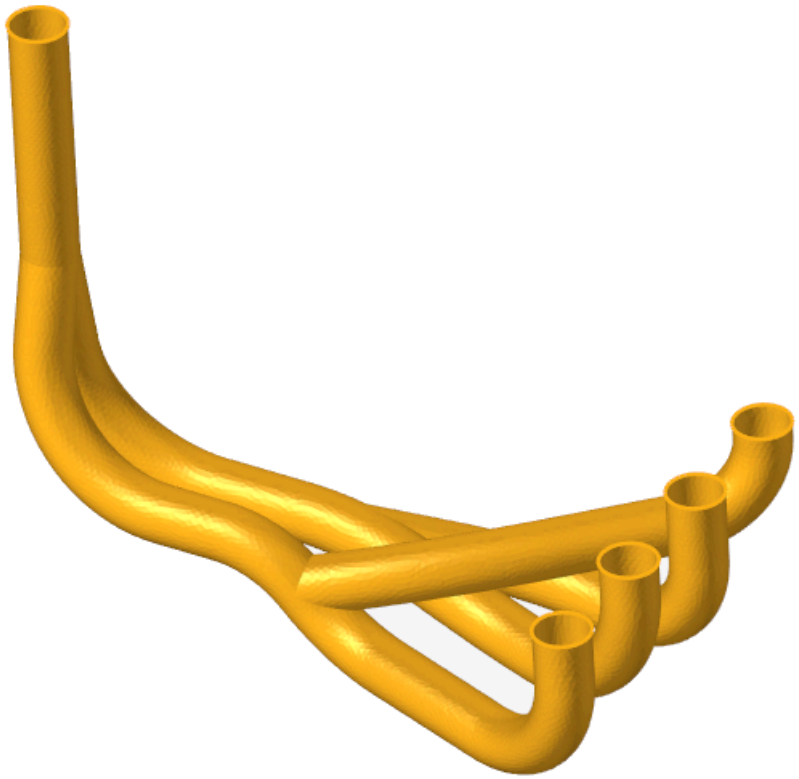


Figure 295:

27. Check that default values are assigned to the thermal boundary conditions.

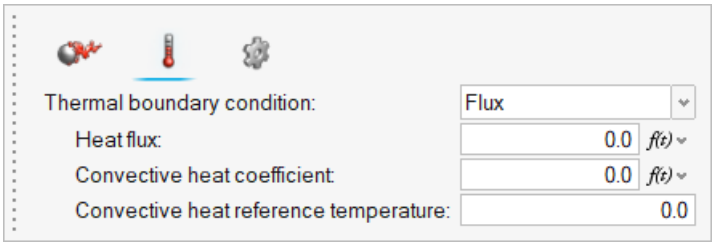


Figure 296:

28. Click ▸ on the guide bar.

29. Using the Boundaries legend, rename the surface group to `Fluid_Walls`.

Link Surface Output

1. In the Boundaries legend, right-click on **Flanges** and select **Edit**.
2. In the microdialog, activate the checkbox for **Create linked surface output**.

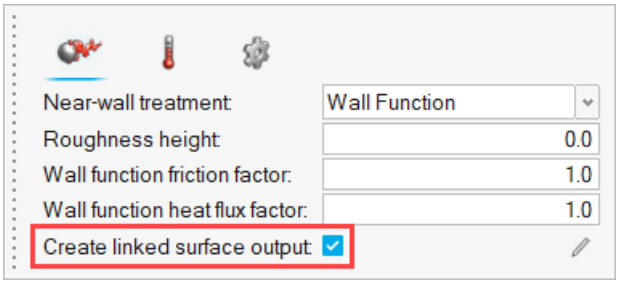


Figure 297:

3. Click ✎ to edit the values.
4. Set the following linked surface output settings for the inlets, the outlet, and all the wall surfaces (Outlet_End, Flange_Bolts, Outer_Solid_Walls, Fluid_Walls).

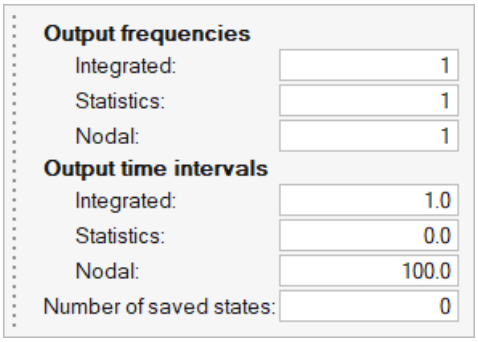


Figure 298:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 299:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand **Default initial conditions** and check that the Pre-compute flow and Pre-compute turbulence options are activated.
6. Set the Temperature to `273.16`.
7. Leave the remaining options as default and click **Run** to launch AcuSolve.

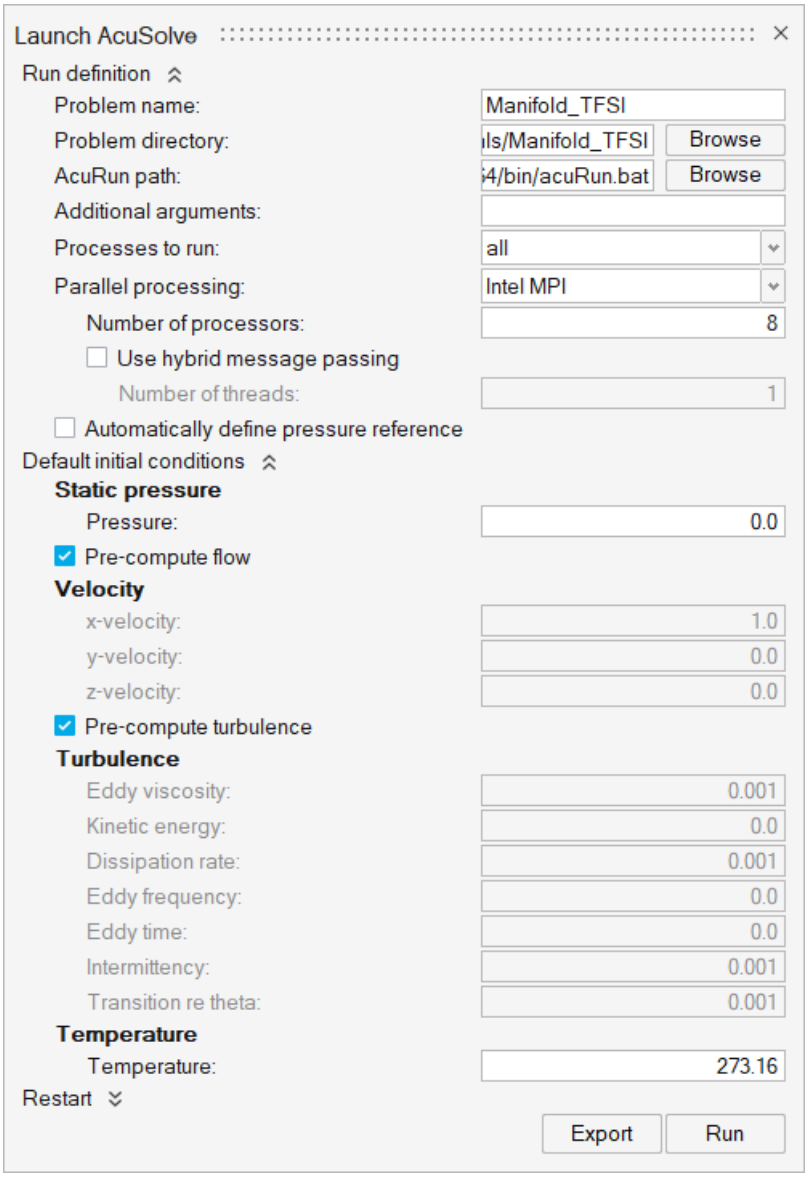


Figure 300:

Run acuOptiStruct

1. After AcuSolve finishes running, open the AcuSolve command prompt and `cd` to your working directory.
2. Execute the following command:

   ```
   acuOptiStruct -solids "Exhaust Steel" -spcsurfs "Flange_Bolts - Output","Outlet_End - Output" -spcsurfsdof 123456,123456 -spcsurfsdofvals 0,0 -type sl
   ```

   You should see a similar output as below when the command executes successfully.

```
acuOptiStruct:                      Problem = Manifold_TFSI
acuOptiStruct:                  Working dir = ACUSIM.DIR
acuOptiStruct:                          Run =          1
acuOptiStruct:    No. solid            Nodes =      46914
acuOptiStruct:    No. solid Tet      Elements =     149687
acuOptiStruct:    No. solid Pyramid  Elements =          0
acuOptiStruct:    No. solid Wedge    Elements =          0
acuOptiStruct:    No. solid Brick    Elements =          0
acuOptiStruct:               Min temperature = 4.112640e+002
acuOptiStruct:               Ave temperature = 5.876673e+002
acuOptiStruct:               Max temperature = 7.012289e+002
acuOptiStruct:               Std temperature = 8.659359e+001
acuOptiStruct:                     Time Step =         31
acuOptiStruct:      Creating OptiStruct file = Manifold_TFSI_steady31.fem
acuOptiStruct:         Total CPU/Elapse time = 4.562000e+000 4.691000e+000 Sec
acuOptiStruct:            Total Memory Usage = 6.189063e+001 Mbytes
```

Figure 301:

Run OptiStruct

1. Start OptiStruct from the Windows Start menu by clicking **Start** > **Altair <version>** > **Compute Console**.
2. Click 🗁 next to Input file(s).
3. Browse to the location that you are using as your working directory.
4. Select the `.fem` file.

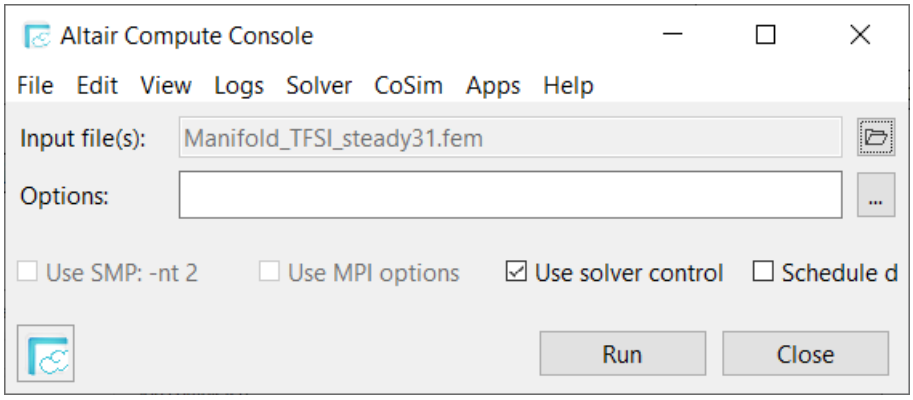


Figure 302:

5. Click **Run** to run the case.

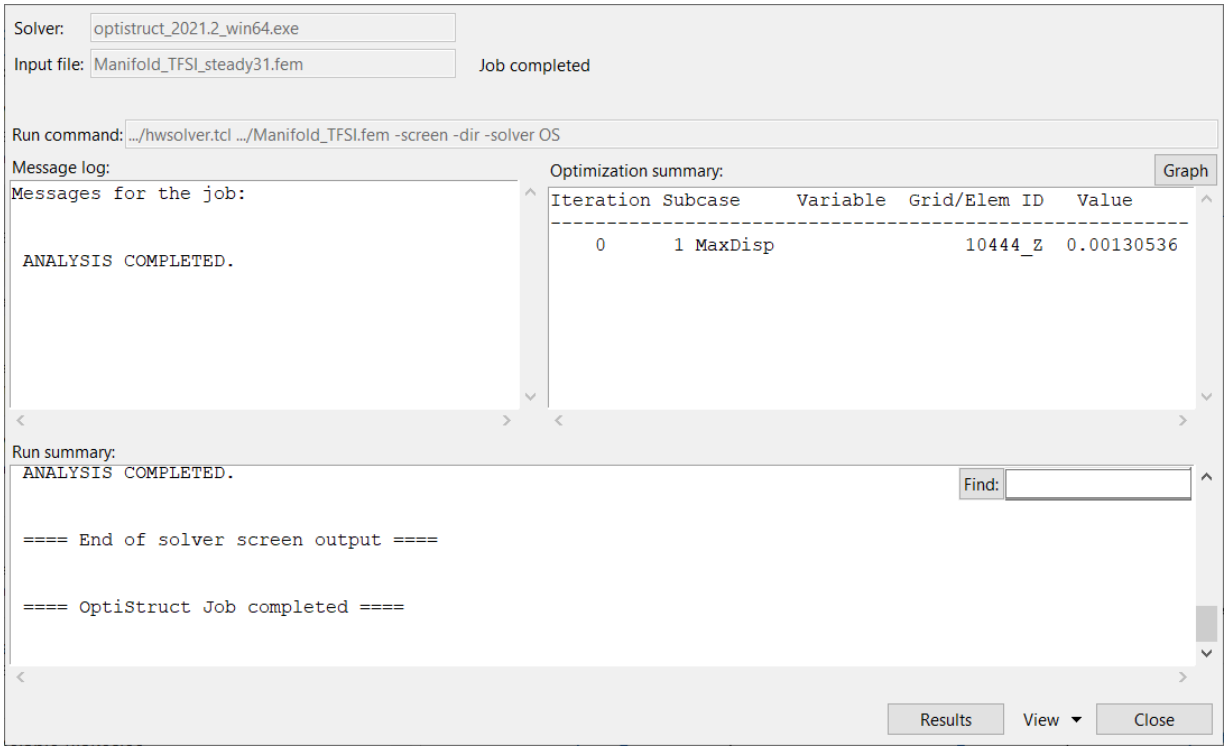


Figure 303:

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Boundary Groups** tool.

Figure 304:

5. Select all the surfaces.
6. In the microdialog, set the display to **temperature** and activate the **Legend** toggle.
7. Click ≡ ˅ and set the Colormap name to **Rainbow Uniform**.

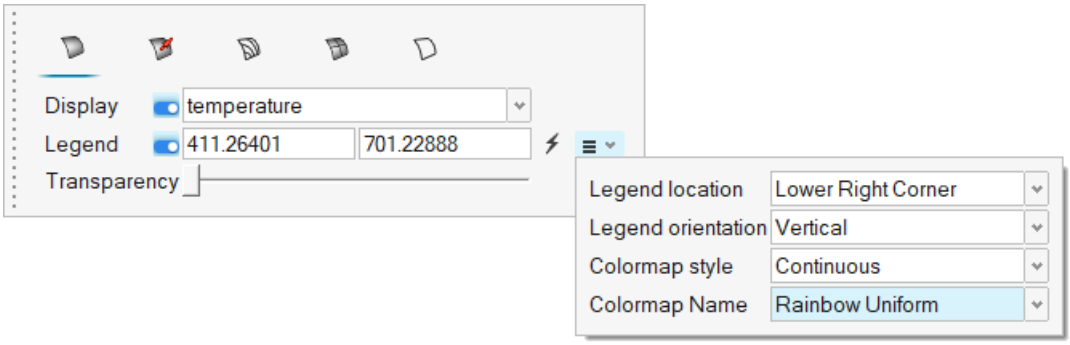


Figure 305:

8. Click ✓ on the guide bar.

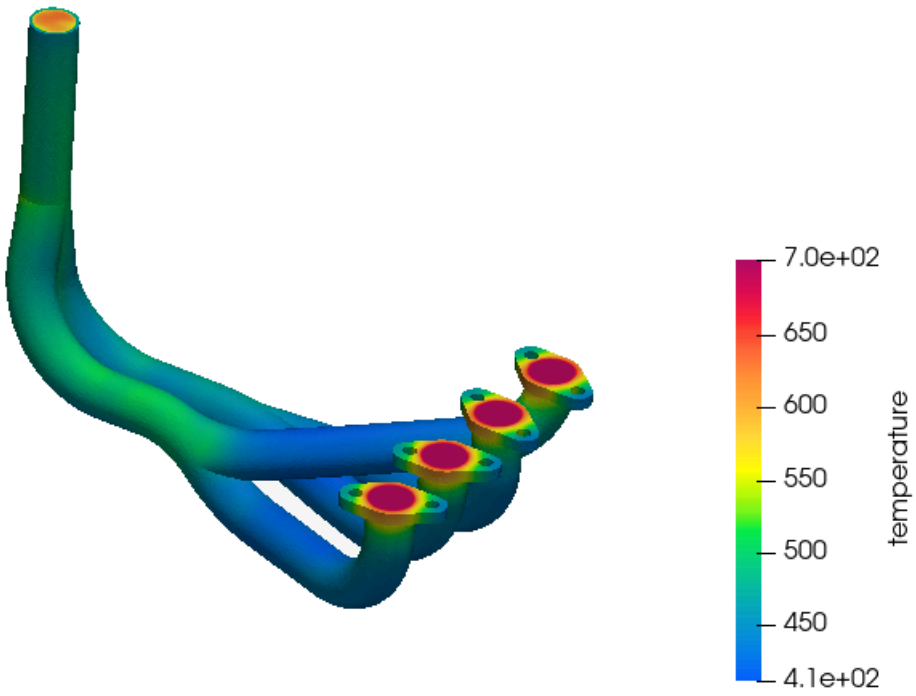


Figure 306:

9. In the Post Browser right-click on **Fluid_Walls** and select **Edit**.

10. In the microdialog, change the display to **pressure** and activate the **Legend** toggle.

11. Click ≡ and set the Colormap name to **Rainbow Uniform**.

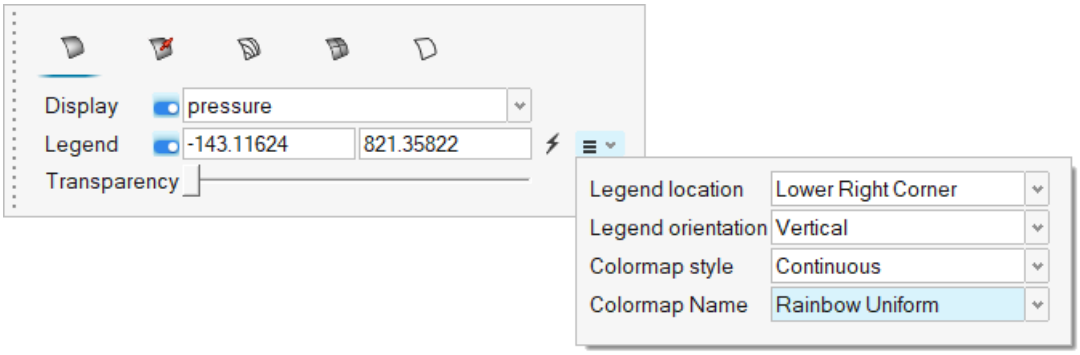


Figure 307:

12. Hide all other flow boundaries.

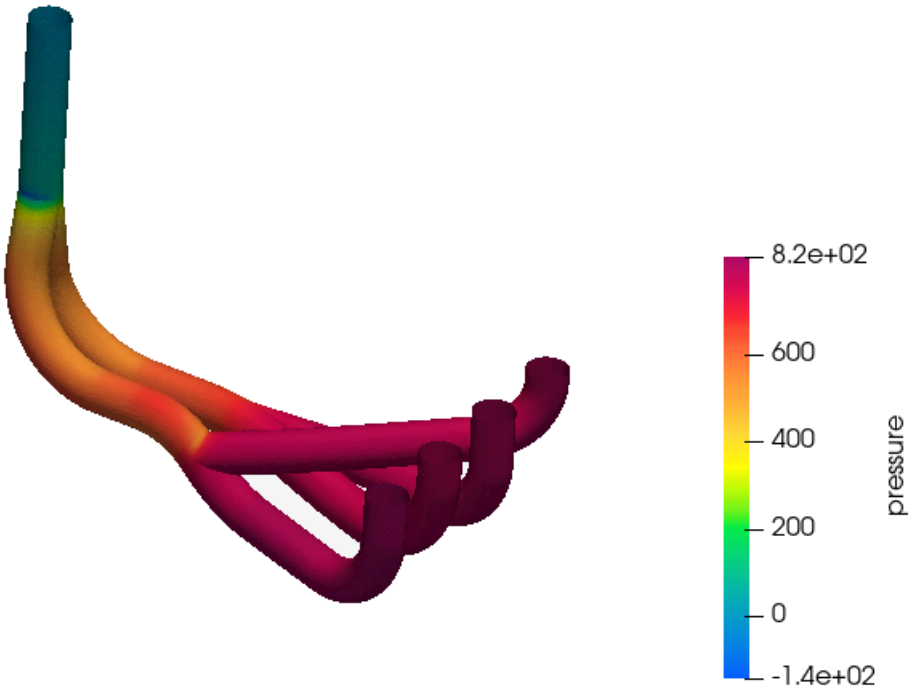


Figure 308:

Post-Process the OptiStruct Results with HyperView

1. Start HyperView from the Windows Start menu by clicking **Start** > **All Programs** > **Altair <version>** > **HyperView**.

Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.

2. In the Load model and results panel, click next to Load model.

3. In the **Load Model File** dialog, navigate to your working directory and select `Manifold_TFSI.h3d`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results from the OptiStruct results file.
   Observe that this model contains only the solid domain since only the solid is included while generating the OptiStruct solver deck.
6. Click on the Results toolbar to open the Contour panel.
7. In the panel area, set the Result type to **Displacement (v)**.
8. Click **Apply** to plot the displacement magnitude contours.

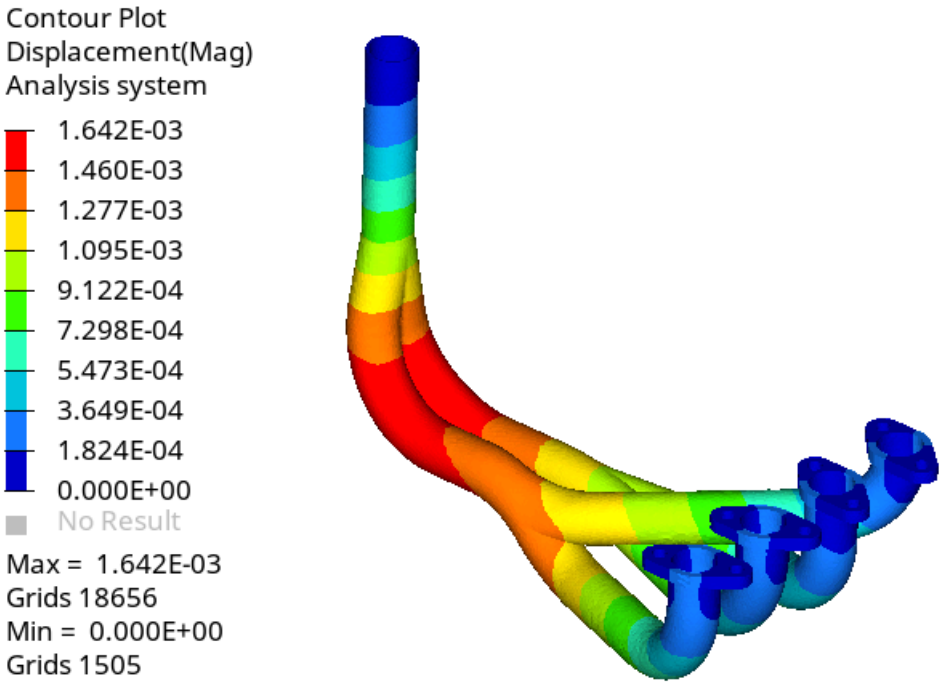


Figure 309:

9. Change the Result type to **Element Stresses (2D & 3D) (t)** and select **vonMises** from the drop-down below.
10. Click **Apply**.

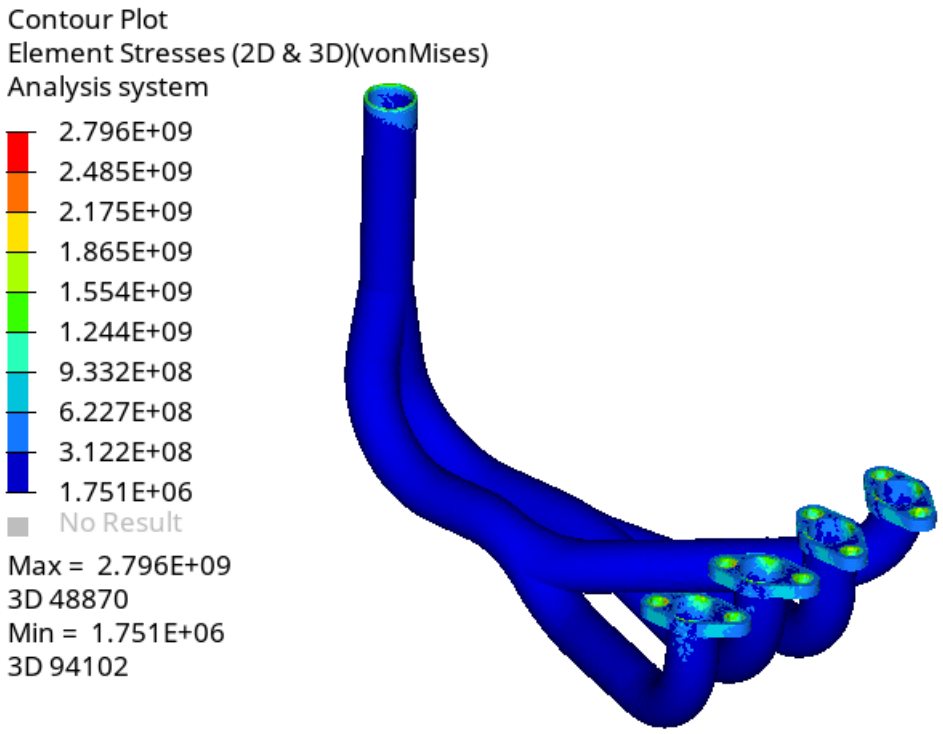


Figure 310:

Summary

In this tutorial, you learned how to set up a conjugate heat transfer problem using HyperMesh CFD and solve it using AcuSolve. Once you computed the solution, you used acuOptiStruct to generate the input deck for OptiStruct. Once the solution for the structural analysis was computed, you post-processed the results using HyperMesh CFD post and HyperView and created contour plots of Temperature, Pressure, Displacement, and Stress.

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/obm5781ohi

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial introduces you to the workflow for setting up a thermal radiation problem in a headlamp using the enclosure radiation model in HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3200_headlamp.x_t`

Problem Description

The problem to be solved is shown schematically in Figure 311 and Figure 312. It consists of a simple headlamp with a housing, lens, and a bulb. The inner cavity of the bulb is filled with air and the wattage of the bulb is 1W, which is modeled as a volumetric heat source. The Boussinesq density model is used for the air to consider the natural convection effects in the fluid volume. The heat generated in the bulb is transferred by three means: conduction from the bulb to the housing, natural convection in the air volume, and radiation from the bulb surface to other surfaces. The external reference temperature is 300 K for the outer surfaces of the headlamp. You will use the Enclosure radiation model to simulate the surface to surface radiation.

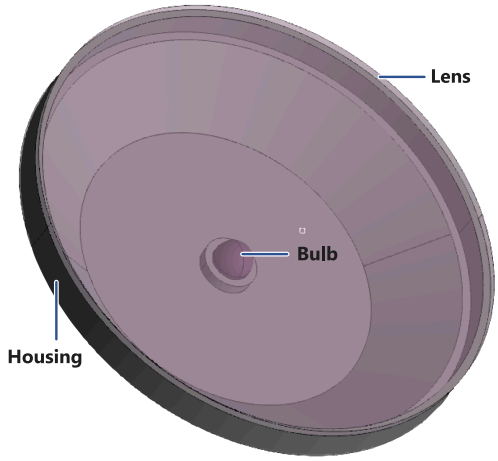


Figure 311:

The enclosure radiation methodology in AcuSolve involves a two-step process: view factor computation and heat flux addition. View factor is the proportion of radiation incident one surface due to another surface. The view factors are computed, and the radiative heat fluxes are added to the energy equation during the solver run. These radiative heat fluxes are computed based on the view factors using the Stefan-Boltzmann law. The enclosure radiation model is supported only on fluid mediums.

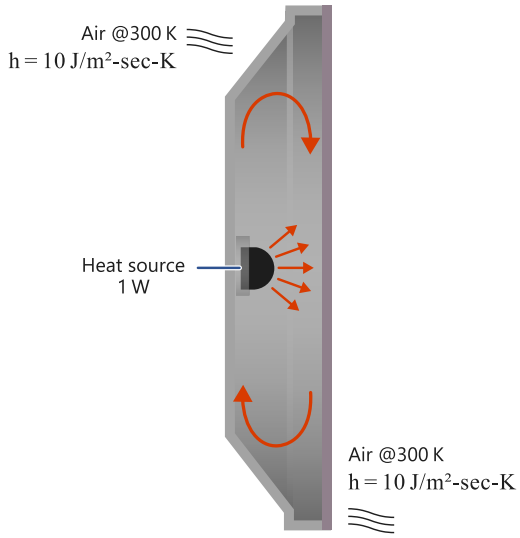


Figure 312:

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.

- From the Home tools, Files tool group, click the **Save As** tool.

Figure 313:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `Headlamp_Enclosure` as the name for the database then click **Save**.
   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T3200_headlamp.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

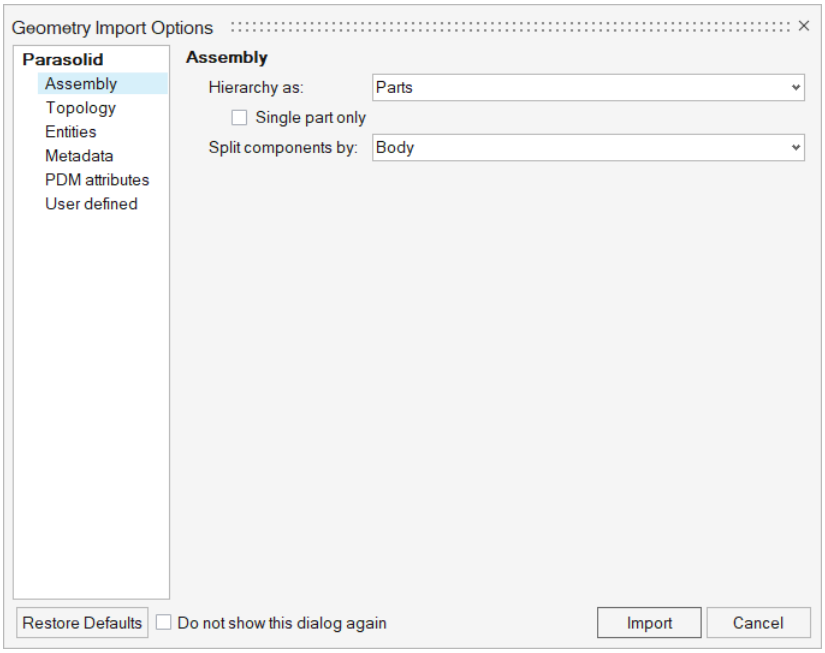


Figure 314:

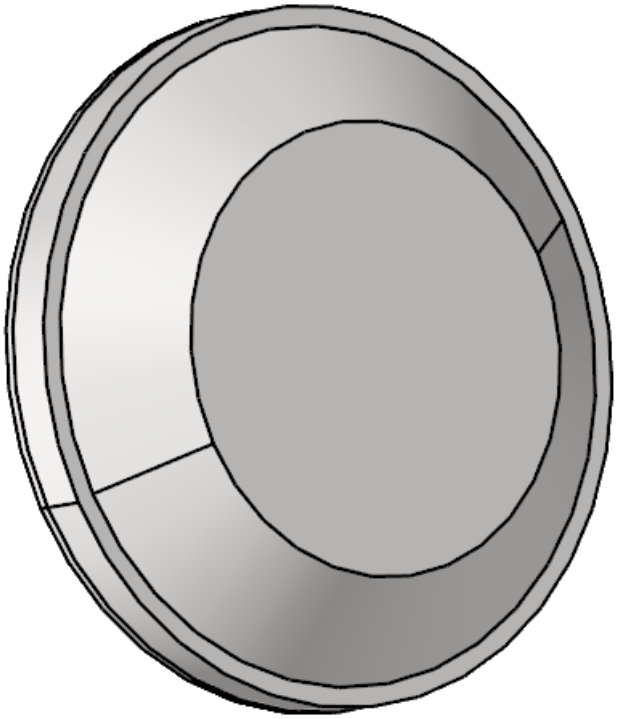


Figure 315:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 316:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The surface and solid errors display in the list below the tool.

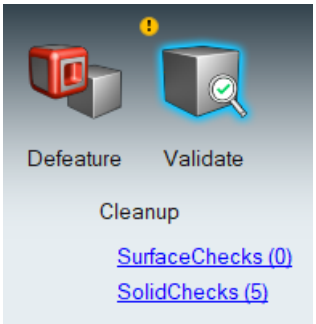


Figure 317:

2. Click **SolidChecks**.
   The Solid Repair tool opens, which you can use to fix the geometric errors in the model.

   From the SolidChecks legend, you can see the model's solids have five intersections.

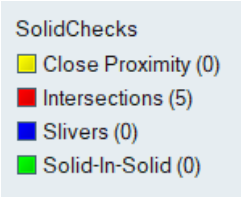


Figure 318:

3. Click **Intersections**.
   A guide bar used to fix intersecting solids displays.
4. Optional: Click → and ← to review each error.
5. Activate the **Keep common interface** option then click **Combine All**.
   The SolidChecks legend should now display zero for all errors.
6. Click the **Validate** tool once again.
   Observe that a blue check mark now appears on the top-left corner of the tool icon. This indicates that no issues are detected and you are ready to continue.

Figure 319:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

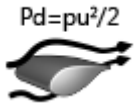


Figure 320:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Laminar** as the Turbulence model.
   c) Activate the **Include gravitational acceleration** checkbox and set the gravity in the y direction to `-9.81`.
   d) Activate the **Heat transfer** checkbox.

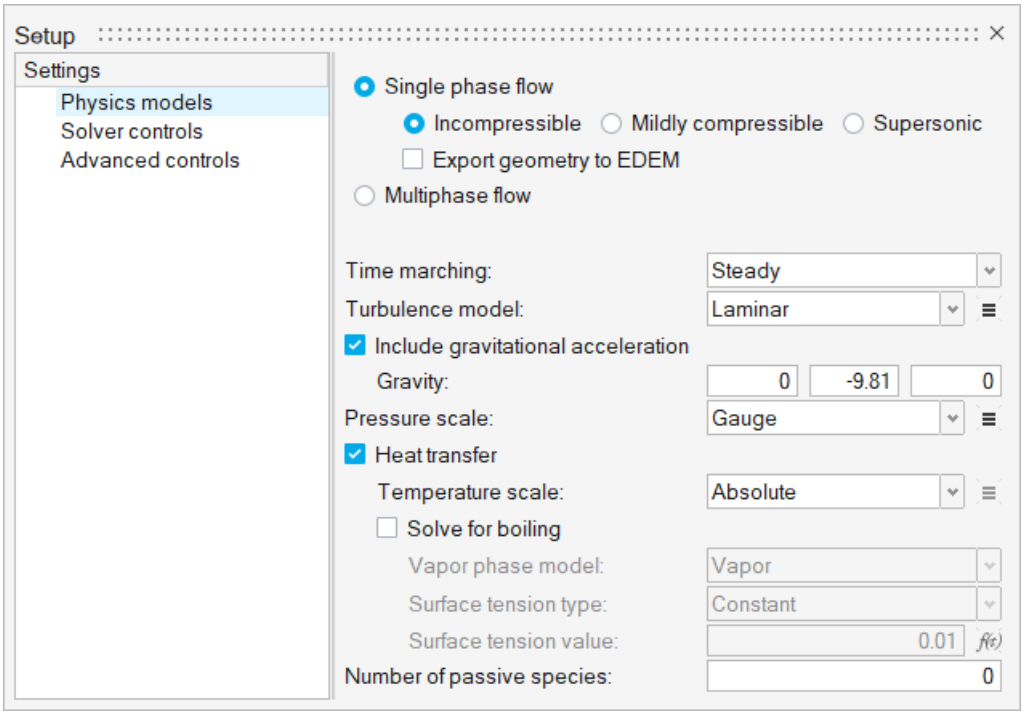


Figure 321:

3. Click the **Solver controls** setting and activate the **Thermal flow** equation.

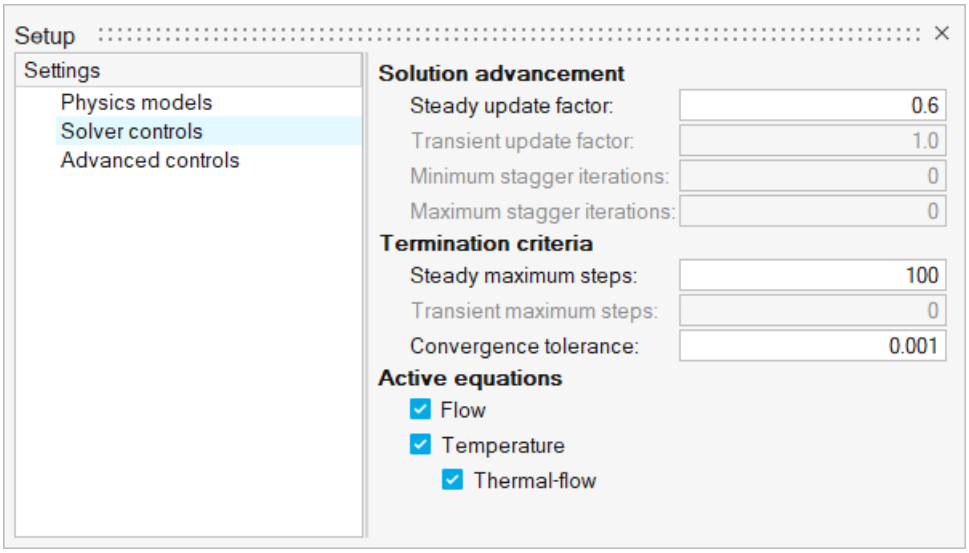


Figure 322:

4. Close the dialog and save the model.

Define the Material Models

1. From the Flow ribbon, click the **Material Library** tool.

Figure 323:

The **Material Library** dialog opens.

2. Click the **My Materials** tab.
3. Click + to add a new fluid material model.
4. In the material creation dialog, click the name in the top-left corner and rename the material to `Air_Boussinesq`.
5. In the Density tab,
   a) Set the Type to **Boussinesq**.
   b) Set the Density value to `1.225`.
   c) Set the Expansivity value to `0.00347222`.
   d) Reference temperature value to `288`.

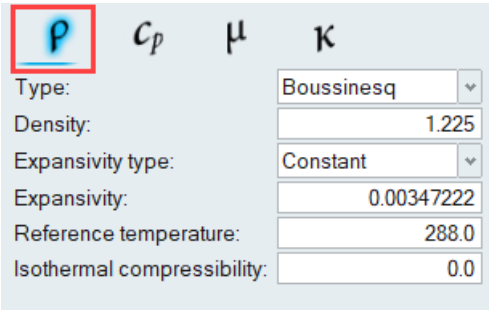


Figure 324:

6. Click the **Specific Heat** tab and set the Specific heat value to `1005`.

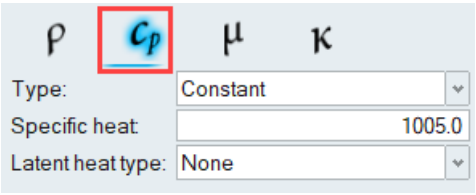


Figure 325:

7. Click the **Viscosity** tab and set the Viscosity value to `1.781e-05`.

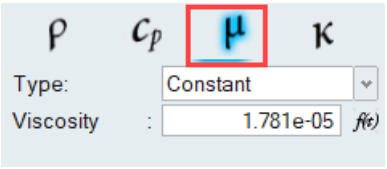


Figure 326:

8. Click the **Conductivity** tab and set the Conductivity value to `0.02521`.

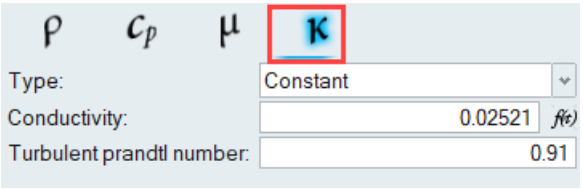


Figure 327:

9. Close the material creation dialog to return to the **Material Library** dialog.
10. Select **Solid** in the Settings menu, click the **My Materials** tab, then click **+** to create a new solid material model.
11. Name the material `Plastic` and set the following values.

    The Type should be Constant for each property.

    - Density: `1270`

- Specific Heat: `1900`
- Conductivity: `0.2`

12. Close the material creation dialog to return to the **Material Library** dialog.
13. Similarly, create new solid material models named `Arnite` and `LED` with the following properties. The Type should be Constant for each property.

    `Arnite`:

    - Density: `1670`
    - Specific Heat: `2050`
    - Conductivity: `1.6`

    `LED`:

    - Density: `5500`
    - Specific Heat: `0.3`
    - Conductivity: `5.0`

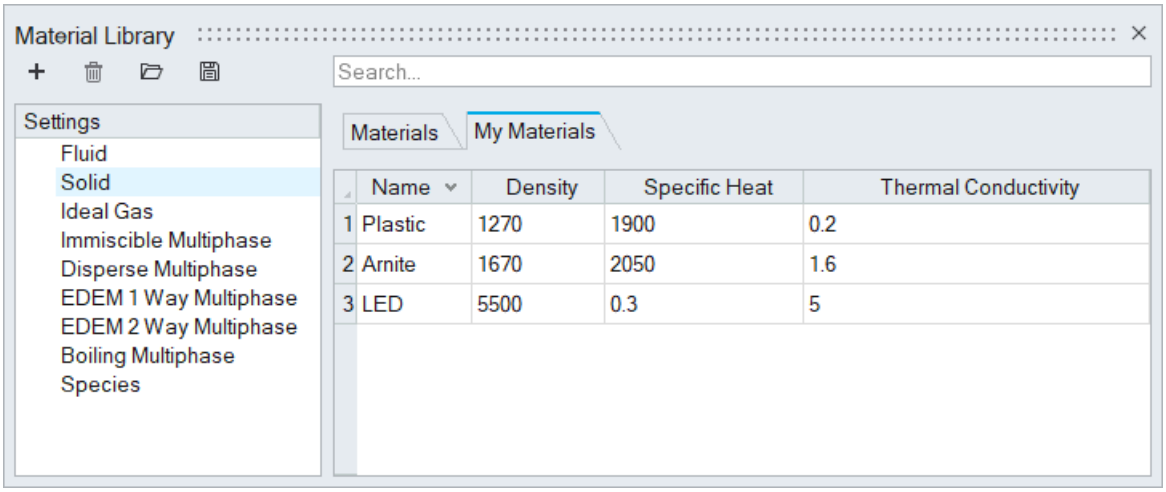


Figure 328:

14. Close all dialogs and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 329:

2. Click the lens volume highlighted in the figure below and select **Arnite** from the Material drop-down menu.

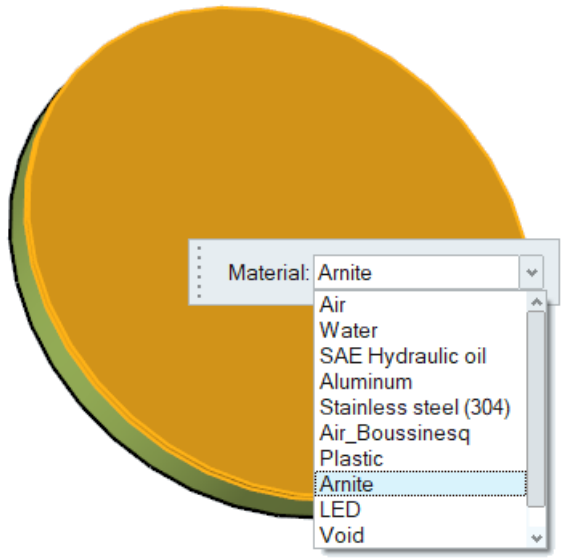


Figure 330:

3. On the guide bar, click ▸ to execute the command and remain in the tool.
4. Click the housing volume and assign the **Plastic** material model.

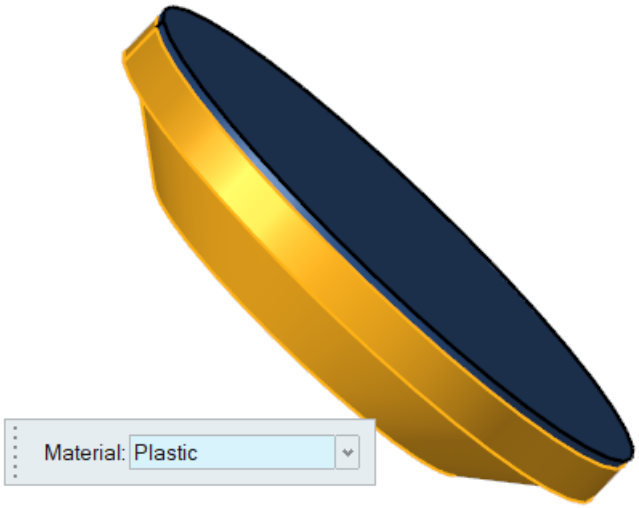


Figure 331:

5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. In the Materials legend, right-click on **Air** and select **Isolate**.
7. Click the air volume and assign the **Air_Boussinesq** material model.

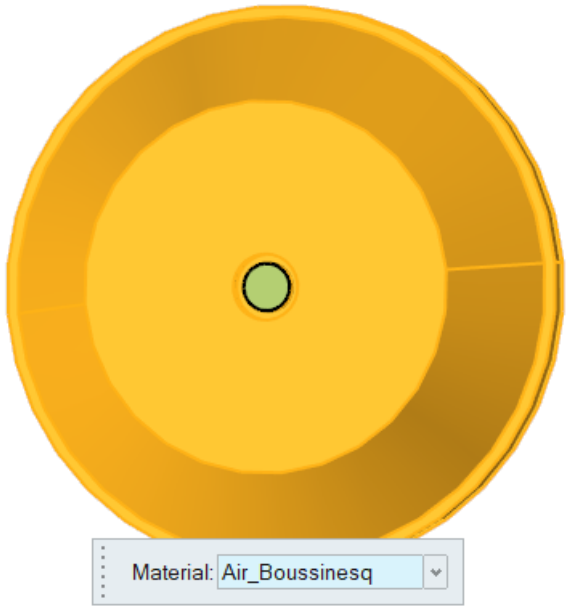


Figure 332:

8. On the guide bar, click ▸ to execute the command and remain in the tool.
9. In the Materials legend, right-click on **Air** and select **Isolate**.
10. Click the bulb volume and assign the **LED** material model.

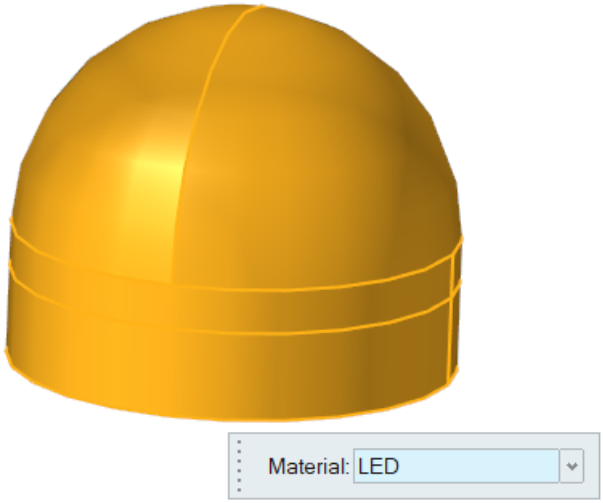


Figure 333:

11. On the guide bar, click ✓ to execute the command and exit the tool.
12. Save the model.

Define the Heat Source

1. From the Flow ribbon, click the **Sources** > **Heat** tool.

Figure 334:

2. In the modeling window, select the bulb volume.
3. In the **Heat Source** dialog, set the heat source value to `2049180` W/m^3.

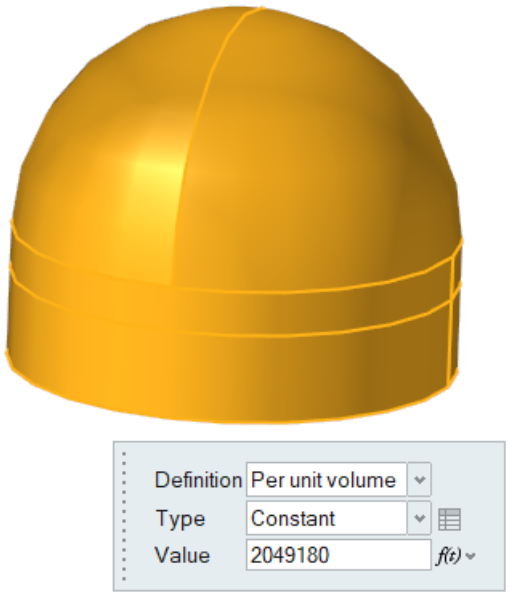


Figure 335:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Press **Esc** to exit the Sources tool then press the **A** key to turn on the display of all solids.
6. Save the model.

Define Flow Boundary Conditions

In this problem, all the surfaces are walls and will therefore be assigned the default wall boundary condition. The outer walls of the headlamp will be given a no-slip wall boundary condition with a convective heat flux boundary condition.

1. From the Flow ribbon, click the **No Slip** tool.

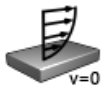


Figure 336:

2. In the modeling window, select the surfaces highlighted in the figure below.

Figure 337:

3. In the microdialog, enter the values shown in the figure below.

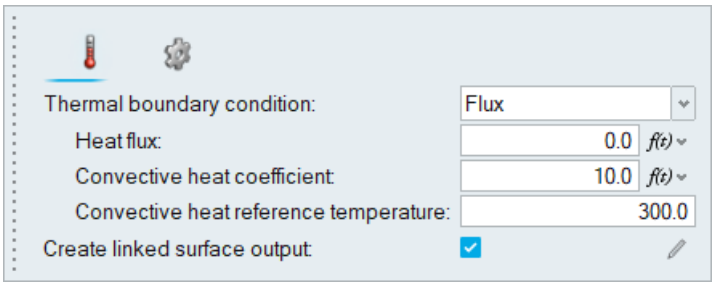


Figure 338:

4. In the Boundaries legend, double-click **Wall**, rename it to `Outerwalls`, then press **Enter**.
5. On the guide bar, click ✓ to execute the command and exit the tool.
6. Save the model.

Set Up Radiation

In this step, you will specify the parameters related to the thermal radiation setup.

Define Radiation Model Settings

1. From the Radiation ribbon, Thermal Radiation tools, click the **Physics** tool.

Figure 339:

The **Radiation Settings** dialog opens.

2. Activate **Thermal radiation** and set the Radiation model to **Enclosure** (if not set already).

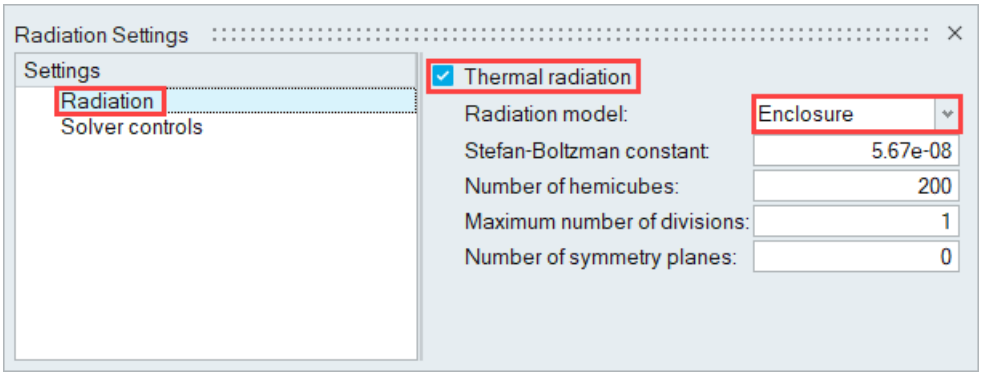


Figure 340:

3. Close the dialog.

Define the Emissivity Models

1. From the Radiation ribbon, click the **Surface Finish Library** tool.

Figure 341:

The **Surface finish library** opens.

2. Click + to add a new emissivity model.
3. Set the Name of the model to `Walls` and the Emissivity value to `0.7` by double-clicking on the entity fields.

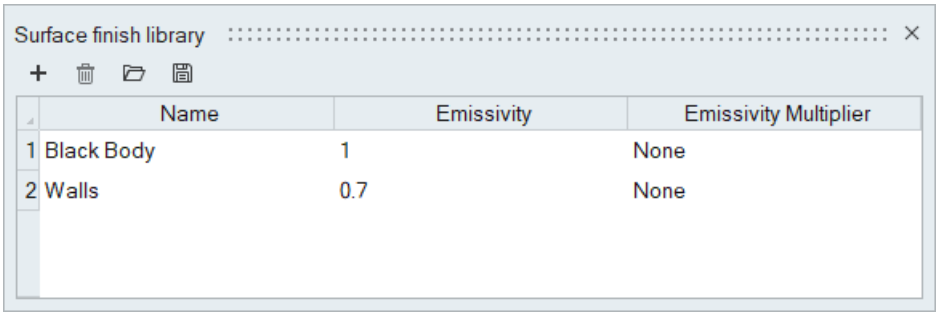


Figure 342:

4. Close the dialog.

Assign Surface Finish Models

1. From the Radiation ribbon, click the **Surface Finish** tool.

Figure 343:

2. Using the window selection method, select all the surfaces in the model.

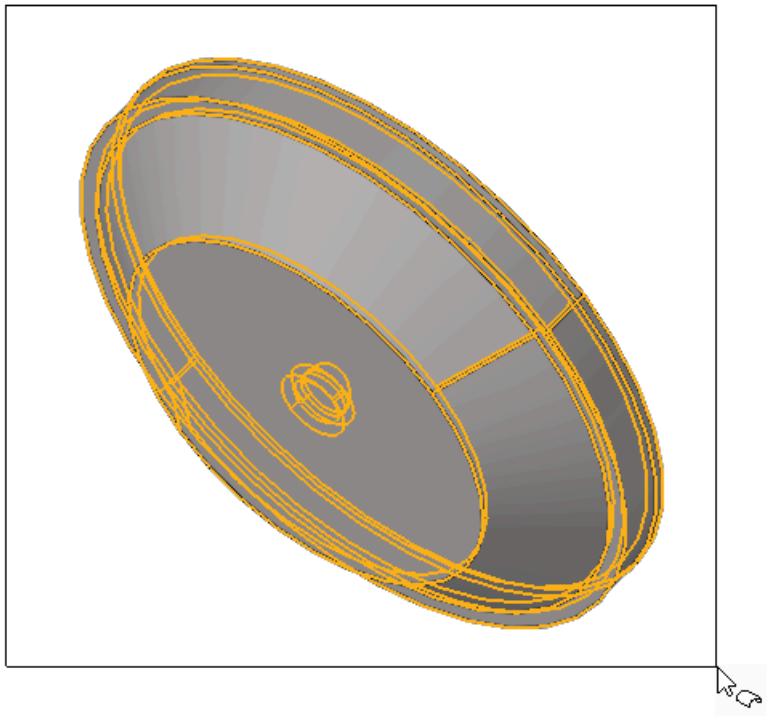


Figure 344:

3. In the microdialog, assign the **Walls** emissivity model.
4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Save the model.

Generate the Mesh

In this step, you will define the mesh controls and then generate the mesh.

Define the Surface Mesh Controls

1. From the Mesh ribbon, click the **Surface** tool.

Figure 345:

2. Using the window selection method, select all the surfaces in the model.
3. In the microdialog, set the Average element size to `0.002`.

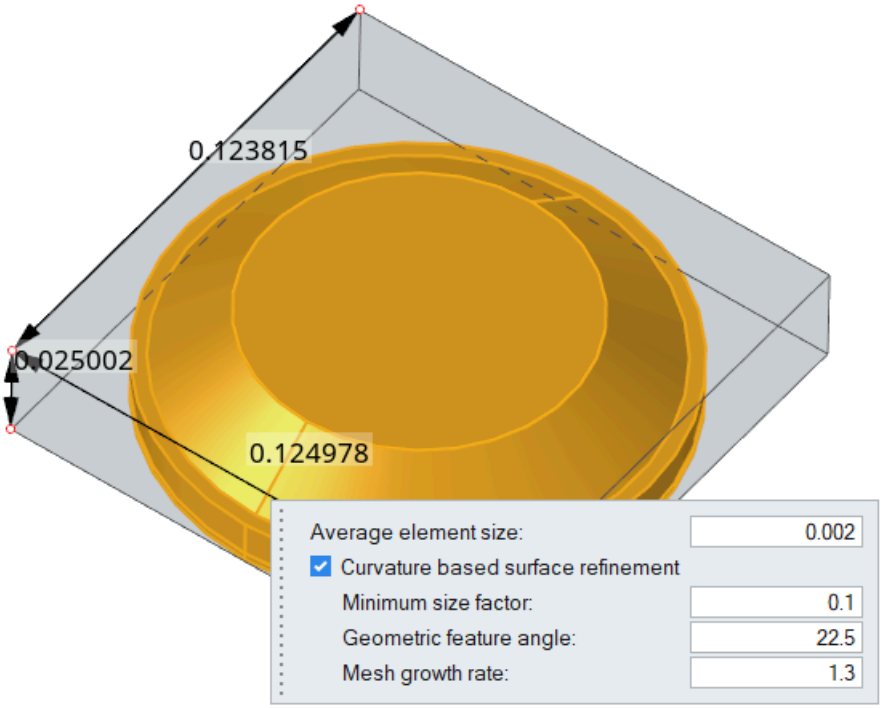


Figure 346:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Define the Boundary Layer Controls

1. From the Mesh ribbon, click the **Boundary Layer** tool.

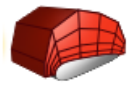


Figure 347:

2. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Material** > **Air_Boussinesq**.
   All the fluid wall surfaces should be selected and a microdialog for BL specification appears.
3. Enter the following values in the microdialog:
   - First layer thickness definition: **Constant**
   - First layer thickness: `0.0005`
   - Total number of layers: `4`
   - Growth method: **Constant**
   - Initial growth rate: `1.3`

- Termination policy: **Truncate**
- Activate the **Enable surface mesh modification** option

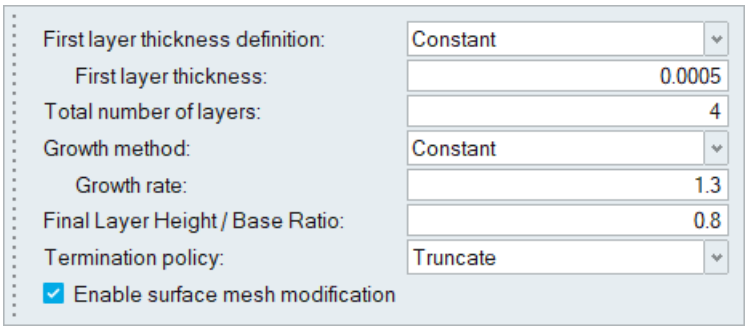


Figure 348:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Define the Volume Mesh Controls

Since the thickness of the housing and the lens solids are small, you will use the thin layer meshing tool so that when the volume mesh is generated, there will be two layers across the thickness of those solids.

1. From the Mesh ribbon, click the **Volume Mesh** tool.

Figure 349:

2. Select the housing and the lens solids.
3. In the microdialog,
   a) Set the Average size to `0.001`.
   b) Set the Growth rate to `1.0`.
   c) Activate the **Thin layer meshing** option and set the Number of layers to `2`.

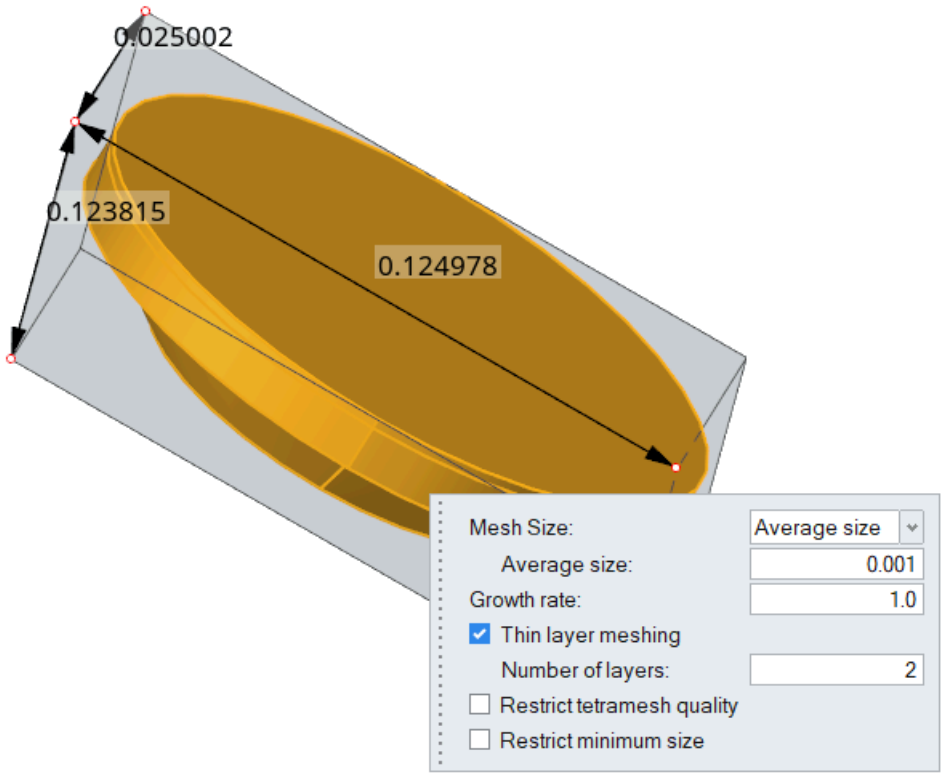


Figure 350:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

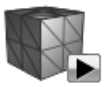


Figure 351:

The **Meshing Operations** dialog opens.

2. Set the Mesh growth rate to `1`.
3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 352:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand the **Default initial conditions** menu and deactivate the **Pre-compute flow** option.
5. Set the x-velocity to `0` and the Temperature to `300`.
6. Leave the remaining options as default and click **Run** to launch AcuSolve.

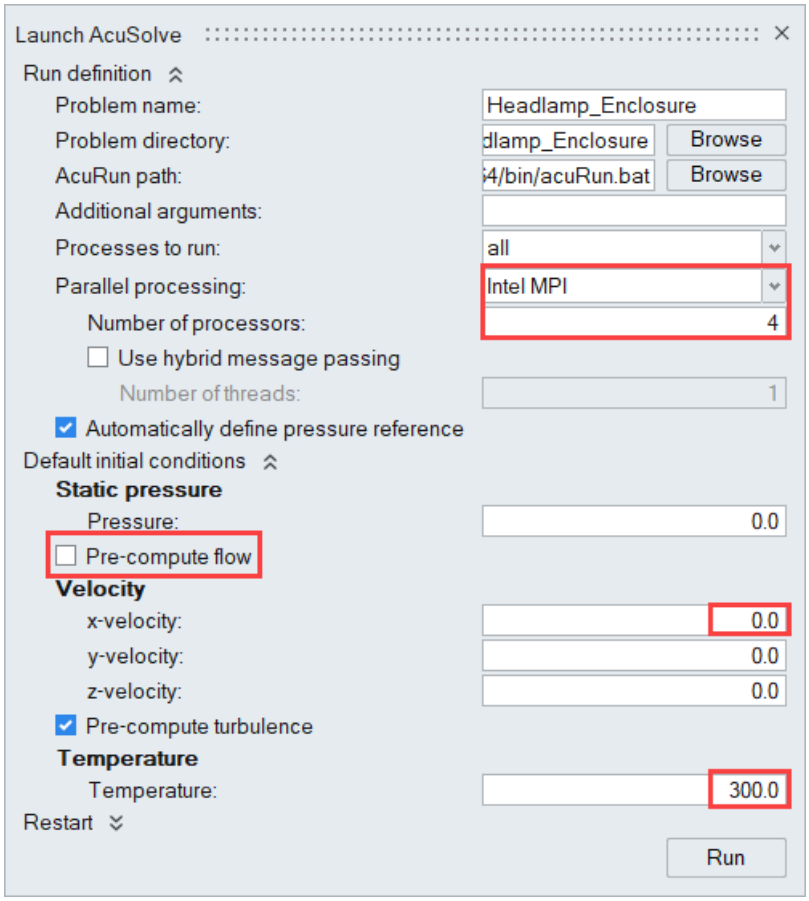


Figure 353:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Hide the **Outerwalls** boundary surface in the Post Browser.

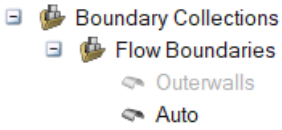


Figure 354:

5. Select the inner surface of the lens in the modeling window, right-click, and select **Hide**.

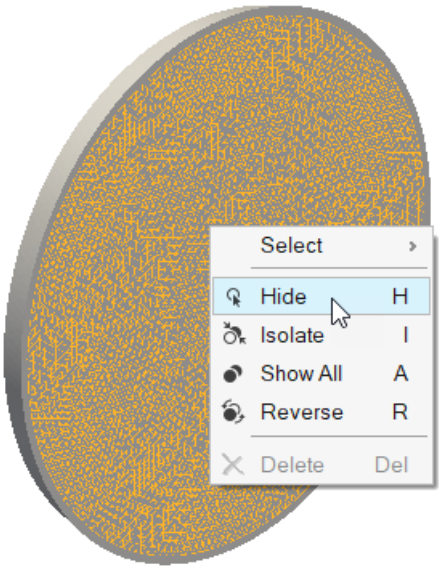


Figure 355:

6. Similarly, hide the outer fluid surface.
   You should end up with a view as shown below.

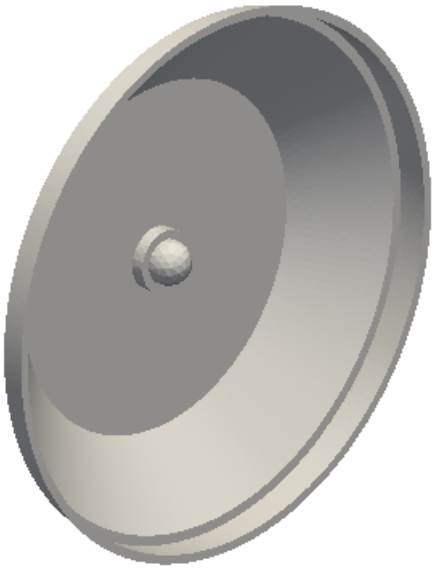


Figure 356:

7. Click the **Boundary Groups** tool.

Figure 357:

8. Select all visible surfaces in the modeling window.
   Rotate the model if needed to select the rear surfaces.
9. In the display properties microdialog, set the display to **temperature**.
10. Activate the **Legend** toggle and click ⚡ to reset the range.
11. Click ≡ ˅, set the Legend location to **Upper Right Corner**, and the Colormap Name to **Rainbow Uniform**.

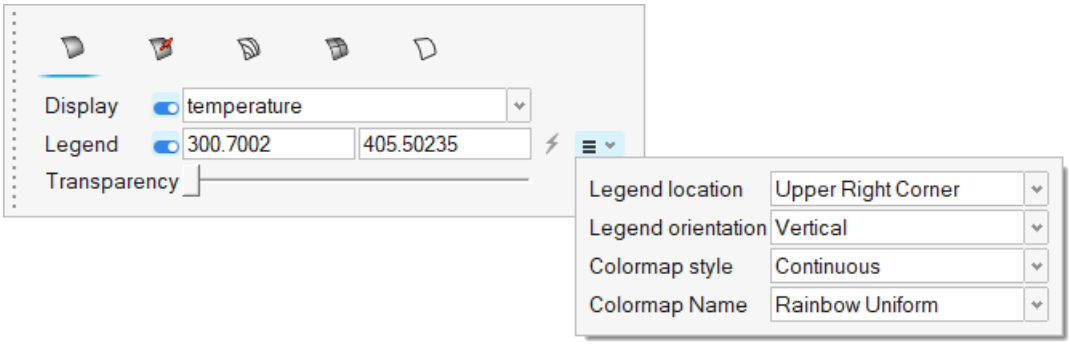


Figure 358:

12. Click ✓ on the guide bar.

13. Click the **Top** face on the View Cube to align the model.

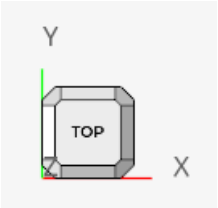


Figure 359:

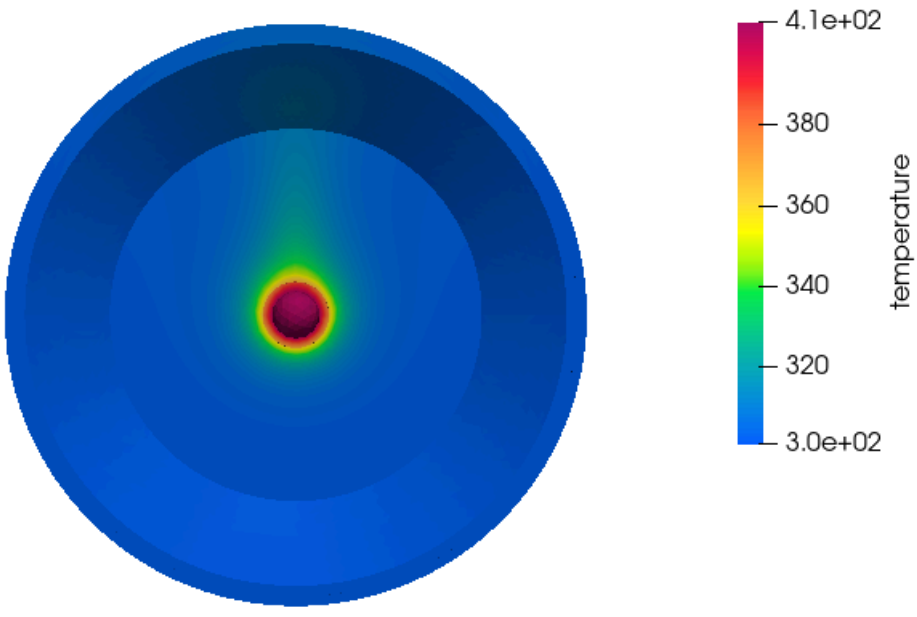


Figure 360:

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer problem in a headlamp using the enclosure radiation model in AcuSolve using HyperMesh CFD. You started by importing the headlamp geometry file, then you set up the simulation parameters and boundary conditions. Once the solution was computed, you processed the results using the Post ribbon where you created contour plots of temperature.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/006iqnnmz3

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

This tutorial introduces you to setting up a CFD simulation involving solar radiation and thermal shells using AcuSolve and HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3201_Atrium.x_t`

`SolarLoad.dat`

Problem Description

The problem to be addressed is shown schematically in Figure 361. The model consists of an atrium with a couch and chairs in the center. Air flows into the atrium through the inlet vent and exits through the outlet. The front portion of the atrium consists of glass walls supported by an aluminum frame. This aluminum frame will be modeled as a thermal shell; hence, this tutorial introduces you to the process of setting up a transient solar radiation simulation and thermal shells in HyperMesh CFD.

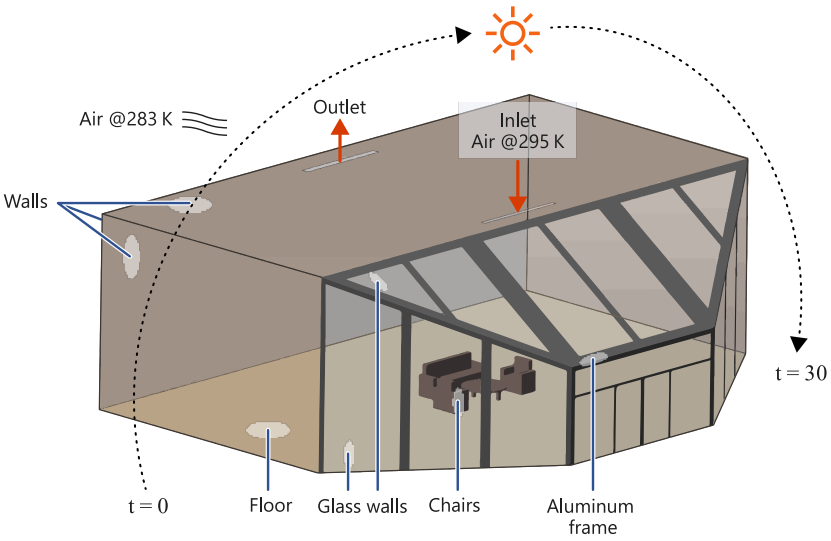


Figure 361:

Solar Radiation Parameters

AcuSolve uses an ideal gray surface solar radiation model to calculate the solar heat flux. The fluxes are computed using a ray trace algorithm and five optical properties of the surface, specular transmissivity (τ_s), diffuse transmissivity (τ_d), specular reflectivity (ρ_s), diffuse reflectivity (ρ_d) and absorptivity (a).

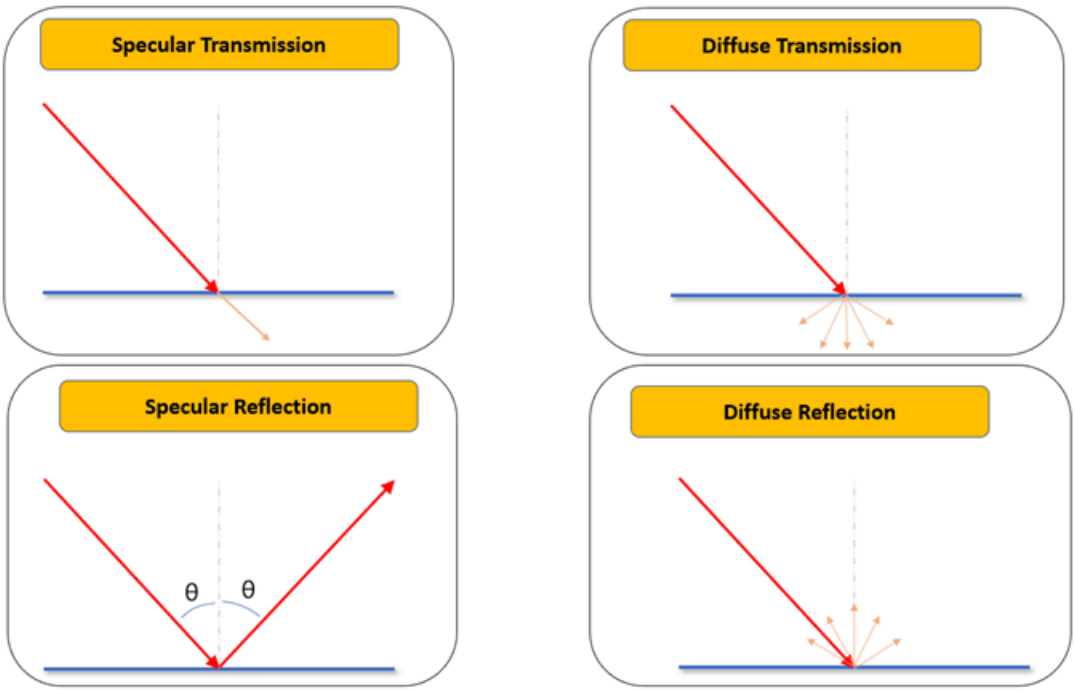


Figure 362:

A specular transmission occurs when a photon passes straight through a surface with no change of direction. In a diffuse transmission the photon penetrates the surface, but its outgoing energy is uniformly distributed in solid angle over the hemisphere, weighted by projected surface area. For a

specular reflection, the angle of reflection is equal to the angle of incidence. Diffuse reflections are similar to diffuse transmissions, except the hemisphere over which the outgoing energy is distributed is on the same side of the surface as the incident photon. Finally, the photon may be absorbed by the surface. These five interactions are associated with five surface properties that together must obey the following constraint:

$$\tau_s(\theta)+\tau_d(\theta)+\rho_s(\theta)+\rho_d(\theta)+\alpha(\theta)=1 \quad (3)$$

Where,

τ_s	Specular transmissivity
τ_d	Diffuse transmissivity
ρ_s	Specular reflectivity
ρ_d	Diffuse reflectivity
α	Absorptivity
θ	Angle of incidence

For the solar radiative heat fluxes to be computed, a solar radiation surface needs to be defined on that given surface.

In this tutorial, the solar flux loading is given in the form of a data file which was generated using the `acuSflux` script available in AcuSolve. The script can be used to generate a data file with a four-column array of solar flux vector data values. The piecewise linear type is used in this tutorial to emulate the pattern of sunrise to sunset over the atrium.

For example, to generate the solar load data file for a location with known geological coordinates, enter the following command in the AcuSolve Command Prompt: `acuSflux -time "dec-3-2019 11:00:00" -tinc 1800 -nts 25 -lat 42.6064 -lon -83.1498 -ndir "1,0,0" -udir "0,0,1"`

Here,

time	The start time in GMT (ex: "dec-3-2019 21:00:00")
tinc	The time increment in seconds
nts	Number of discrete time steps
lat	Latitude coordinates of the location in degrees North (ex: 45.112 or -37.56 (equal to 37.56 S))
lon	Longitude coordinates of the location in degrees East (ex: 86.26 or -54.84 (equal to 54.84 W))
ndir	The north direction unit vector in model coordinates (should be enclosed in double quotes) (ex: "0,1,0")
udir	The upward direction unit vector in model coordinates (should be enclosed in double quotes) (ex: "0,1,0")

Thermal Shell Modeling

The thermal shell in AcuSolve is a feature that creates zero physical thickness volumetric shell elements from surface elements. This is useful when the thickness of the component is too small to be modeled as a solid medium. The thermal shell can have multiple layers, each with different thicknesses and material models. A schematic of the thermal shell is shown below.

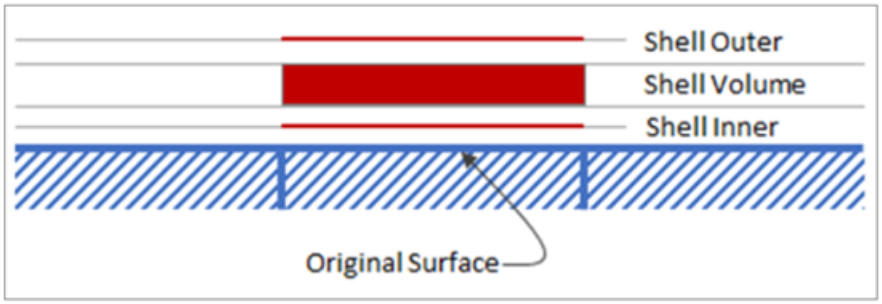


Figure 363:

When defining a thermal shell on a surface, two sets of boundary conditions are needed. One for the Primary Wall surface i.e. Shell Inner and one for the Shell Outer Wall surface. In this tutorial, a solar radiation surface will be defined on the outer shell surface so that it receives solar heat flux, whereas the inner shell surface will be modeled as a default wall.

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.
   - From the Home tools, Files tool group, click the **Save As** tool.

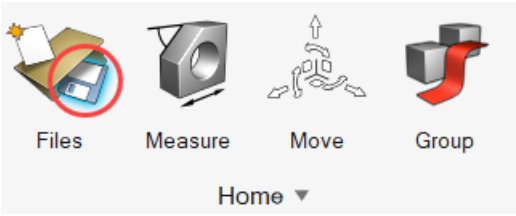


Figure 364:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `Atrium_Solar` as the name for the database then click **Save**.
   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T3201_Atrium.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

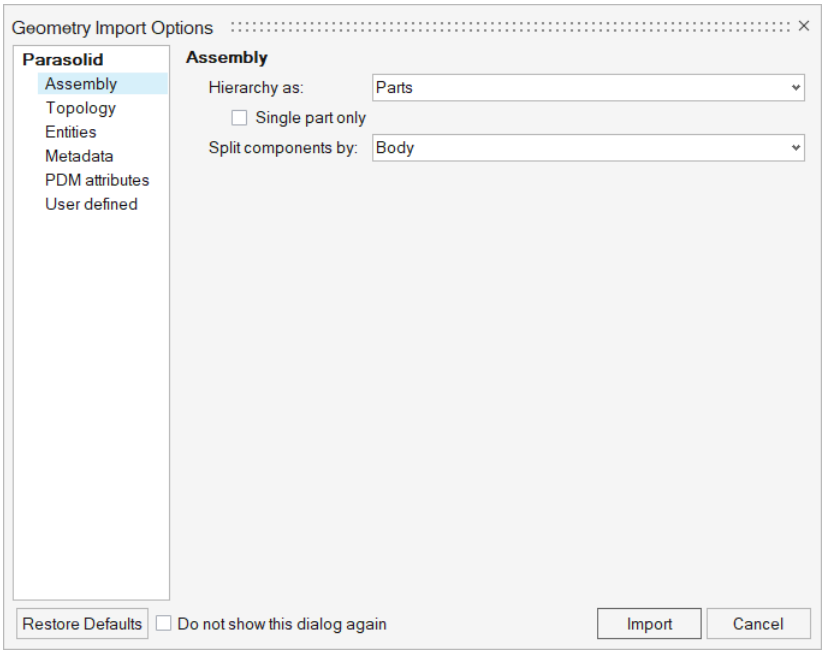


Figure 365:

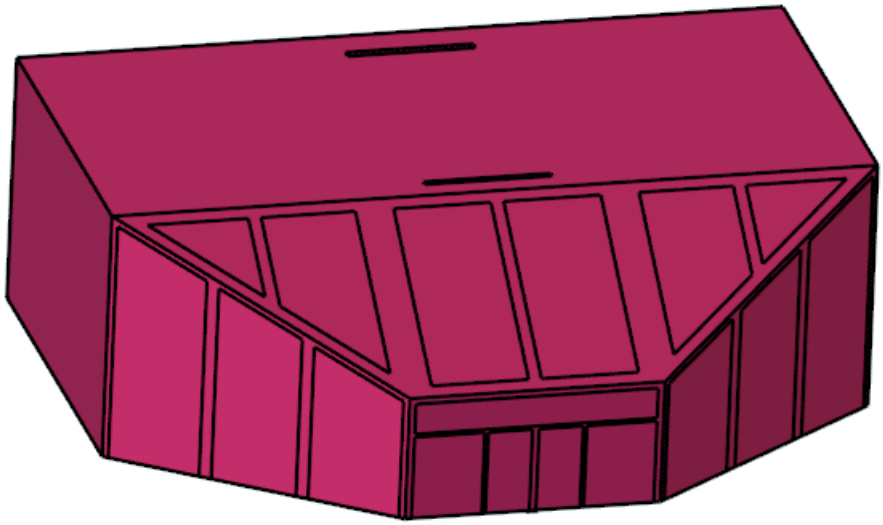


Figure 366:

The model contains an atrium with glass panes supported by an aluminum frame in the front. Air enters from the opening on the roof in the front and exits through the outlet in the rear.

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 367:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 368:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

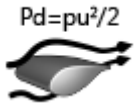


Figure 369:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `1` and the Final time to `30`.
   c) Set the Turbulence model to **Laminar**.
   d) Activate the **Heat transfer** checkbox.

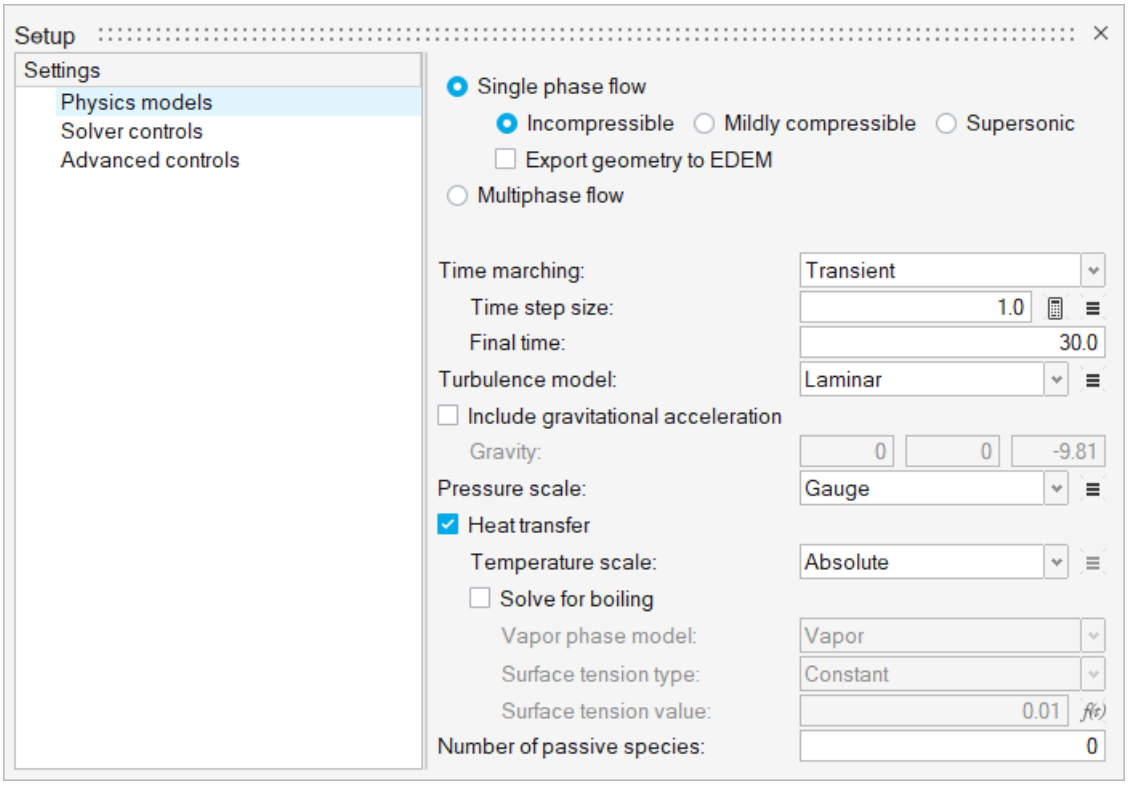


Figure 370:

3. Click the **Solver controls** setting and set the Maximum stagger iterations to `3`.

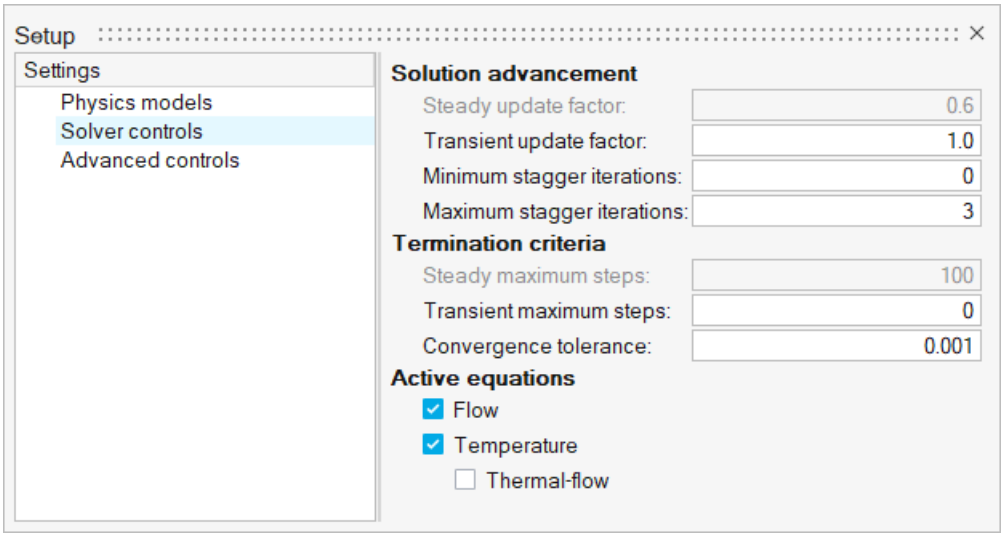


Figure 371:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 372:

2. Verify that Air is assigned as the material for the fluid domain.

   The legend in the top-left corner of the modeling window lists all the material models assigned to the current model.

   Since this model has a single volume, by default air is assigned as the material for the fluid domain.

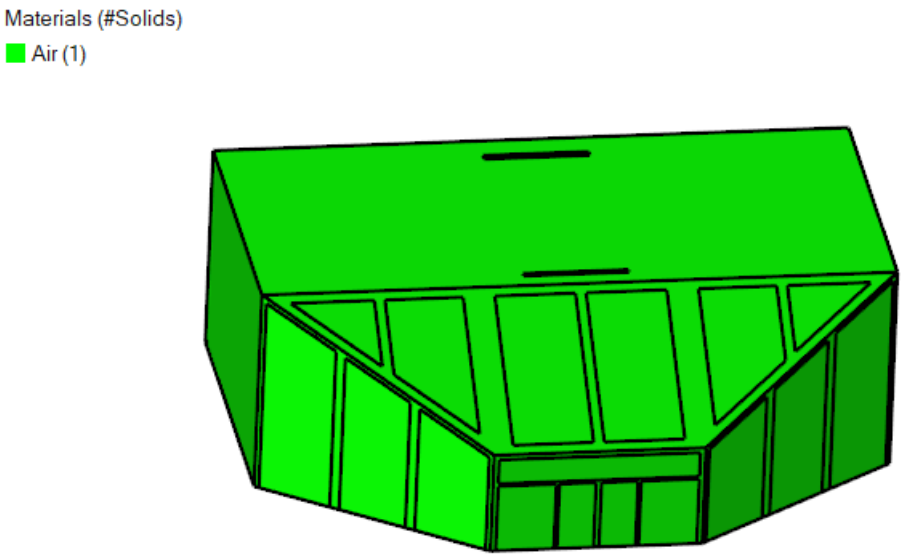


Figure 373:

3. On the guide bar, click ✓ to execute the command and exit the tool.

Define Thin Solids

In this simulation, you will model the aluminum frames as a thin solid.

1. From the Flow ribbon, click the **Thin** tool.

Figure 374:

2. In the modeling window, select the four surfaces highlighted in the image below.

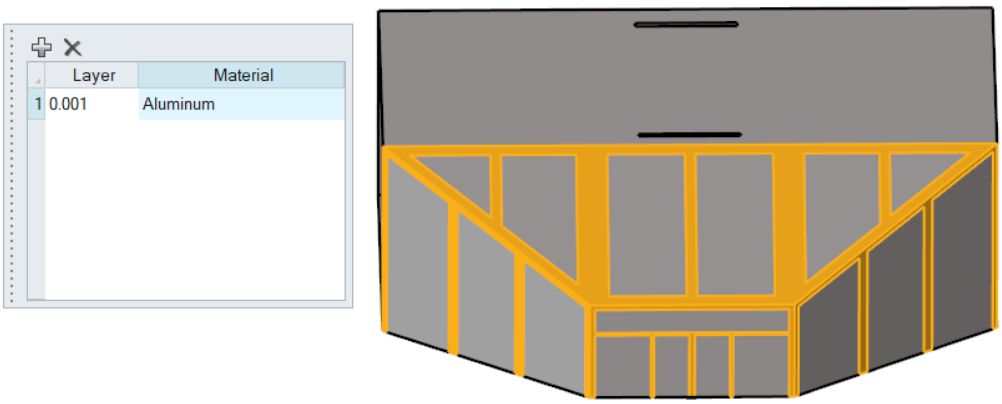


Figure 375:

3. In the microdialog, set the Layer thickness to `0.025` and the Material to **Aluminum**.
4. On the guide bar, verify that the number of Parent Surfaces selected is 4 then click ▶ to execute the command.
   Once the command is executed, the legend should be updated accordingly to reflect the changes.
5. Save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 376:

2. In the modeling window, click the inlet surface highlighted in the figure below.

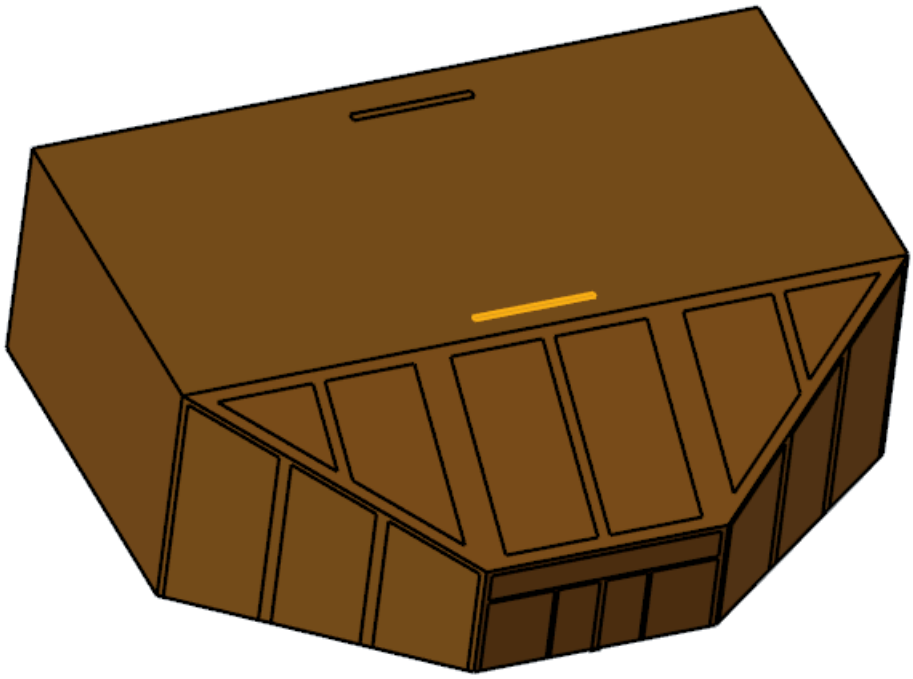


Figure 377:

3. In the microdialog, enter the values shown below.

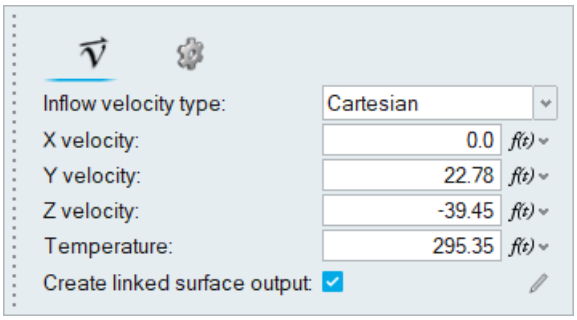


Figure 378:

4. Click ▸ on the guide bar to execute the changes.
5. Click the **Outlet** tool.

Figure 379:

6. Select the surface highlighted in the figure below then click ▸ on the guide bar.

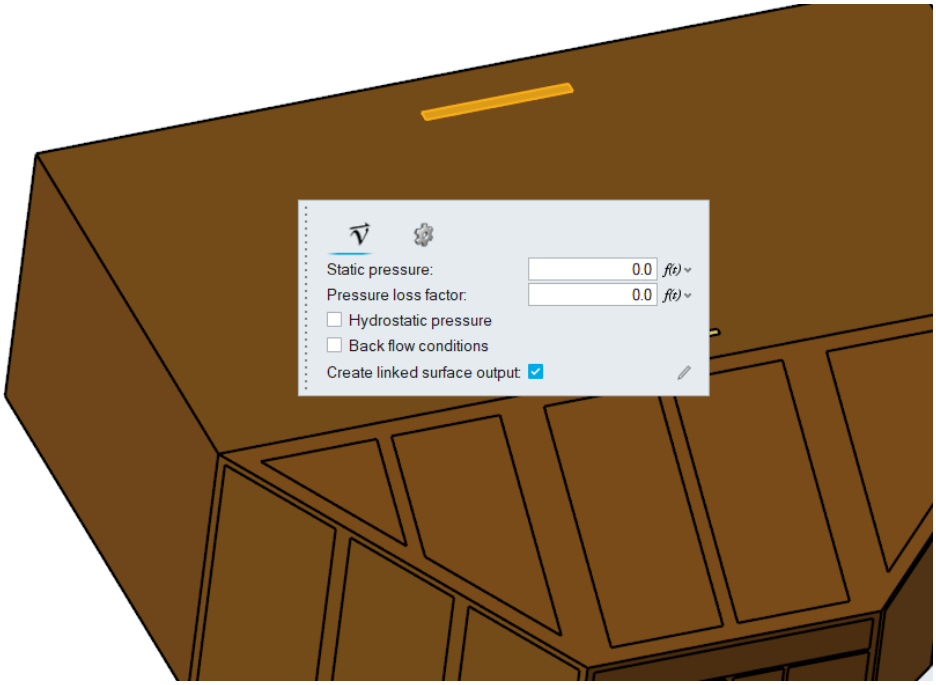


Figure 380:

7. Click the **No Slip** tool.

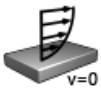


Figure 381:

8. Select all three wall surfaces of the atrium, the roof, and the front glass walls.

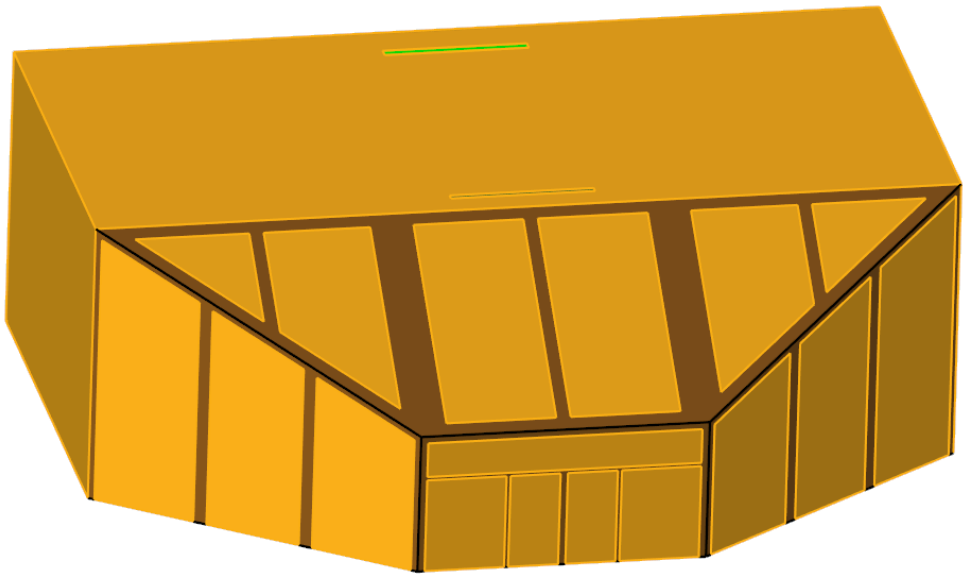


Figure 382: Front-Side of the Model

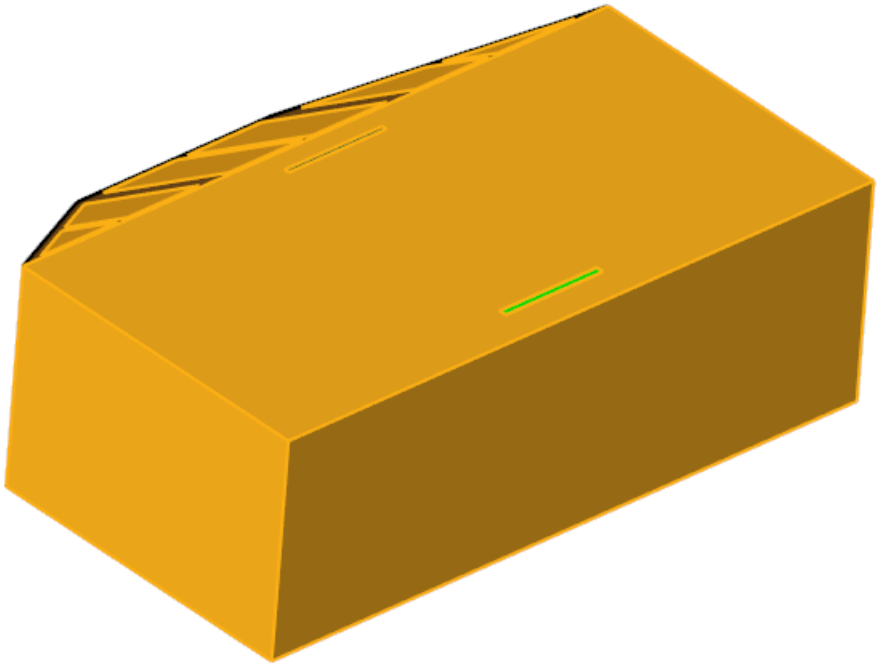


Figure 383: Back-Side of the Model

In total, 21 surfaces should be selected.

9. In the microdialog, enter the values shown in the figure below.

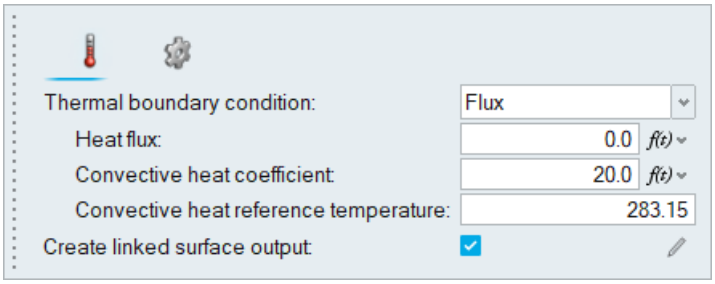


Figure 384:

10. Click ✎. In the new microdialog that appears, set the Nodal Output frequency to `1`.

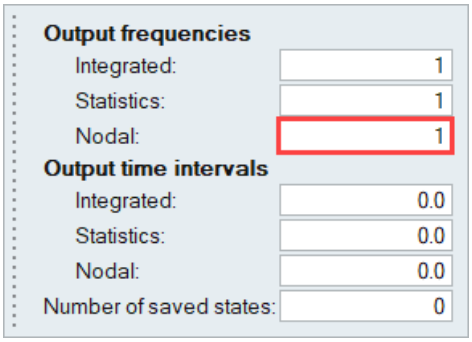


Figure 385:

11. On the guide bar, click ▸ to execute the command and remain in the tool.

12. On the guide bar, click the drop-down menu next to Surfaces and change the selection entity to **Thin Solids**.
The visibility of all the surfaces except the thin solids is made transparent.

13. Select all the thin solid surfaces using the window selection method.

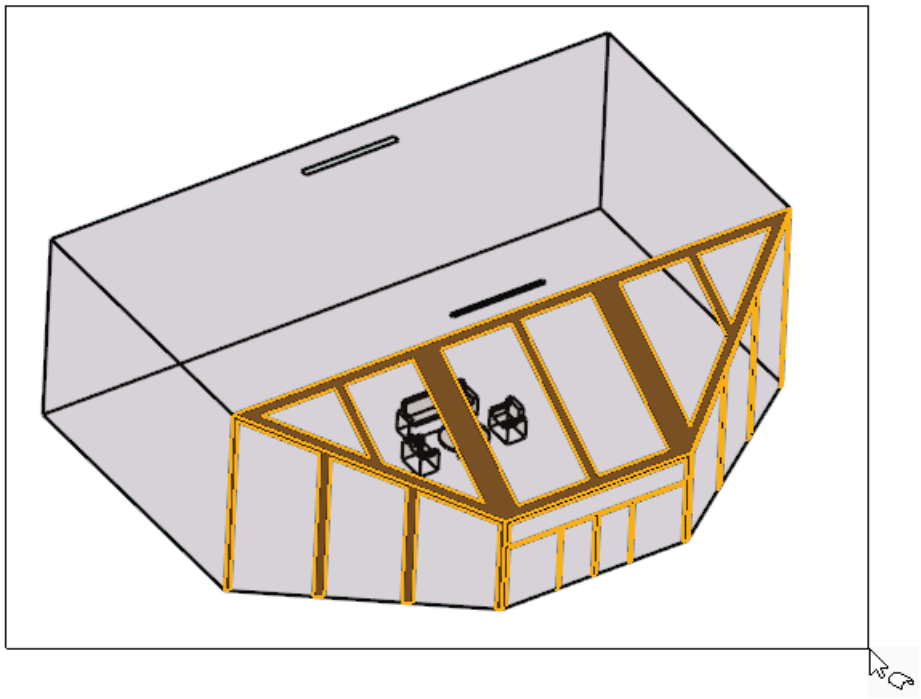


Figure 386:

14. In the microdialog, enter the values shown in the figure below.

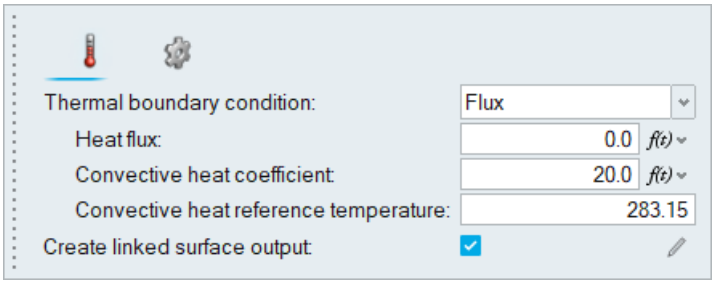


Figure 387:

15. Click ✎. In the new microdialog that appears, set the Nodal Output frequency to `1`.

16. On the guide bar, verify that the number of Thin Solids selected is 4 and the Direction is set to **Away from Parent Surface**, then click ▸.

17. Change the selection entity on the guide bar back to **Surfaces**.

18. From the Boundaries legend, right-click on **Default Wall** and select **Isolate**.

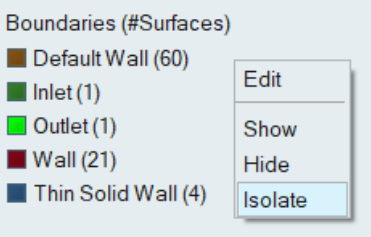


Figure 388:

19. Select all the surfaces except the aluminum frame, as highlighted in the figure below.

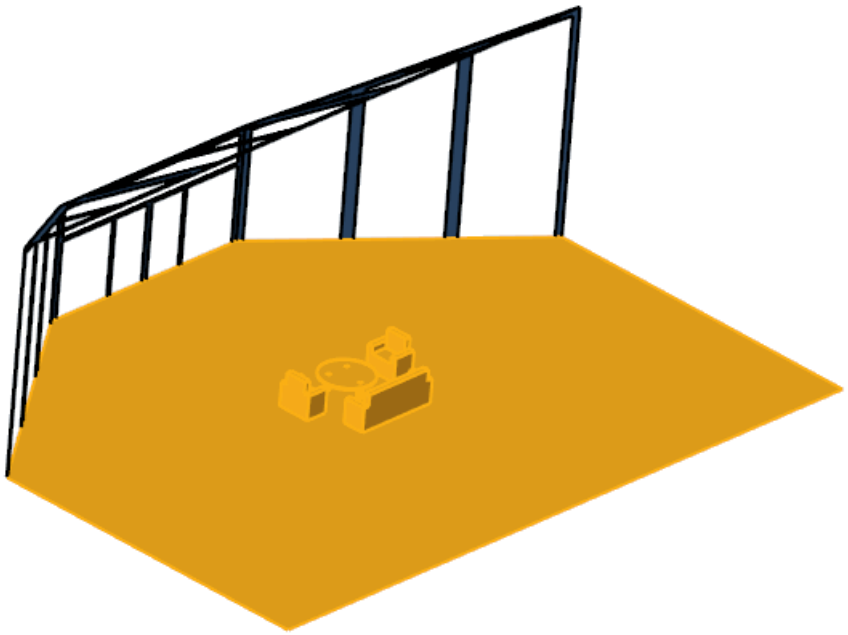


Figure 389:

20. In the microdialog, enter the values shown in the figure below.

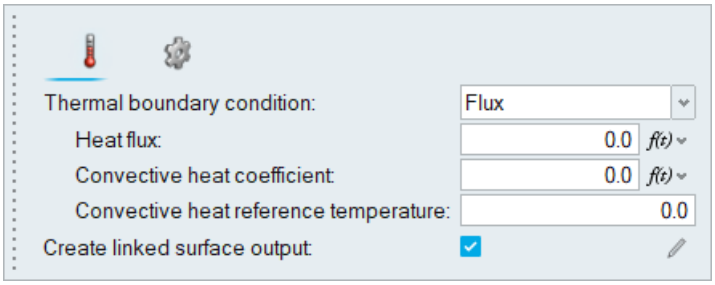


Figure 390:

21. Click ✎. In the new microdialog that appears, set the Nodal Output frequency to `1`.

22. From the Boundaries legend, rename Wall 1 to `Floor` by double-clicking on it.

23. Click ▸ on the guide bar.
The updated Boundaries legend should look similar to the one shown below.

Figure 391:

24. Turn on the visibility of all the surfaces by right-clicking in the modeling window and selecting **Show All** or by simply pressing the **A** key.

25. Save the model.

Set Up Solar Radiation

Set Up the Solar Radiation Parameters

1. From the Radiation ribbon, Solar Radiation tools, click the **Physics** tool.

Figure 392:

2. In the **Solar Radiation Settings** dialog, activate the **Solar radiation equation**.

3. Click 🗁 to load the solar flux input from a file.

4. In the **Open file** dialog, set the filter to **Dat file (.dat)** and select the **SolarLoad.dat** file provided with the input file for this tutorial.

5. Click **Open**.
The plot in the dialog should look like the one shown in the figure below.

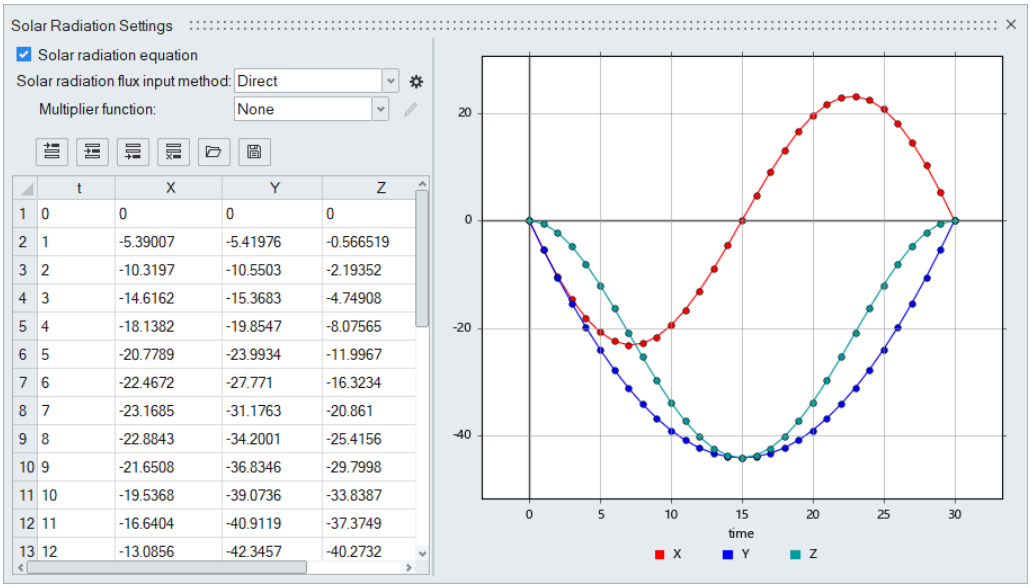


Figure 393:

6. Close the dialog and save the model.

Define the Solar Radiation Models

1. From the Radiation ribbon, Solar Radiation tools, click the **Model** tool.

Figure 394:

2. In the **Solar radiation model library**, click + to add a new solar radiation model.
3. In the Name column, enter `BB out` and set the Side to **Outward**.
4. Similarly, create the other models and enter the values as shown in the figure below.

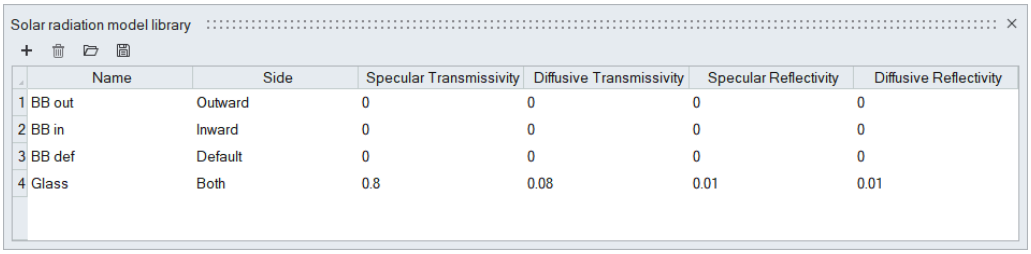


Figure 395:

5. Close the dialog and save the model.

Assign the Solar Radiation Models

1. From the Radiation ribbon, click the **Surface** tool.

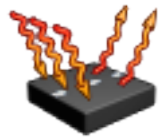


Figure 396:

2. Select the three wall surfaces, the inlet, and the roof surface, as highlighted in the figures below.

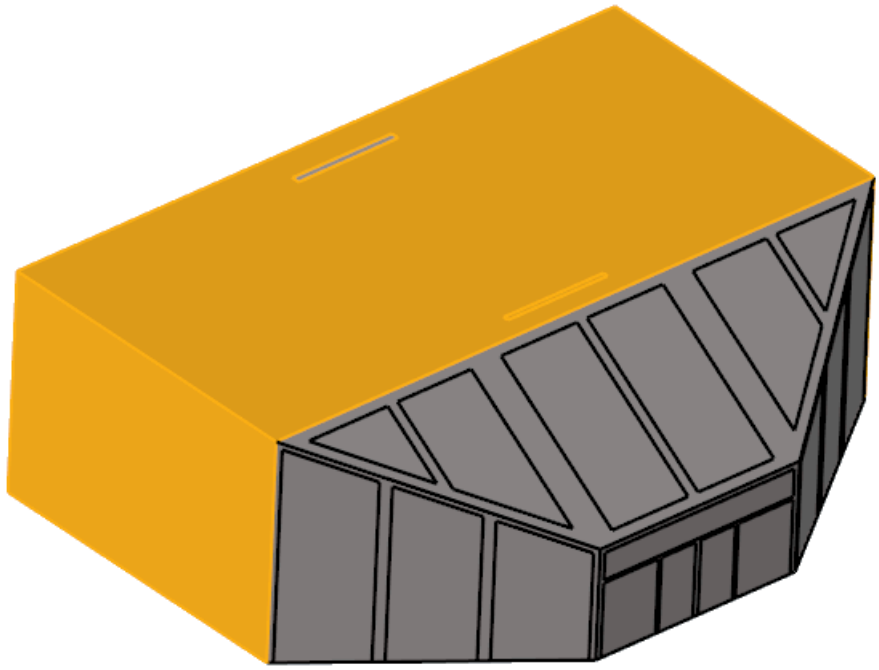


Figure 397: Front-Side of the Model

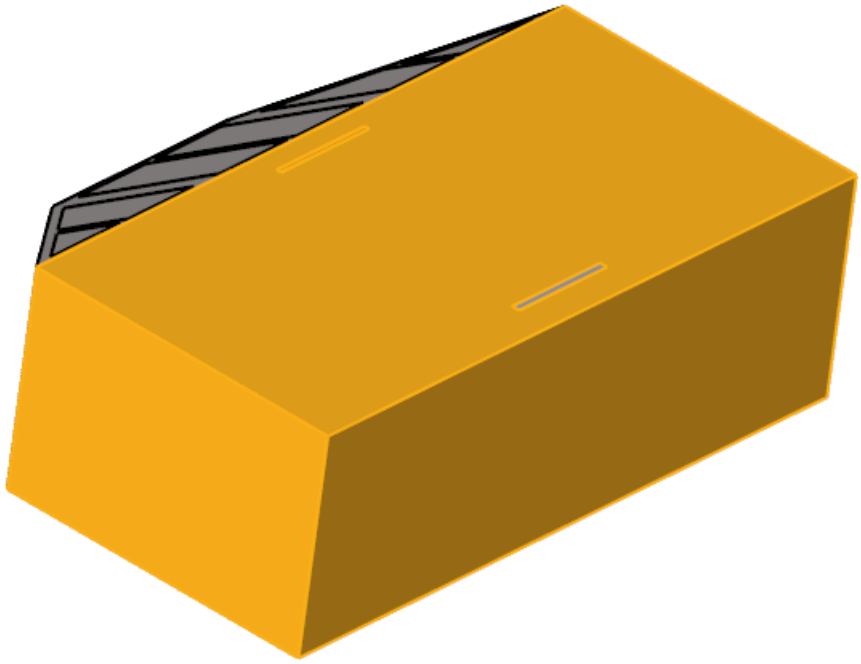


Figure 398: Back-Side of the Model

3. In the microdialog, set the Solar radiation model to **BB out** then click ▸ on the guide bar.
4. Select all the glass surfaces shown in the figure below, assign the **Glass** model to them, then click ▸ on the guide bar.

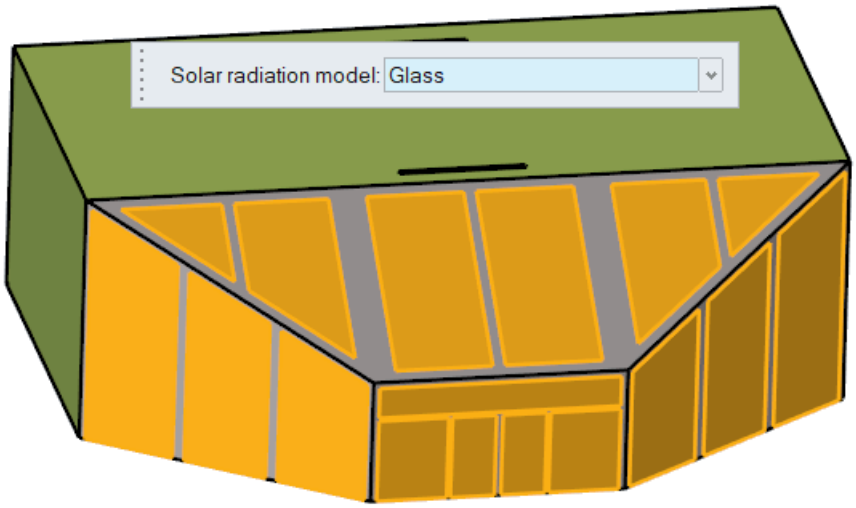


Figure 399:

5. Rotate the model and select the floor surface. In the microdialog, assign the **BB in** model then click ▸ on the guide bar.

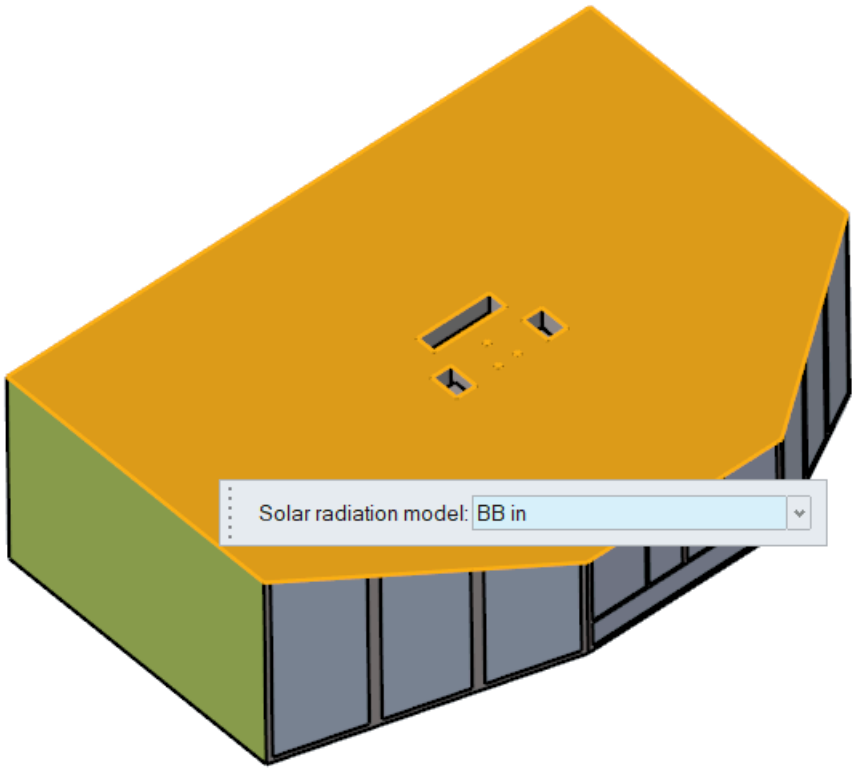


Figure 400:

6. From the Solar Radiation Model legend, right-click on **Unassigned** and select **Isolate**.
7. Select the surfaces of the couch, table, and chairs, assign the **BB def** model to them, then click ▸ on the guide bar.

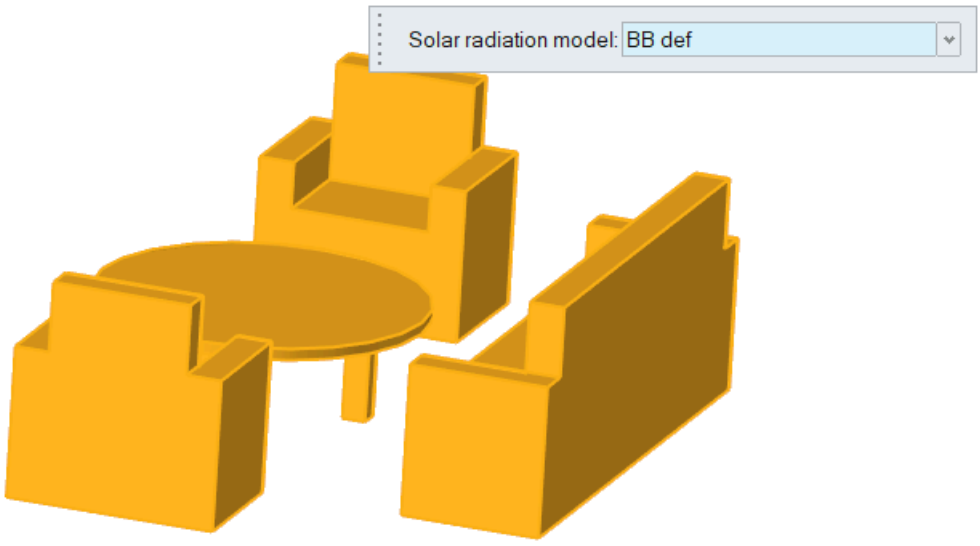


Figure 401:

8. On the guide bar, change the selection entity to **Thin Solids**.
9. Using the window selection method, select the four thin solid surfaces and assign the **BB out** model to them.

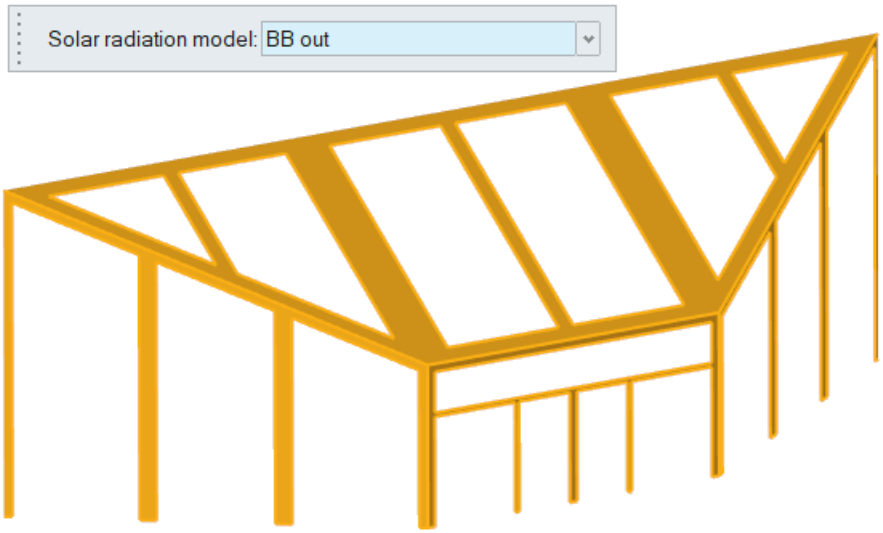


Figure 402:

10. On the guide bar, verify that the Direction is set to **Away from Parent Surface** then click ✓ to execute the changes.
11. Turn on the display of all the surfaces and save the model.

Generate the Mesh

In this step, you will define the mesh controls and then generate the mesh.

Define the Surface Mesh Controls

1. From the Mesh ribbon, click the **Surface** tool.

Figure 403:

2. Using the window selection method, select all the surfaces in the model.
3. In the microdialog, set the Average element size to `0.15` and the Mesh growth rate to `1.0`.

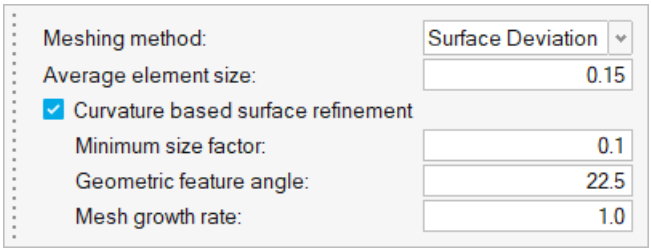


Figure 404:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

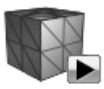


Figure 405:

2. In the **Meshing Operations** dialog, set the Mesh growth rate to `1.1` then click **Mesh** to start the meshing process.
   The **Run Status** dialog opens and the status of the meshing process is shown.
3. Once the mesh is generated, close the **Run Status** dialog and save the model.

> **Note:** Considering the run time of the simulation, a very coarse mesh with no boundary layers is used for this tutorial. Otherwise, a relatively fine mesh with boundary layers to adequately resolve the gradients in the flow and temperature fields should be used.

Compute the Solution

Define the Nodal Output Frequency

1. From the Solution ribbon, click the **Field** tool.

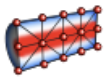


Figure 406:

2. In the **Field Output** dialog, activate the check box for **Write initial conditions**.
3. Set the Time step interval to `1`.

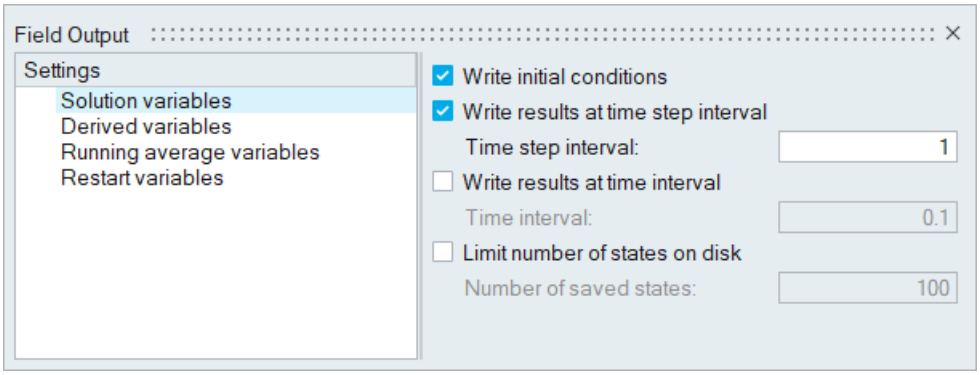


Figure 407:

Define the Nodal Initial Conditions and Compute the Solution

1. From the Solution ribbon, click the **Run** tool.

Figure 408:

2. In the **Launch AcuSolve** dialog, set the Parallel processing option to **Intel MPI**.

3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand the **Default initial conditions** menu.
6. Deactivate the **Pre-compute flow** option.
7. Set the Temperature to `288.15`.
8. Verify that all the values are set as shown in the figure below.

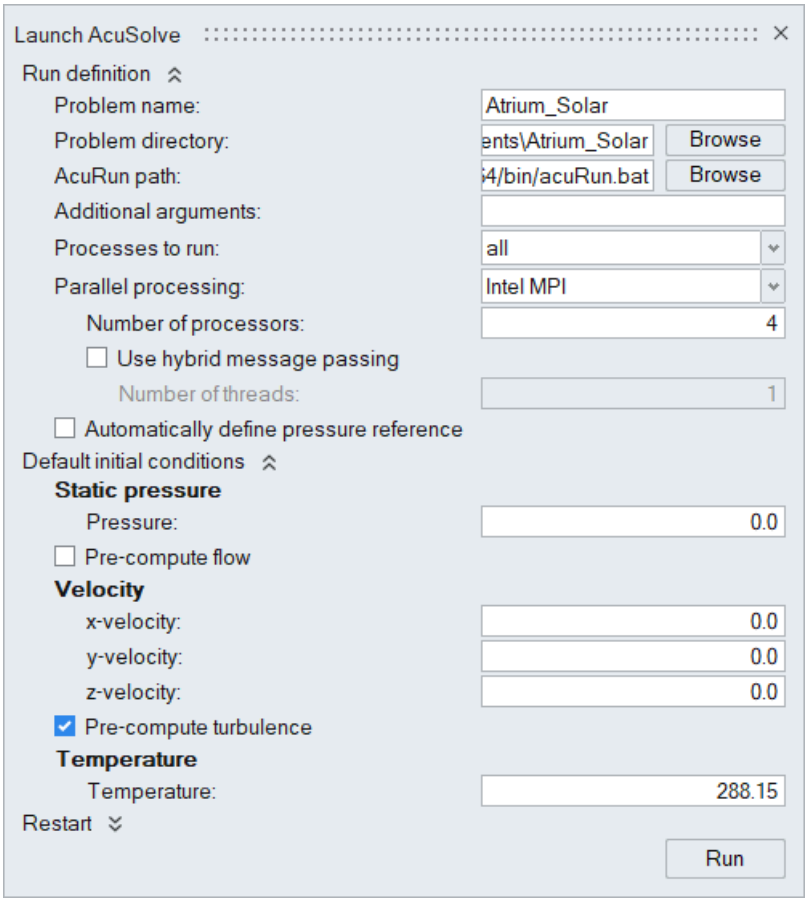


Figure 409:

9. Click **Run**.
   Once the solution process is started, the **Run Status** dialog appears.
10. In the dialog, right-click on the AcuSolve run and select **View log file**.
    Once the run is complete, a summary of the solution process is shown in the log file.

```
acuSolve: ---------- End Time Step ----------
acuSolve: ---------- Offset Values ----------
acuSolve:          Pressure Offset = 0.000000e+000
acuSolve:       Temperature Offset = 0.000000e+000
acuSolve: ----------------------------------------------------------------
acuSolve: Input      CPU/Elapse time= 2.956400e+002 1.329820e+002 Sec
acuSolve: Output     CPU/Elapse time= 8.790700e+001 2.246400e+001 Sec
acuSolve: Exchange   CPU/Elapse time= 5.184000e+000 1.244000e+000 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =         90         90
acuSolve:   Elem.Form CPU/Elapse time= 5.437490e+002 1.368370e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.268465e+003 3.206390e+002 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =         90         90
acuSolve:   Elem.Form CPU/Elapse time= 2.916740e+002 7.311800e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 3.272090e+002 8.233200e+001 Sec
acuSolve: LnSlv. (host)  Elapse time= 4.014690e+002 Sec
acuSolve: Total CPU/Elapse time     = 3.038594e+003 8.246440e+002 Sec
acuSolve: Total Memory Usage        = 5.090000e+002 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
acuTrans:  Variable                 Min         Max         Ave
acuTrans: x_coordinates            -9.144000e+000 9.144000e+000 5.463695e-003
acuTrans: y_coordinates            -1.371600e+001 1.337690e-009 -7.748491e+000
acuTrans: z_coordinates            -9.675669e-007 6.096000e+000 2.861749e+000
acuTrans: x_velocity               -5.116260e+000 6.557373e+000 -9.593589e-003
acuTrans: y_velocity               -6.131809e+000 2.278000e+001 -2.074645e-001
acuTrans: z_velocity               -3.945000e+001 2.835495e+001 6.968431e-002
acuTrans: pressure                 0.000000e+000 5.436678e+002 4.818405e+002
acuTrans: temperature               2.833745e+002 2.953500e+002 2.889658e+002
```

Figure 410:

Post-Process the Results with HM-CFD Post

Create Surface Groups

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Boundary Groups** tool.

Figure 411:

5. Click ⋯ on the guide bar to open Advanced Selection.

6. Set the drop-down to **By Boundaries** then select **Thin Solid Wall** and **Floor**.

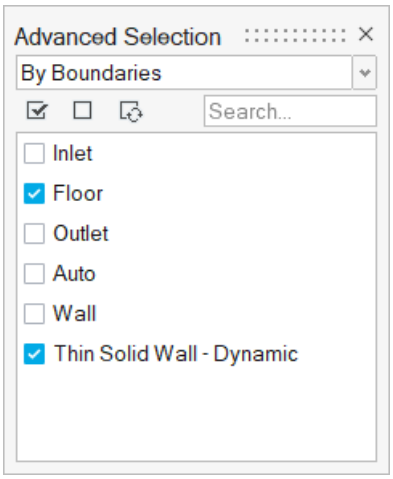


Figure 412:

7. Exit the dialog then click ✓ on the guide bar.
   The two boundaries are grouped together under the User Defined surface group in the Post Browser.
8. Isolate the new surface group and rename it to `ThinSolid_Floor`.

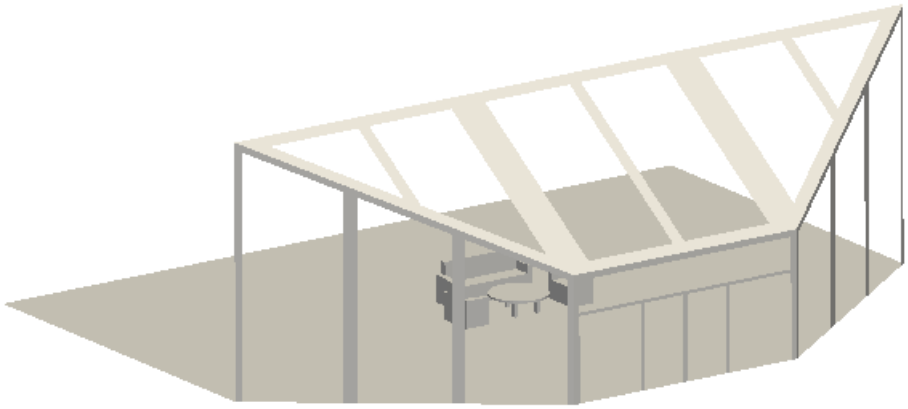


Figure 413:

Plot the Temperature Contour

1. In the Post Browser, right-click on the **ThinSolid_Floor** surface group and select **Edit**.
2. In the display properties microdialog, set the display to **temperature** and activate the **Legend** toggle.
3. Set the legend bounds to `287` and `292`.
4. Click ≡ ˅ and set the Colormap Name to **Rainbow Uniform**.

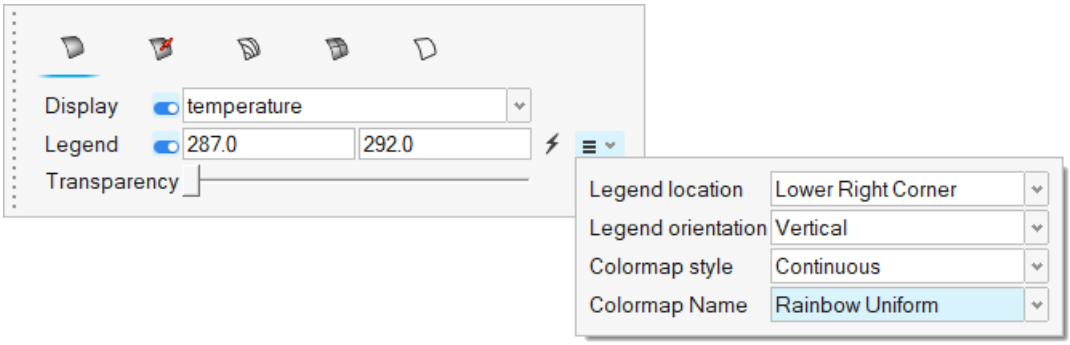


Figure 414:

5. Click ✓ on the guide bar.
6. Drag the animation slider at the bottom of the modeling window to the 31st frame.

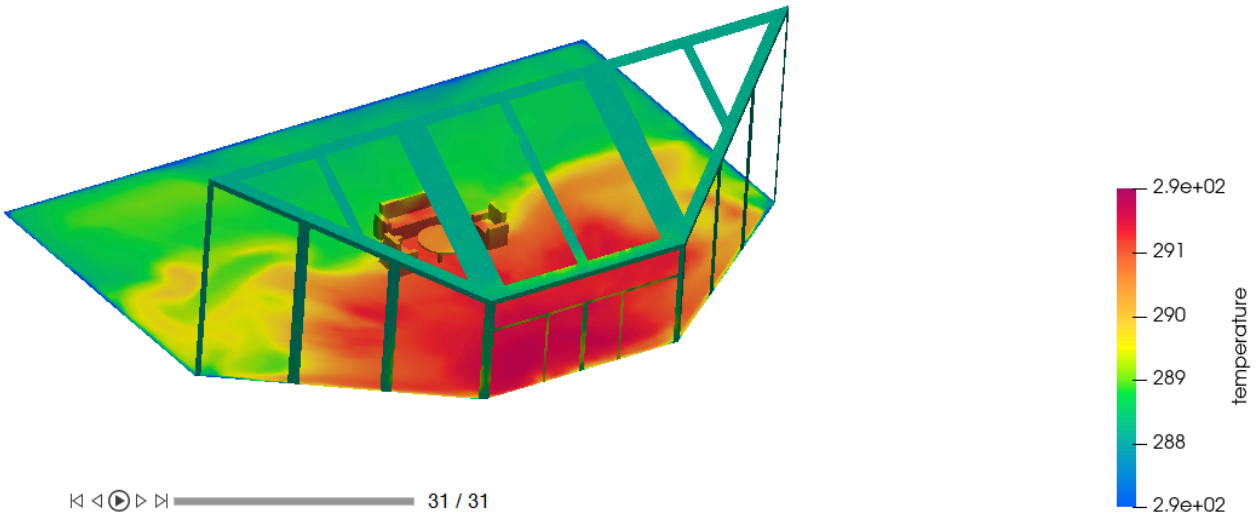


Figure 415:

Summary

In this tutorial, you learned how to set up and solve a CFD analysis involving solar radiation. You started by importing a geometry model into HyperMesh CFD and setting up the simulation parameters and boundary conditions. Once you computed the solution, you post-processed the results using the Post ribbon.

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/06kz8feyba

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up, solving, and viewing results for a steady state simulation of radiation heat transfer between concentric spheres using the P-1 Radiation model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3202_P1Rad.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 416. In this problem, a P1 radiation model is used to simulate the heat transfer due to radiation between concentric spheres. The inside surface of the inner and the outside surface of the outer sphere are both held at constant temperature while the gap between them radiates the heat from one sphere to the other.

The problem consists of a fluid region with arbitrary material properties between two concentric spheres with surfaces held at fixed temperature, as shown in the following figure, which is not drawn to scale. The radius of the outer sphere is 0.04 m and the radius of the inner sphere is 0.01 m. The inner surface of the inner sphere is defined to have a constant wall temperature at 300.0 K (26.85 ºC). The outer surface of the outer sphere is defined to have a constant wall temperature at 1300.0 K (1026.85 ºC). The fluid within the spheres is defined as a non-conducting material, allowing heat to transfer via radiation only.

The problem is solved as a steady state case to allow the heat transfer in the solid and fluid regions to reach an equilibrium.

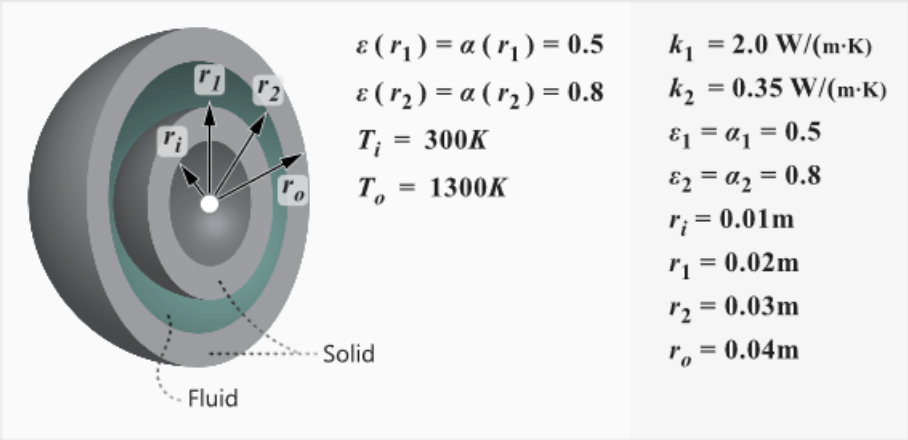


Figure 416:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 417:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3202_P1Rad.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `P1_radiation` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `P1_radiation` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate

icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 418:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

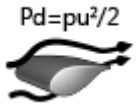
   

   Figure 419:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Set Time marching to **Steady**.
   b) Set the Turbulence model to **Laminar**.
   c) Activate the **Heat transfer** checkbox.

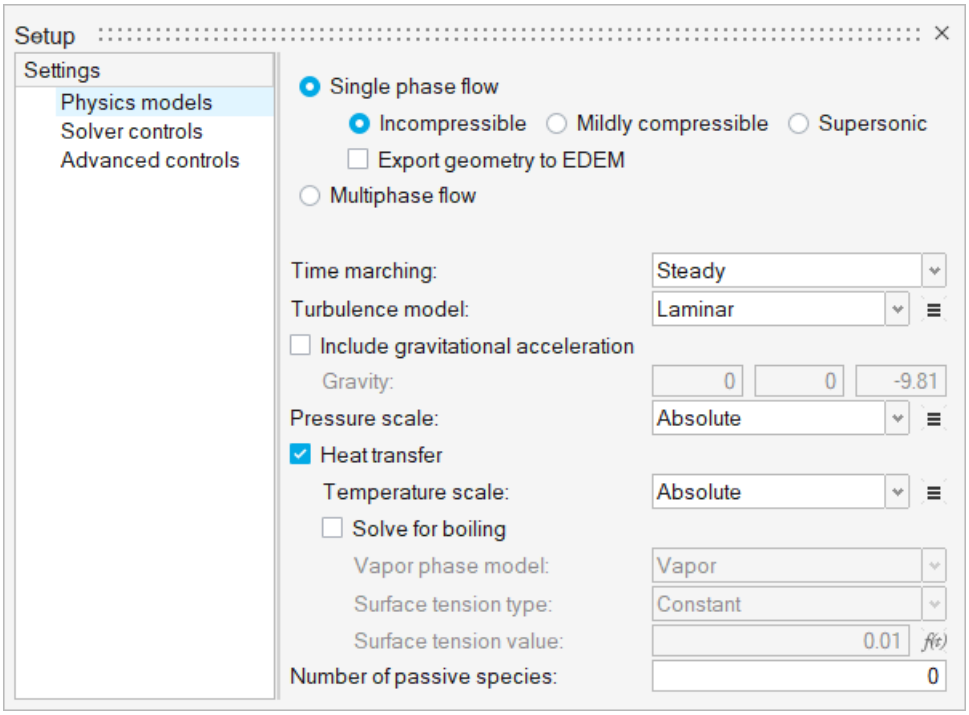


Figure 420:

3. Click the **Solver controls** setting.
4. Deactivate the **Flow** checkbox.

   In this tutorial, you will only be solving for the temperature field.

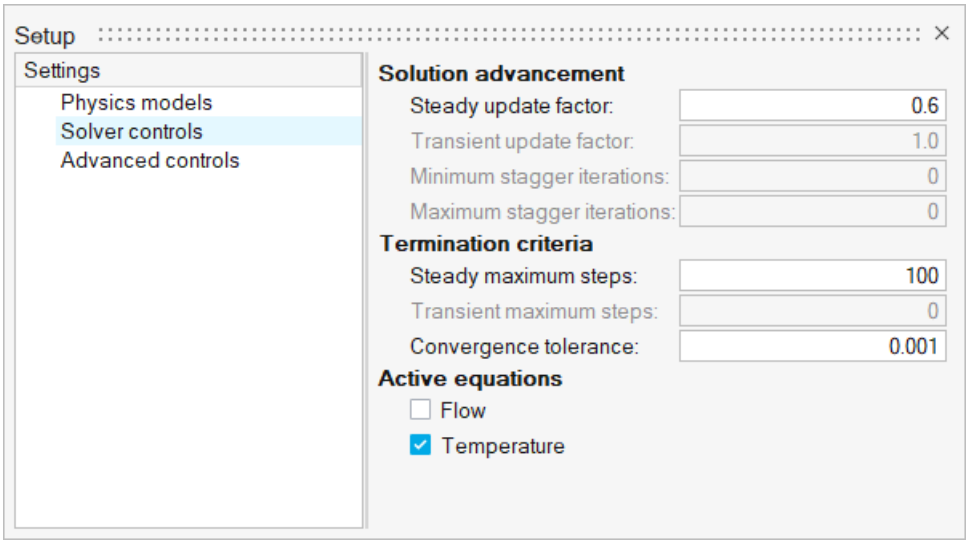


Figure 421:

5. Close the dialog and save the model.

Create Material Models

1. From the Flow ribbon, click the **Material Library** tool.

Figure 422:

The **Material Library** dialog opens.

2. Under Settings, click **Fluid** then click the **My Materials** tab.
3. Click + to create a new material.
4. In the new dialog, rename the material to `Radiating` by clicking the name in the top-left corner.
5. Enter the following values for each of the material property tabs then close the dialog.
   - Density – `1000` kg/m^3
   - Specific Heat – `10000` J/kg-K
   - Viscosity – `1e-5` kg/m-sec
   - Conductivity – `1e-6` W/m-K

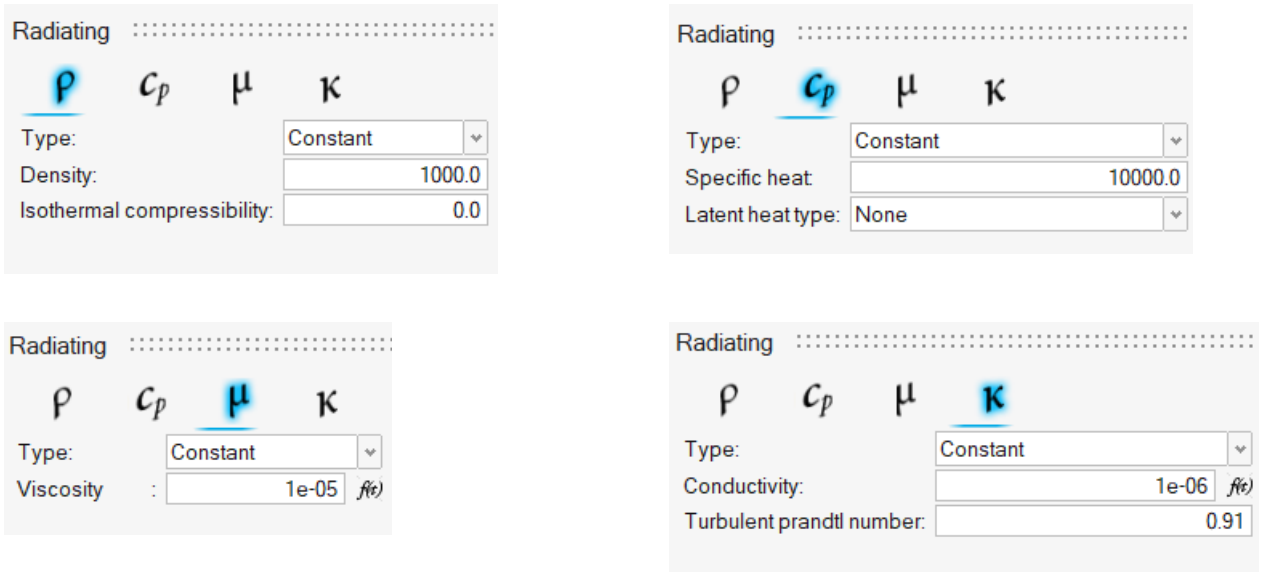


Figure 423:

6. Click the **Solid** setting.
7. Repeat the above steps to create two material models (`Inner` and `Outer`) using the following parameters:

Inner

- Density – 1000 kg/m^3
- Specific Heat – 10000 J/kg-K
- Conductivity – 2 W/m-K

Outer

- Density – 1000 kg/m^3
- Specific Heat – 10000 J/kg-K
- Conductivity – 0.35 W/m-K

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 424:

2. Select the outer solid and assign the **Outer** material model from the microdialog.

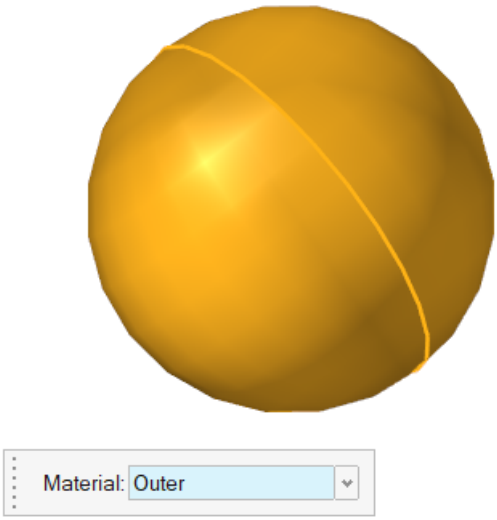


Figure 425:

3. On the guide bar, click ▸ to execute the command and remain in the tool.
4. In the Materials legend, right-click on **Air** and select **Isolate**.
5. Select the outer solid and assign the **Radiating** material model to it.
6. On the guide bar, click ▸.

7. In the Materials legend, right-click on **Air** again and select **Isolate**.
8. Select the remaining solid and assign the **Inner** material model to it.
9. On the guide bar, click ✓ to execute the command and exit the tool.
10. Save the model.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **No Slip** tool.

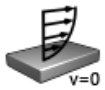


Figure 426:

2. Select the two faces of the sphere then right-click and select **Hide**.
3. Select the two faces remaining in the modeling window. In the microdialog, assign a **Temperature** boundary condition at `300` K.

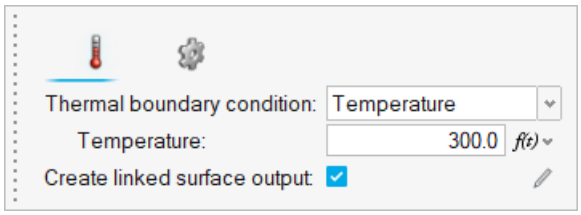


Figure 427:

4. In the Boundaries legend, double-click on **Wall** and rename it to `Inner`.
5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. Press the **A** key to turn on the display of all surfaces.
7. Select the two outer faces. In the microdialog, assign a **Temperature** boundary condition at `1300` K.
8. In the Boundaries legend, double-click on **Wall** and rename it to `Outer`.
9. On the guide bar, click ✓ to execute the command and exit the tool.
10. Save the model.

Set Up Radiation

Select the Radiation Model

1. From the Radiation ribbon, Thermal Radiation tools, click the **Physics** tool.

Figure 428:

The **Radiation Settings** dialog opens.

2. Activate the **Thermal radiation** checkbox and set the Radiation model to **P1 Model**.

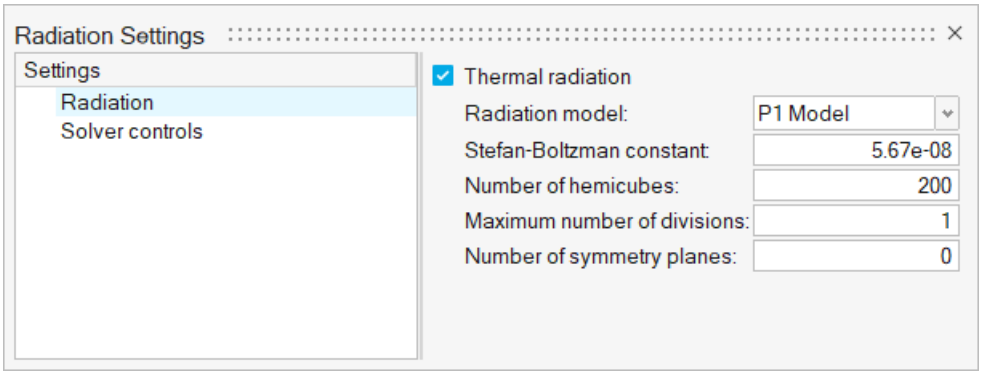


Figure 429:

Define the Radiation Material Properties

1. Click the **Surface Finish Library** tool.

Figure 430:

The **Surface finish library** opens.

2. Click **+**.
3. In the table, double-click on **Surface Finish**, rename it to `Inner`, and the Emissivity to `0.5`.
4. Similarly, create another Emissivity model named `Outer` with an Emissivity of `0.8`.

Surface finish library

	Name	Emissivity	Emissivity Multiplier
1	Black Body	1	None
2	Inner	0.5	None
3	Outer	0.8	None

Figure 431:

5. From the Participating Media tools, click the **Model** tool.

Figure 432:

The **Participating media model library** opens.

6. Click + and create a model with the following properties.

Participating media model library

	Name	Absorption coefficient	Scattering coefficient	Phase coefficient	Refractive index
1	Air	0.001	0	0	1

Figure 433:

Assign the Participating Media Model

1. From the Participating Media tools, click the **Assign** tool.

Figure 434:

2. In the modeling window, right-click and select **Select** > **Advanced** > **By Material** > **Radiating**

3. In the microdialog, verify that the Air model is selected.
4. Click ▸ on the guide bar.
5. In the Participating Media legend, right-click on **Air** and select **Isolate**.

Assign the Emissivity Model

1. From the Thermal Radiation tools, click the **Surface Finish** tool.

Figure 435:

2. Select the two outer surfaces.
3. In the microdialog, select the **Outer** surface finish model.
4. Click ▸ on the guide bar.
5. In the Participating Media legend, right-click on **Black Body** and select **Isolate**.
6. Select the two visible surfaces and assign the **Inner** surface finish model.
7. Click ✓ on the guide bar.
8. Save the model.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

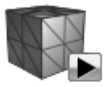


Figure 436:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.0025 and the Mesh growth rate is 1.0.
3. Accept all other default parameters.

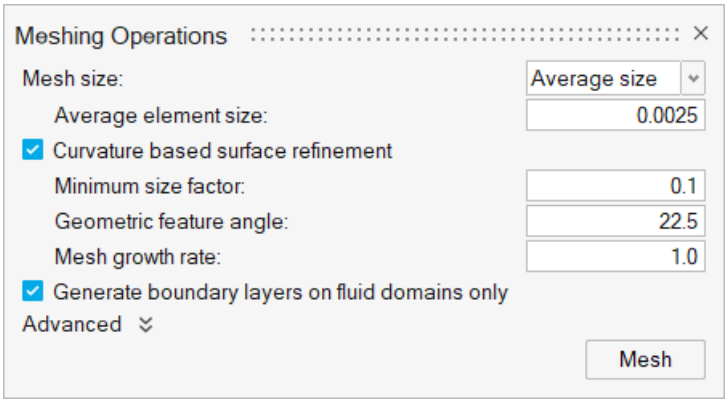


Figure 437:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 438:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Leave the remaining options as default and click **Run** to launch AcuSolve.

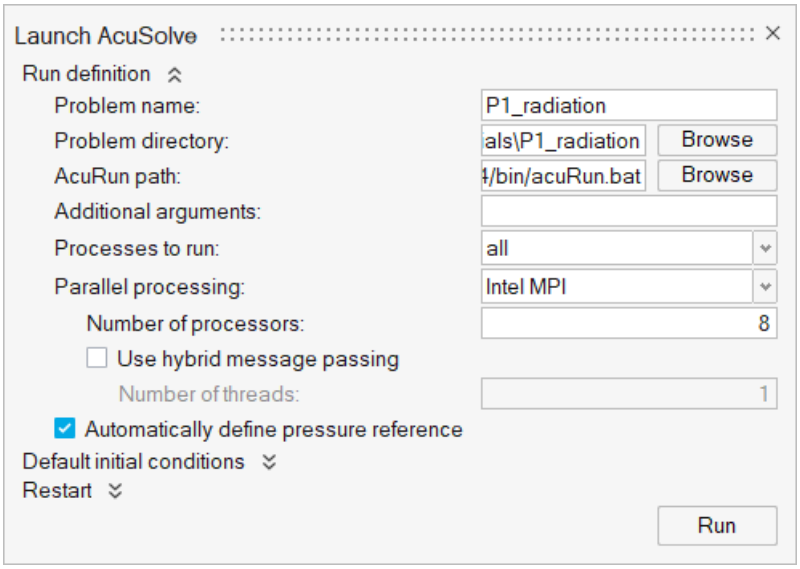


Figure 439:

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. In the browser, click the icon beside Flow Boundaries to turn off the display of all the surfaces.

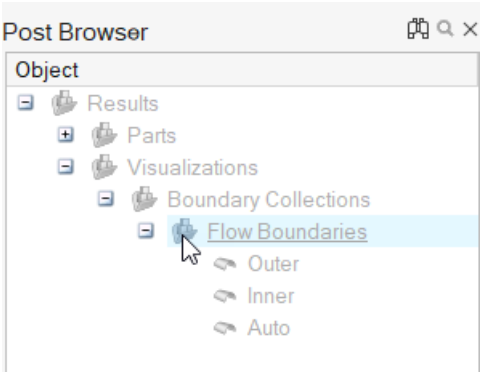


Figure 440:

5. Click the **Slice Planes** tool.

Figure 441:

6. Select the x-y plane in the modeling window.
7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, set the display to **Temperature** and activate the **Legend** toggle.
9. Click and set the Colormap Name to **Rainbow Uniform**.

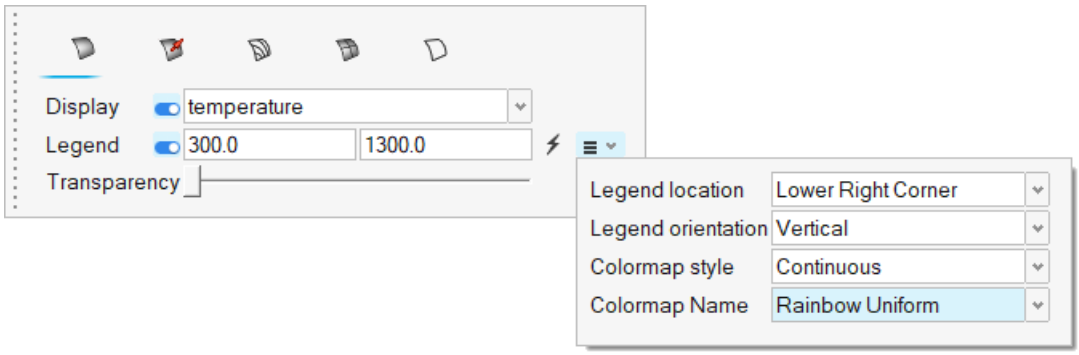


Figure 442:

10. Click on the guide bar then press **F** to fit the section cut to the screen.

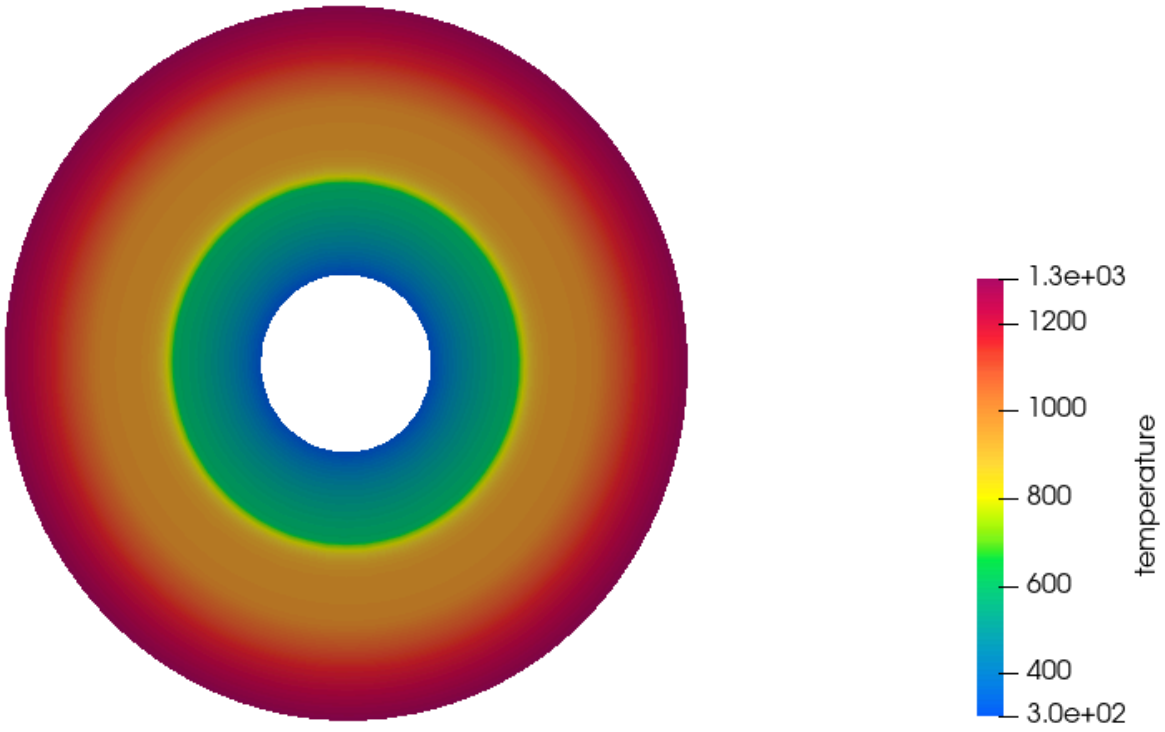


Figure 443:

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer simulation using the P1 radiation model. You started by importing the HyperMesh CFD input database and defining the flow and radiation setup. Then, you generated the mesh and submitted the AcuSolve simulation. Once the solution was computed, created a contour plot of temperature distribution on a section cut using HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/dyv3m05iok

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up, solving, and viewing results for a steady state simulation of radiation heat transfer between concentric spheres using the Discrete Ordinates (DO) Radiation model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3203_DO_Rad.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 444. In this problem, a DO radiation model is used to simulate the heat transfer due to radiation between concentric spheres. The inside surface of the inner and the outside surface of the outer sphere are both held at constant temperature while the gap between them radiates the heat from one sphere to the other.

The problem consists of a fluid region with arbitrary material properties between two concentric spheres with surfaces held at fixed temperature, as shown in the following figure, which is not drawn to scale. The radius of the outer sphere is 0.04 m and the radius of the inner sphere is 0.01 m. The inner surface of the inner sphere is defined to have a constant wall temperature at 300.0 K (26.85 ºC). The outer surface of the outer sphere is defined to have a constant wall temperature at 1300.0 K (1026.85 ºC). The fluid within the spheres is defined as a non-conducting material, allowing heat to transfer via radiation only.

The problem is solved as a steady state case to allow the heat transfer in the solid and fluid regions to reach an equilibrium.

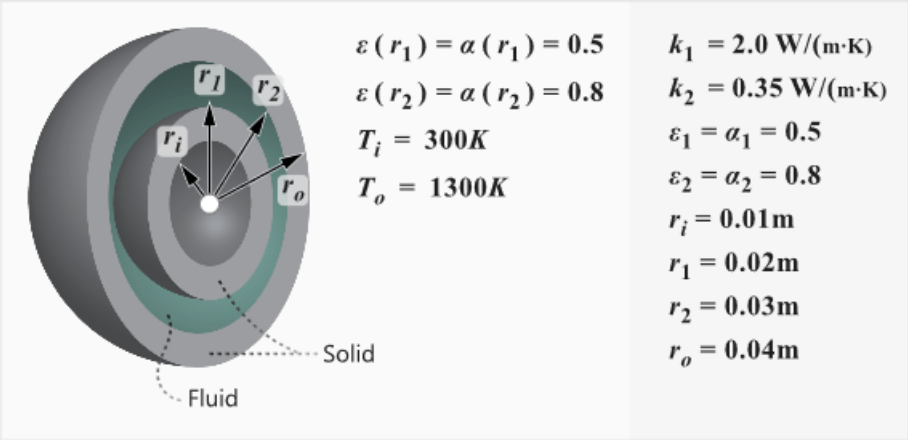


Figure 444:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 445:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3203_DO_Rad.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `DO_radiation` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `DO_radiation` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate

icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 446:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

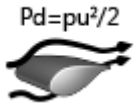
   

   Figure 447:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Set Time marching to **Steady**.
   b) Set the Turbulence model to **Laminar**.
   c) Activate the **Heat transfer** checkbox.

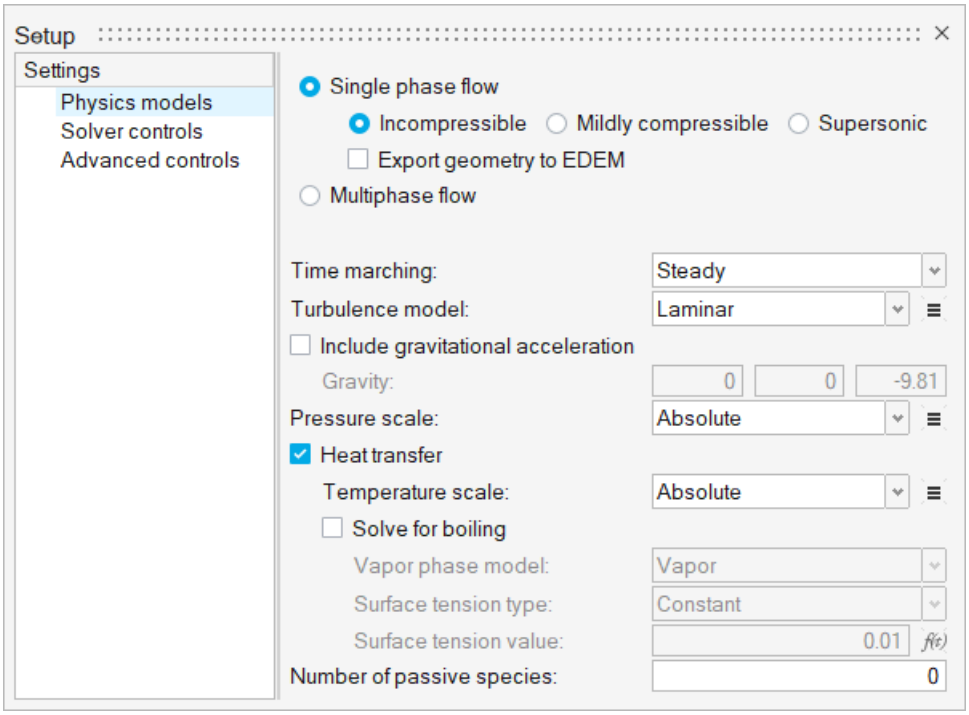


Figure 448:

3. Click the **Solver controls** setting.
4. Deactivate the **Flow** checkbox.

   In this tutorial, you will only be solving for the temperature field.

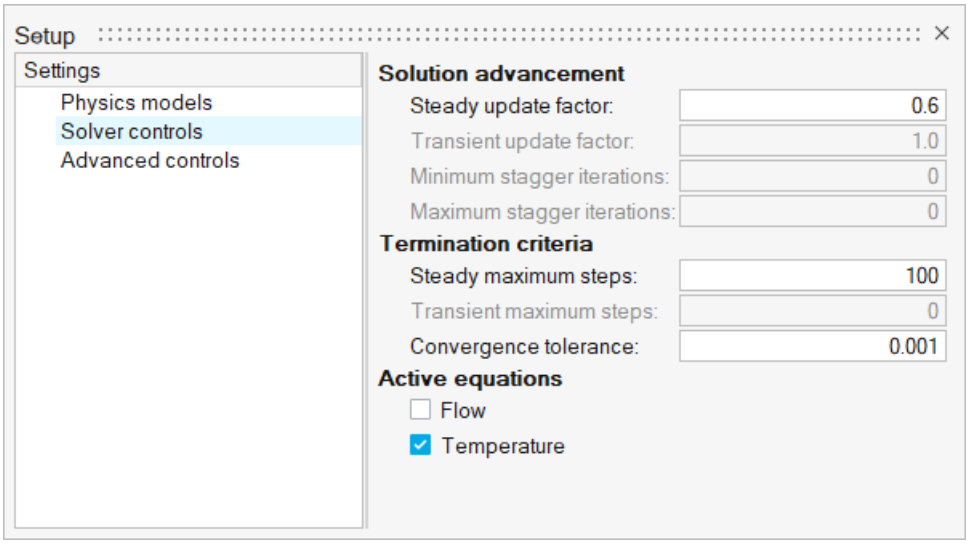


Figure 449:

5. Close the dialog and save the model.

Create Material Models

1. From the Flow ribbon, click the **Material Library** tool.

Figure 450:

The **Material Library** dialog opens.

2. Under Settings, click **Fluid** then click the **My Materials** tab.
3. Click + to create a new material.
4. In the new dialog, rename the material to `Radiating` by clicking the name in the top-left corner.
5. Enter the following values for each of the material property tabs then close the dialog.
   - Density – `1000` kg/m^3
   - Specific Heat – `10000` J/kg-K
   - Viscosity – `1e-5` kg/m-sec
   - Conductivity – `1e-6` W/m-K

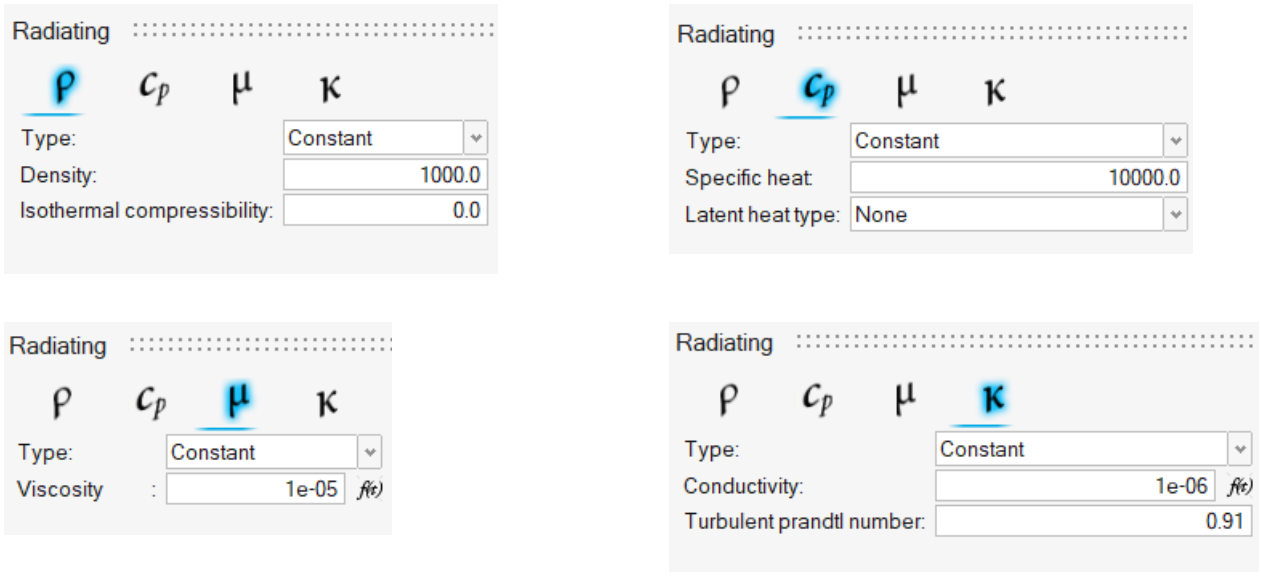


Figure 451:

6. Click the **Solid** setting.
7. Repeat the above steps to create two material models (`Inner` and `Outer`) using the following parameters:

Inner

- Density – 1000 kg/m^3
- Specific Heat – 10000 J/kg-K
- Conductivity – 2 W/m-K

Outer

- Density – 1000 kg/m^3
- Specific Heat – 10000 J/kg-K
- Conductivity – 0.35 W/m-K

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 452:

2. Select the outer solid and assign the **Outer** material model from the microdialog.

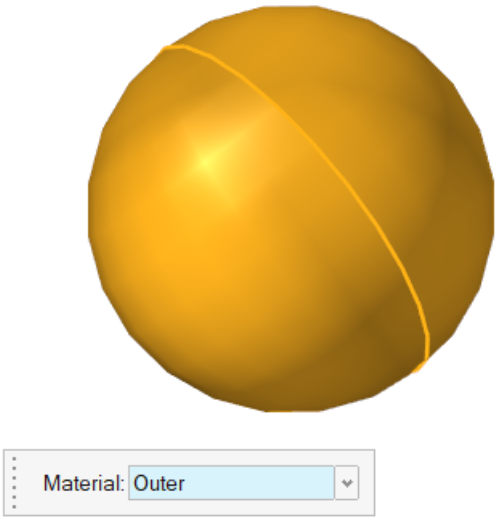


Figure 453:

3. On the guide bar, click ▸ to execute the command and remain in the tool.
4. In the Materials legend, right-click on **Air** and select **Isolate**.
5. Select the outer solid and assign the **Radiating** material model to it.
6. On the guide bar, click ▸.

7. In the Materials legend, right-click on **Air** again and select **Isolate**.
8. Select the remaining solid and assign the **Inner** material model to it.
9. On the guide bar, click ✓ to execute the command and exit the tool.
10. Save the model.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **No Slip** tool.

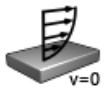


Figure 454:

2. Select the two faces of the sphere then right-click and select **Hide**.
3. Select the two faces remaining in the modeling window. In the microdialog, assign a **Temperature** boundary condition at `300` K.

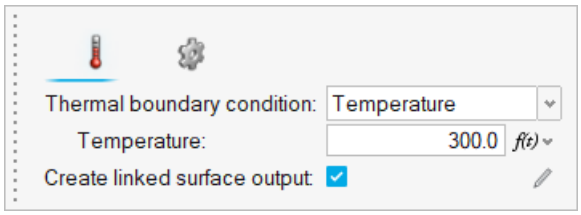


Figure 455:

4. In the Boundaries legend, double-click on **Wall** and rename it to `Inner`.
5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. Press the **A** key to turn on the display of all surfaces.
7. Select the two outer faces. In the microdialog, assign a **Temperature** boundary condition at `1300` K.
8. In the Boundaries legend, double-click on **Wall** and rename it to `Outer`.
9. On the guide bar, click ✓ to execute the command and exit the tool.
10. Save the model.

Set Up Radiation

Select the Radiation Model

1. From the Radiation ribbon, Thermal Radiation tools, click the **Physics** tool.

Figure 456:

The **Radiation Settings** dialog opens.

2. Activate the **Thermal radiation** checkbox and set the Radiation model to **Discrete Ordinate**.

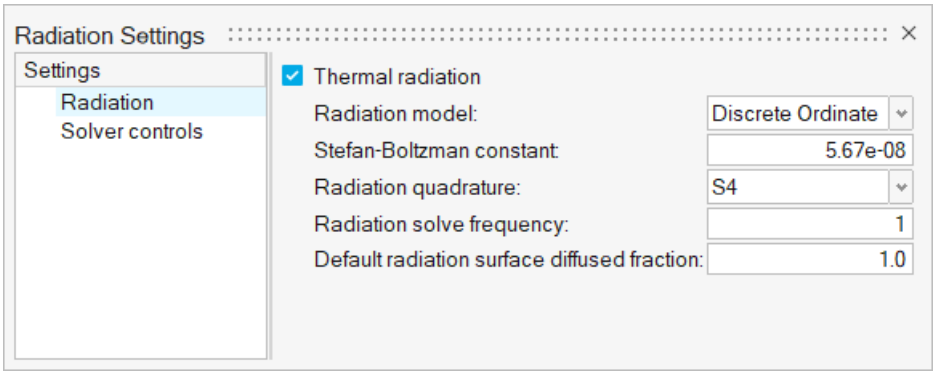


Figure 457:

3. Set the Radiation quadrature to **S4**.
4. Verify that the Default radiation surface diffused fraction is set to 1.

Define the Radiation Material Properties

1. Click the **Surface Finish Library** tool.

Figure 458:

The **Surface finish library** opens.

2. Click **+**.
3. In the table, double-click on **Surface Finish**, rename it to `Inner`, and the Emissivity to `0.5`.
4. Similarly, create another Emissivity model named `Outer` with an Emissivity of `0.8`.

Surface finish library

	Name	Emissivity	Emissivity Multiplier
1	Black Body	1	None
2	Inner	0.5	None
3	Outer	0.8	None

Figure 459:

5. From the Participating Media tools, click the **Model** tool.

Figure 460:

The **Participating media model library** opens.

6. Click + and create a model with the following properties.

Participating media model library

	Name	Absorption coefficient	Scattering coefficient	Phase coefficient	Refractive index
1	Air	0.001	0	0	1

Figure 461:

Assign the Participating Media Model

1. From the Participating Media tools, click the **Assign** tool.

Figure 462:

2. In the modeling window, right-click and select **Select** > **Advanced** > **By Material** > **Radiating**

3. In the microdialog, verify that the Air model is selected.
4. Click ▸ on the guide bar.
5. In the Participating Media legend, right-click on **Air** and select **Isolate**.

Assign the Emissivity Model

1. From the Thermal Radiation tools, click the **Surface Finish** tool.

Figure 463:

2. Select the two outer surfaces.
3. In the microdialog, select the **Outer** surface finish model.
4. Click ▸ on the guide bar.
5. In the Participating Media legend, right-click on **Black Body** and select **Isolate**.
6. Select the two visible surfaces and assign the **Inner** surface finish model.
7. Click ✓ on the guide bar.
8. Save the model.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

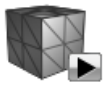


Figure 464:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.0025 and the Mesh growth rate is 1.0.
3. Accept all other default parameters.

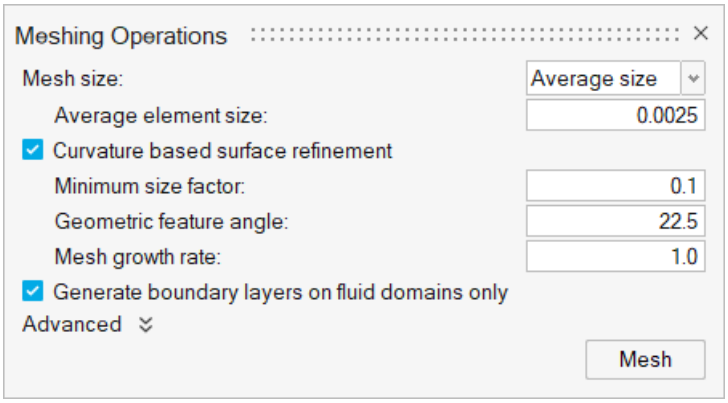


Figure 465:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 466:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial condition** and set the Temperature to `300` K.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

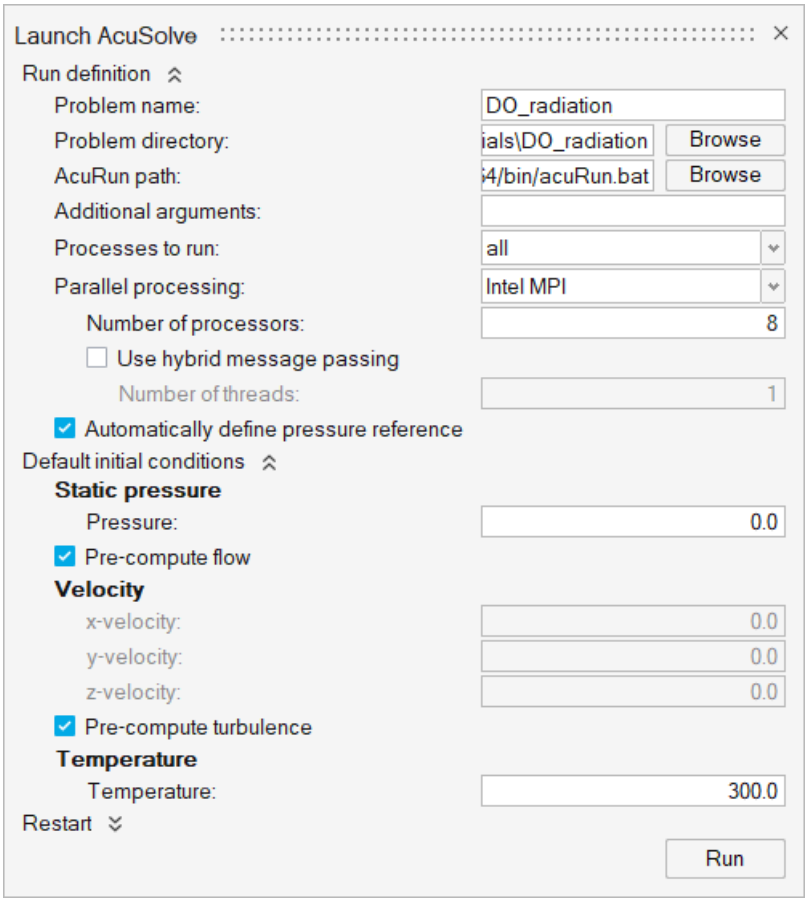


Figure 467:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. In the browser, click the icon beside Flow Boundaries to turn off the display of all the surfaces.

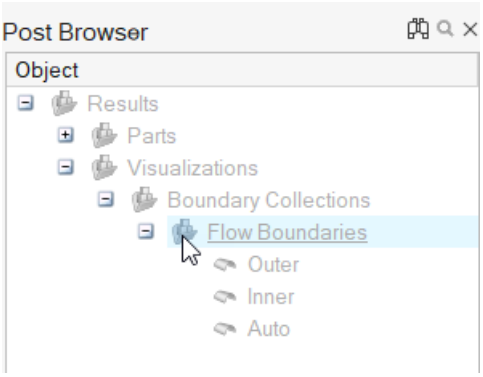


Figure 468:

5. Click the **Slice Planes** tool.

Figure 469:

6. Select the x-y plane in the modeling window.
7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, set the display to **Temperature** and activate the **Legend** toggle.
9. Click and set the Colormap Name to **Rainbow Uniform**.

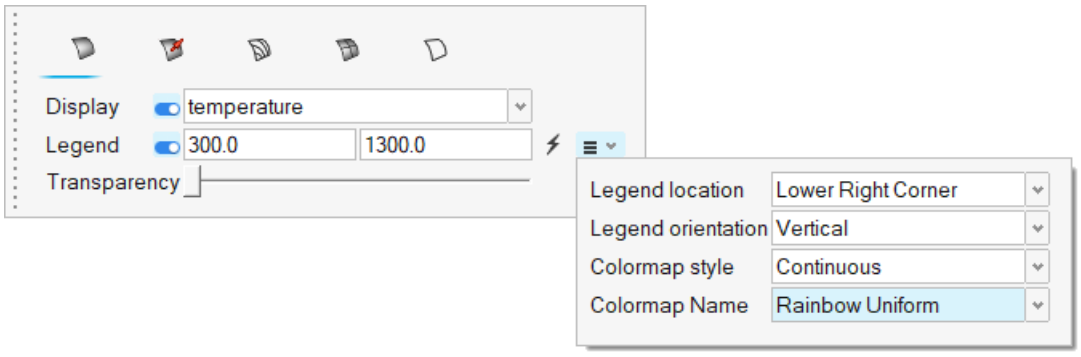


Figure 470:

10. Click on the guide bar then press **F** to fit the section cut to the screen.

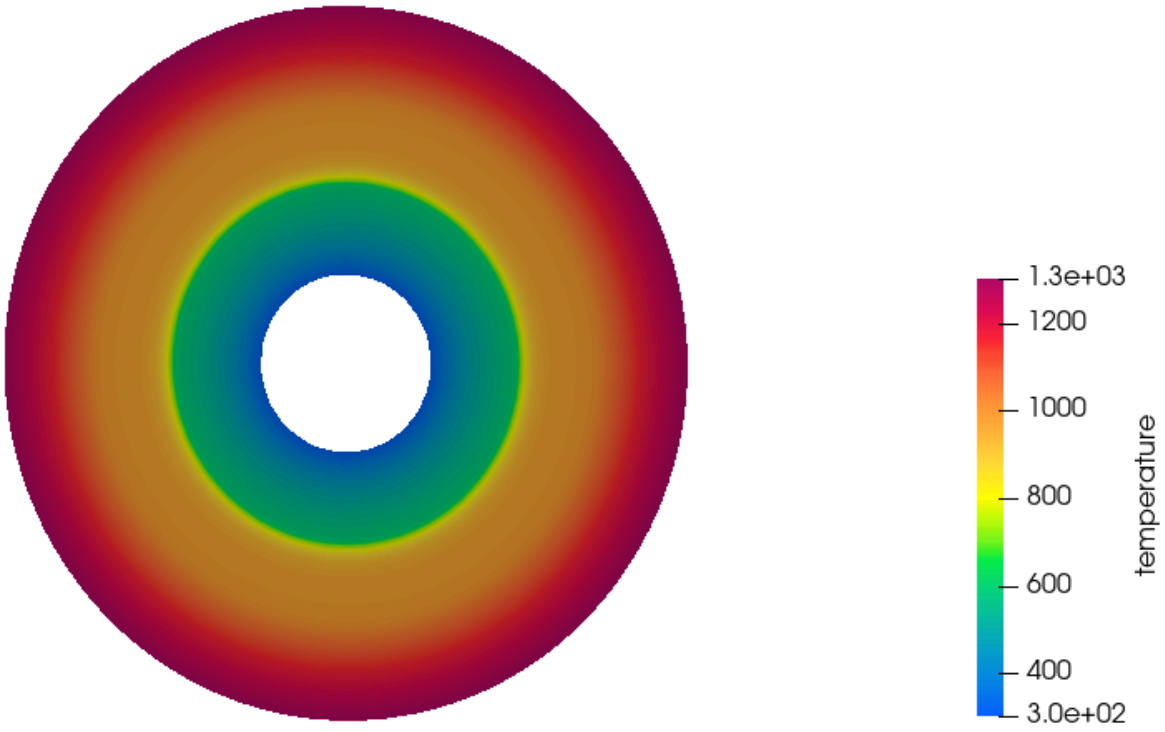


Figure 471:

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer simulation using the Discrete Ordinates radiation model. You started by importing the HyperMesh CFD input database and then defined the flow and radiation setup. Then, you generated the mesh and submitted the AcuSolve simulation. Once the solution was computed, you created a contour plot of temperature distribution on a section cut using HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/6kjs363f29

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial introduces you to setting up a radiation heat transfer problem using the Discrete Ordinate radiation model in HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3204_headlamp.x_t`

Problem Description

The problem to be solved is shown schematically in Figure 472 and Figure 473. It consists of a simple headlamp with a housing, lens, and a bulb. The inner cavity of the bulb is filled with air and the wattage of the bulb is 1W, which is modeled as a volumetric heat source. The Boussinesq density model is used for the air to consider the natural convection effects in the fluid volume. The heat generated in the bulb is transferred by three means: conduction from the bulb to the housing, natural convection in the air volume, and radiation from the bulb to the air and lens volume. The outer surfaces of the headlamp are assumed to be at a constant temperature of 300 K.

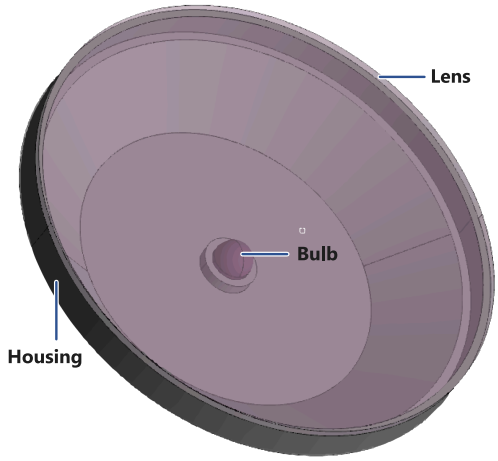


Figure 472:

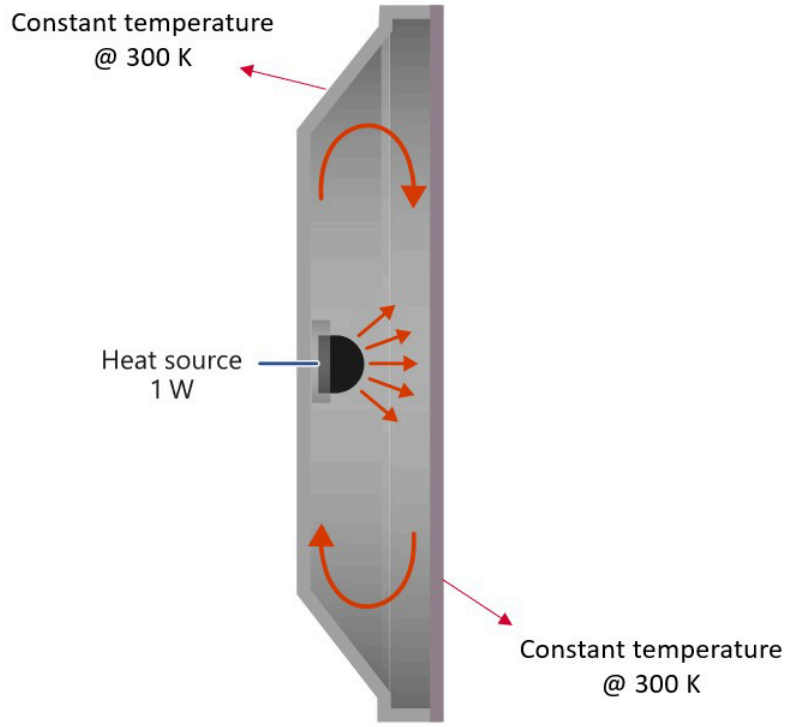


Figure 473:

The air and the lens are modeled as participating media for radiation heat transfer. For the purpose of this tutorial, the absorption coefficient of air is considered to be zero. The following radiation material properties are used for the air and the lens volumes.

Table 1:

	Absorption Coefficient	Refractive Index
Air	0	1.0
Lens	900	1.57

For simulations involving participating media, the following radiation surface types must be defined at the boundaries of each participating medium.

Table 2:

Surface Type	Radiation Surface Type
Participating medium - Participating medium Interface	Radiation Interface - Internal
Participating medium - (Non-Participating) medium Interface	Wall
External boundaries of Participating medium	Radiation Interface - External or Wall

The discrete ordinate radiation model implementation in AcuSolve allows for modeling specular, diffuse, and partially specular interfaces. This can be done by specifying the appropriate value of diffused fraction while defining the radiation surfaces. For the diffused fraction, a value of 1 means the surface is purely diffuse, a value of 0 means it is purely specular. Any value between 0 and 1 means the surface is partially specular. In this tutorial, since both the air and lens are modeled as participating media, the lens-air interface and the lens-outer surfaces will be modeled as internal and external radiation interface respectively. The internal interface will be defined as a specular interface whereas the external interface will be defined as a diffuse interface.

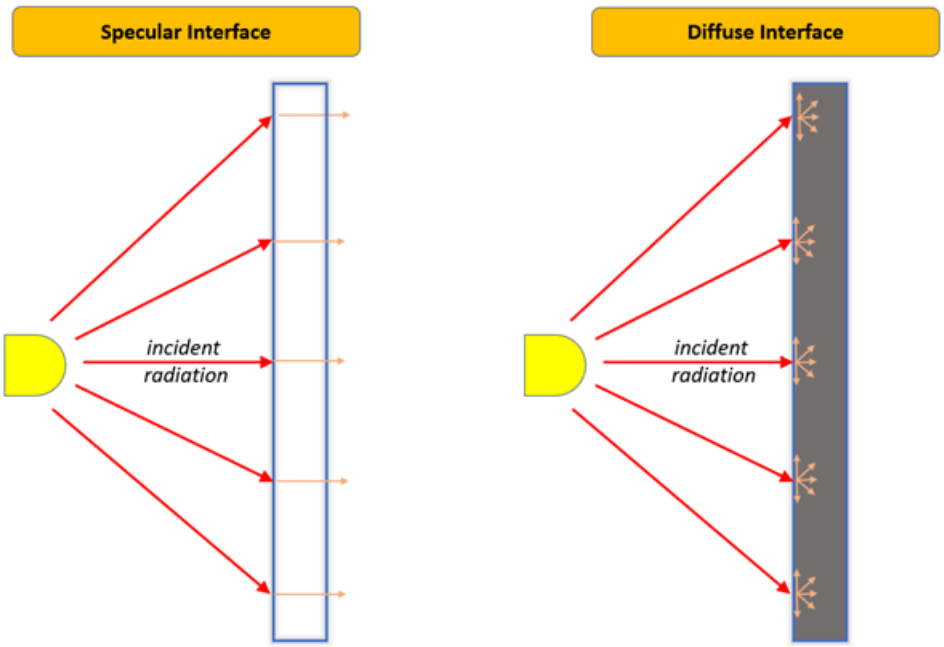


Figure 474:

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.
   - From the Home tools, Files tool group, click the **Save As** tool.

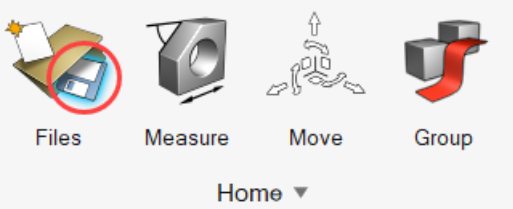


Figure 475:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `Headlamp_DO` as the name for the database then click **Save**.

   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T3204_headlamp.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

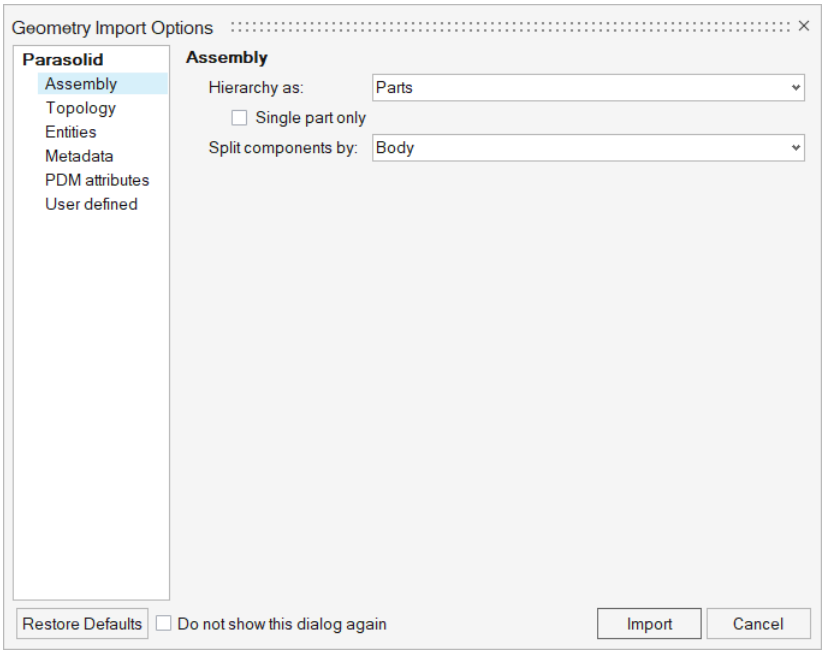


Figure 476:

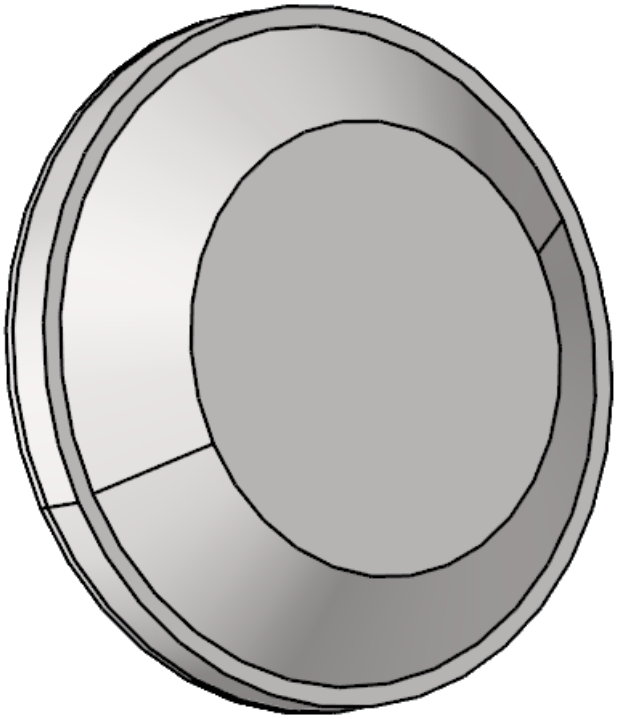


Figure 477:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 478:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The surface and solid errors display in the list below the tool.

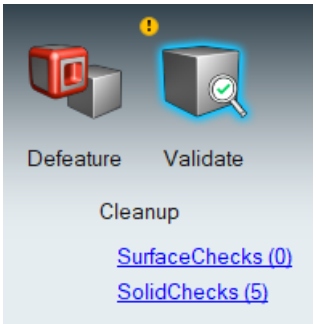


Figure 479:

2. Click **SolidChecks**.
   The Solid Repair tool opens, which you can use to fix the geometric errors in the model.

   From the SolidChecks legend, you can see the model's solids have five intersections.

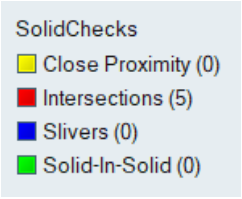


Figure 480:

3. Click **Intersections**.
   A guide bar used to fix intersecting solids displays.
4. Optional: Click → and ← to review each error.
5. Activate the **Keep common interface** option then click **Combine All**.
   The SolidChecks legend should now display zero for all errors.
6. Click the **Validate** tool once again.
   Observe that a blue check mark now appears on the top-left corner of the tool icon. This indicates that no issues are detected and you are ready to continue.

Figure 481:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

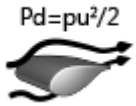


Figure 482:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Laminar** as the Turbulence model.
   c) Activate the **Include gravitational acceleration** checkbox and set the gravity in the y direction to `-9.81`.
   d) Activate the **Heat transfer** checkbox.

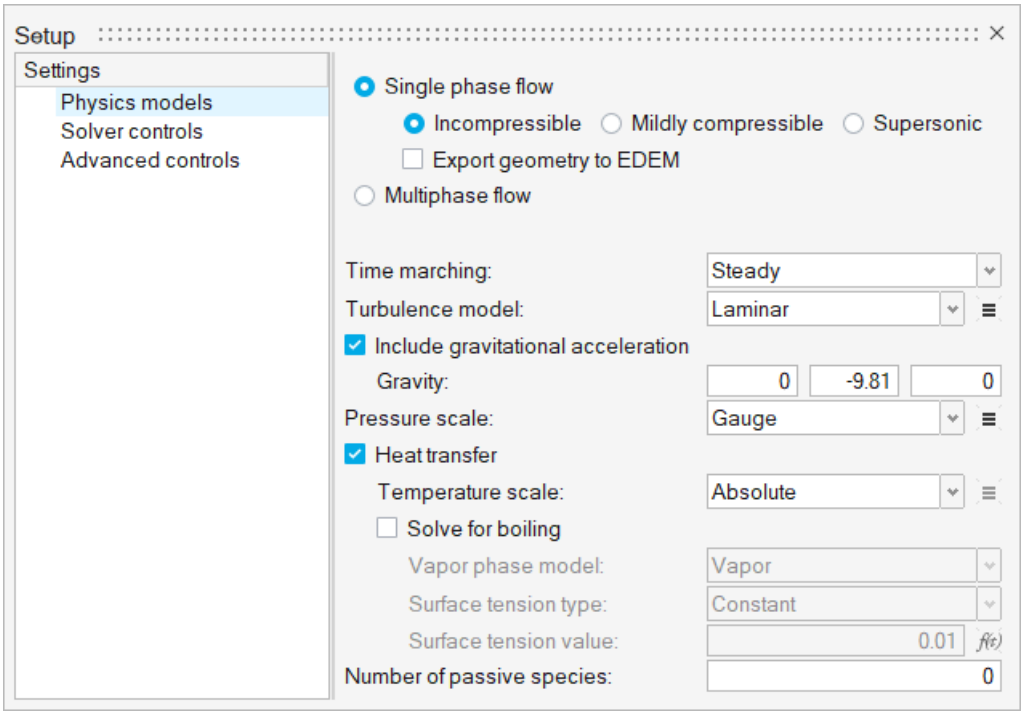


Figure 483:

3. Click the **Solver controls** setting and activate the **Thermal flow** equation.

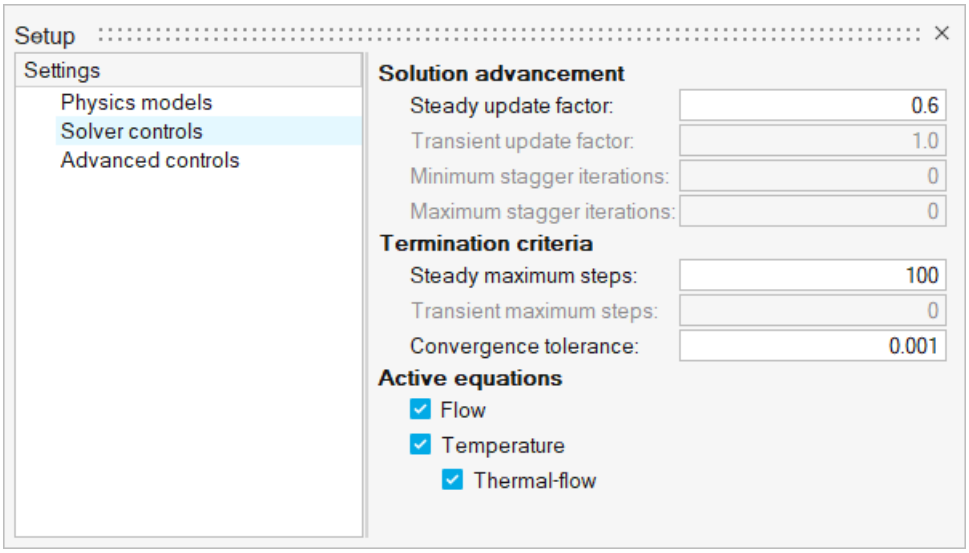


Figure 484:

4. Close the dialog and save the model.

Define the Material Models

1. From the Flow ribbon, click the **Material Library** tool.

Figure 485:

The **Material Library** dialog opens.

2. Click the **My Materials** tab.
3. Click + to add a new fluid material model.
4. In the material creation dialog, click the name in the top-left corner and rename the material to `Air_Boussinesq`.
5. In the Density tab,
   a) Set the Type to **Boussinesq**.
   b) Set the Density value to `1.225`.
   c) Set the Expansivity value to `0.00347222`.
   d) Reference temperature value to `288`.

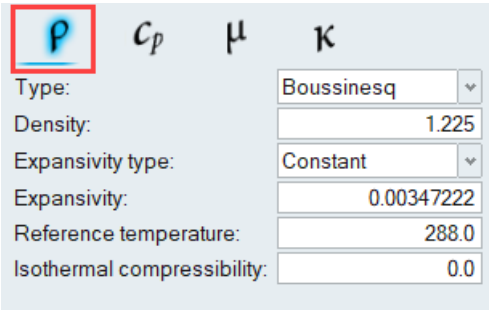


Figure 486:

6. Click the **Specific Heat** tab and set the Specific heat value to `1005`.

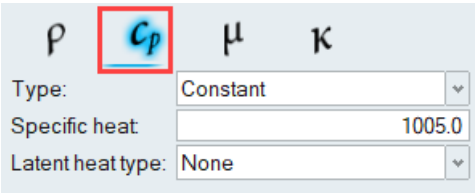


Figure 487:

7. Click the **Viscosity** tab and set the Viscosity value to `1.781e-05`.

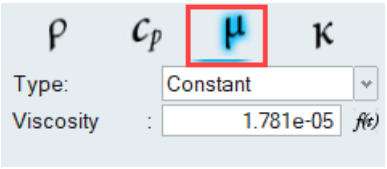


Figure 488:

8. Click the **Conductivity** tab and set the Conductivity value to `0.02521`.

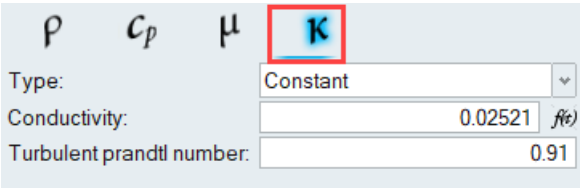


Figure 489:

9. Close the material creation dialog to return to the **Material Library** dialog.
10. Select **Solid** in the Settings menu, click the **My Materials** tab, then click **+** to create a new solid material model.
11. Name the material `Plastic` and set the following values.

    The Type should be Constant for each property.

    - Density: `1270`

- Specific Heat: `1900`
- Conductivity: `0.2`

12. Close the material creation dialog to return to the **Material Library** dialog.
13. Similarly, create new solid material models named `Arnite` and `LED` with the following properties. The Type should be Constant for each property.

    `Arnite`:

    - Density: `1670`
    - Specific Heat: `2050`
    - Conductivity: `1.6`

    `LED`:

    - Density: `5500`
    - Specific Heat: `0.3`
    - Conductivity: `5.0`

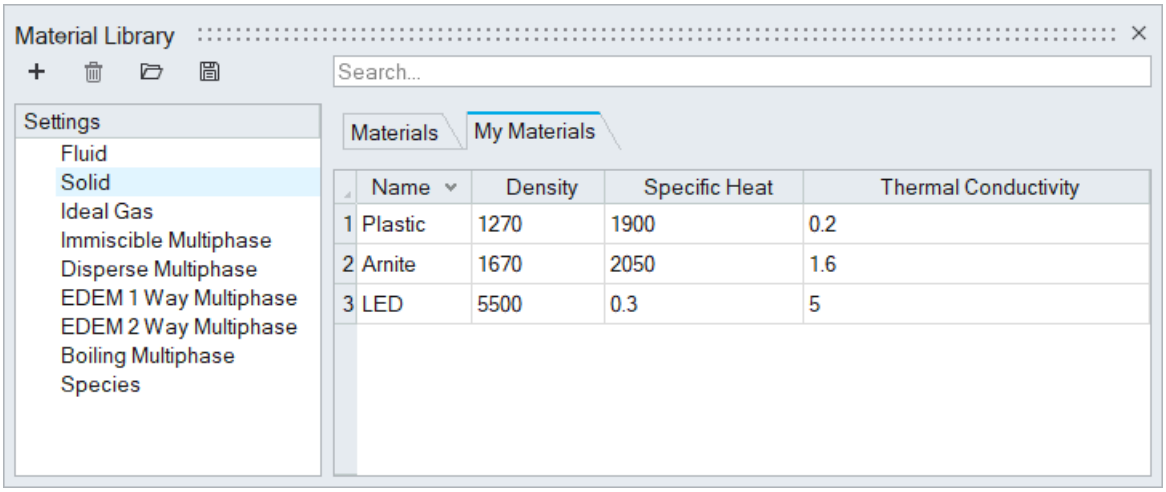


Figure 490:

14. Close all dialogs and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 491:

2. Click the lens volume highlighted in the figure below and select **Arnite** from the Material drop-down menu.

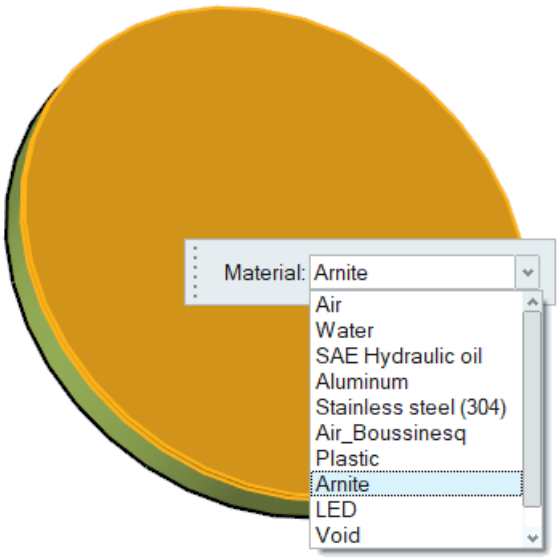


Figure 492:

3. On the guide bar, click ▸ to execute the command and remain in the tool.
4. Click the housing volume and assign the **Plastic** material model.

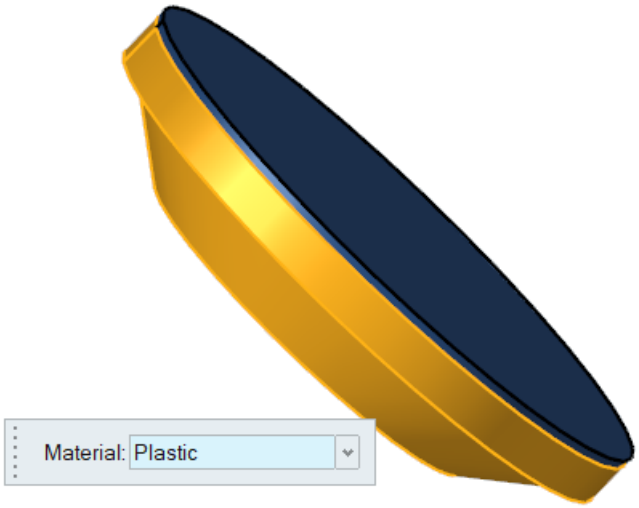


Figure 493:

5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. In the Materials legend, right-click on **Air** and select **Isolate**.
7. Click the air volume and assign the **Air_Boussinesq** material model.

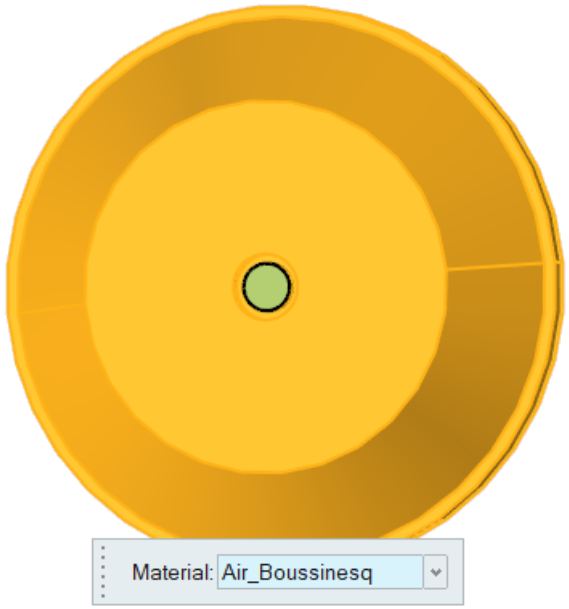


Figure 494:

8. On the guide bar, click ▸ to execute the command and remain in the tool.
9. In the Materials legend, right-click on **Air** and select **Isolate**.
10. Click the bulb volume and assign the **LED** material model.

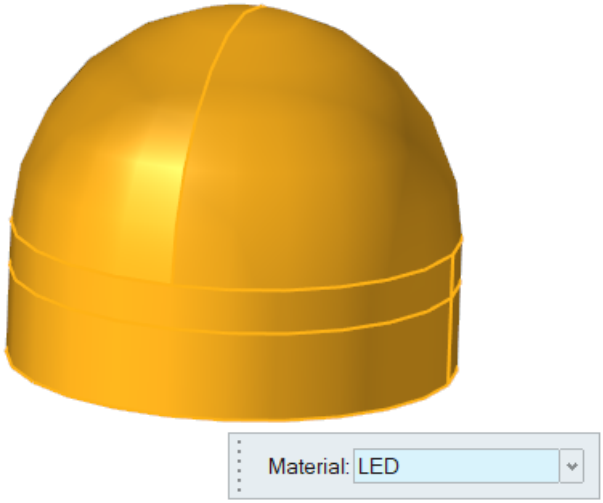


Figure 495:

11. On the guide bar, click ✓ to execute the command and exit the tool.
12. Save the model.

Define the Heat Source

1. From the Flow ribbon, click the **Sources** > **Heat** tool.

Figure 496:

2. In the modeling window, select the bulb volume.
3. In the **Heat Source** dialog, set the heat source value to `2049180` W/m^3.

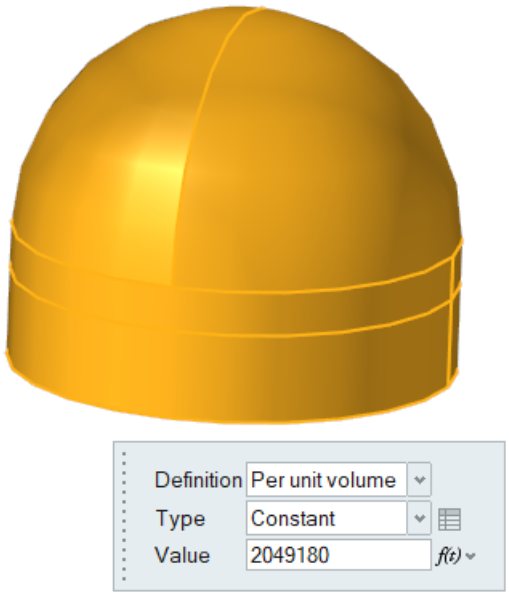


Figure 497:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Press **Esc** to exit the Sources tool then press the **A** key to turn on the display of all solids.
6. Save the model.

Define Flow Boundary Conditions

In this problem, all the surfaces are walls and will therefore be assigned the default wall boundary condition. The outer walls of the headlamp will be given a no-slip wall boundary condition with a constant temperature.

1. From the Flow ribbon, click the **No Slip** tool.

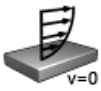


Figure 498:

2. In the modeling window, select the surfaces highlighted in the figure below.

Figure 499:

3. In the microdialog, enter the values shown in the figure below.

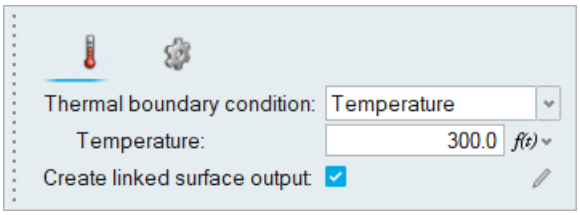


Figure 500:

4. In the Boundaries legend, double-click **Wall**, rename it to `Outerwalls`, then press **Enter**.
5. On the guide bar, click ✓ to execute the command and exit the tool.
6. Save the model.

Set Up Radiation

In this step, you will specify the parameters related to the thermal radiation setup.

Define Radiation Model Settings

1. From the Radiation ribbon, Thermal Radiation tools, click the **Physics** tool.

Figure 501:

The **Radiation Settings** dialog opens.

2. Activate **Thermal radiation** and set the Radiation model to **Discrete Ordinate**.
3. Verify that the Radiation quadrature is set to **S4**.

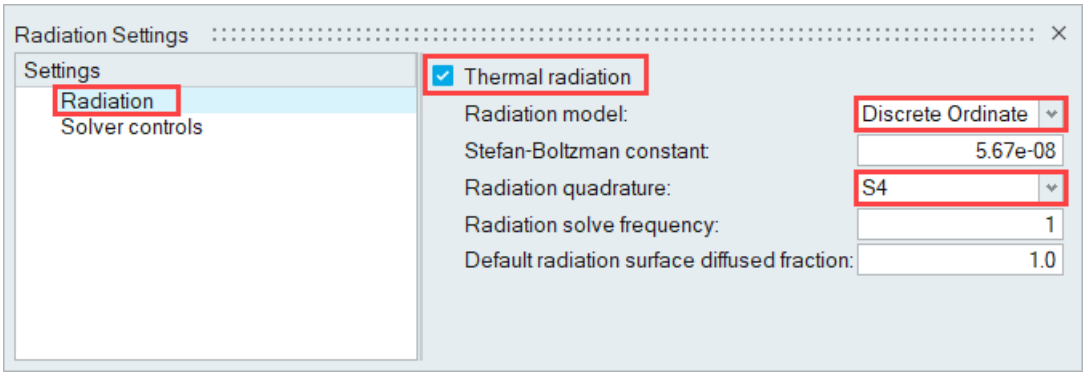


Figure 502:

4. Close the dialog.

Define the Emissivity Models

1. From the Radiation ribbon, click the **Surface Finish Library** tool.

Figure 503:

The **Surface finish library** opens.

Click + to add a new emissivity model.

2. Set the Name of the model to `Inner` and the Emissivity value to `0.05` by double-clicking on the entity fields.

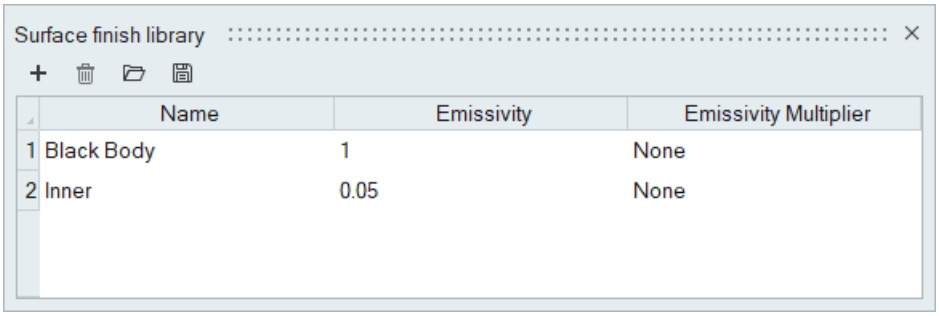


Figure 504:

3. Close the dialog.

Define the Participating Media Radiation Models

1. From the Radiation ribbon, Participating Media tools, click the **Model** tool.

Figure 505:

The **Participating media model library** opens.

2. Click + to add a new model.
3. Name the new model `Air` and assign the following property values:
   - Absorption coefficient - `0`
   - Scattering coefficient - `0`
   - Phase coefficient - `0`
   - Refractive index - `1`
4. Similarly, create another model named `Lens` and assign the following property values:
   - Absorption coefficient - `900`
   - Scattering coefficient - `0`
   - Phase coefficient - `0`
   - Refractive index - `1.57`

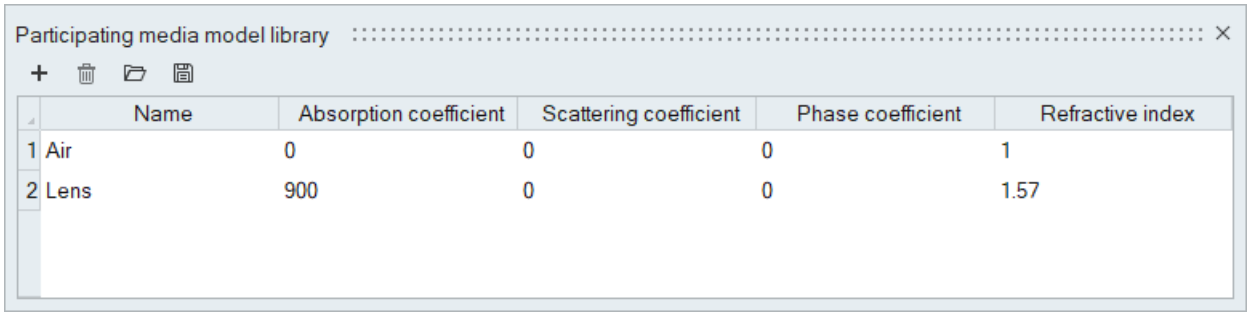


Figure 506:

5. Close the dialog.

Assign Participating Media Models

1. From the Radiation ribbon, click the **Assign** tool.

Figure 507:

2. Click the lens volume and assign the **Lens** model from the microdialog drop-down.

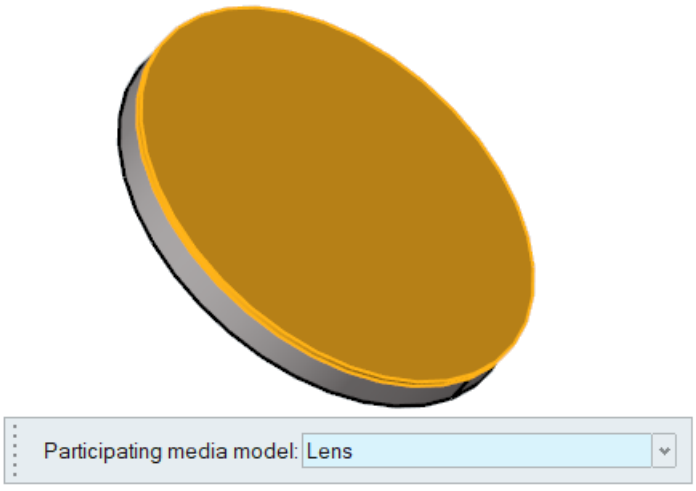


Figure 508:

3. On the guide bar, click ▸ to execute the command and remain in the tool.

4. In the Participating Media legend, right-click on **Lens** and select **Hide**.
5. Select the air volume and assign the **Air** model.

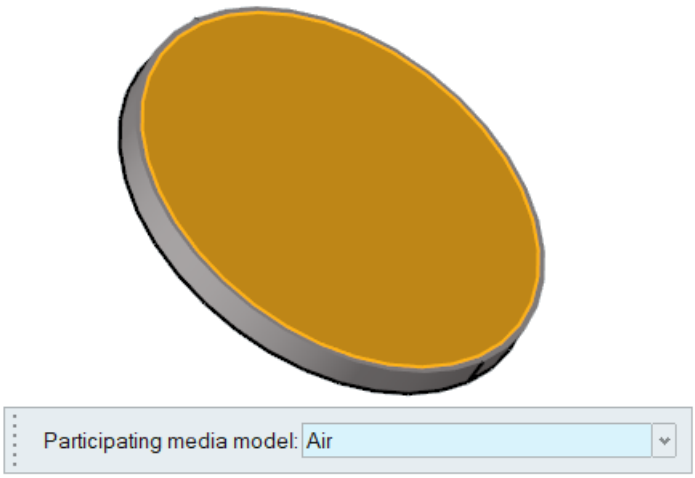


Figure 509:

6. On the guide bar, click ▸ to execute the command and remain in the tool.
7. In the Participating Media legend, right-click on **Air** and select **Hide**.
8. Save the model.

Assign Surface Finish Models

1. From the Radiation ribbon, click the **Surface Finish** tool.

Figure 510:

2. Select only the inner surfaces of the housing - i.e. the housing-air interface.

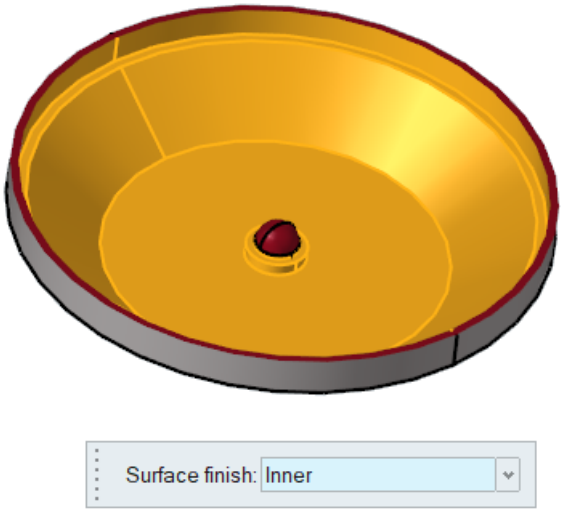


Figure 511:

A total of nine surfaces should be selected.

3. Assign the **Inner** model from the microdialog drop-down
4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Turn on the display of all surfaces in the model by pressing the **A** key.

Define the Radiation Interfaces

Since the lens and air are defined as semi-transparent media, the interface between the two will be modeled as a specular interface of type: internal. Accordingly, the outer surfaces of the lens will be modeled as external interface radiation surface.

1. From the Radiation ribbon, click the **Interface** tool.

Figure 512:

Notice that all the surfaces except the Lens-Air interface surface are made transparent. Since the Air and Lens volumes are defined as participating media, HyperMesh CFD automatically detects the surfaces that can be defined as an internal radiation interface and only those surfaces will be available for selection when this tool is active.

2. Using the box selection method, draw a window around the model.
Observe that the lens-air interface is the only surface that is selected.

3. In the microdialog, set the Radiation diffused fraction to `0`.

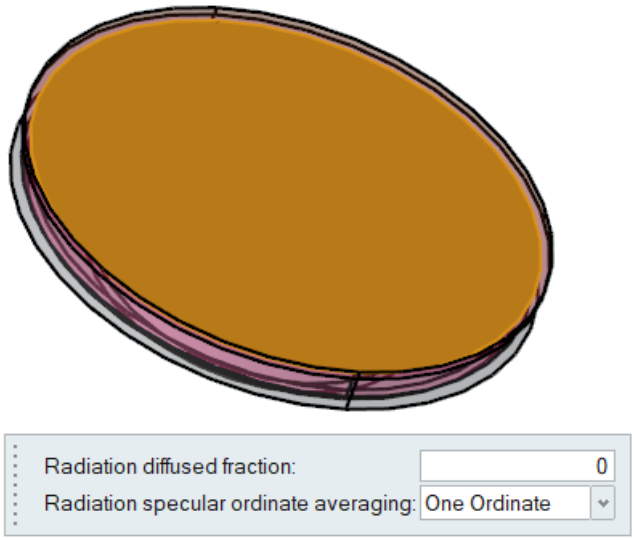


Figure 513:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **External Boundary** tool.

Figure 514:

6. Select the three outer surfaces of the lens volume.
7. In the microdialog, set the Radiation diffused fraction to `1`, the External refractive index to `1`, and the External temperature to `300` K.

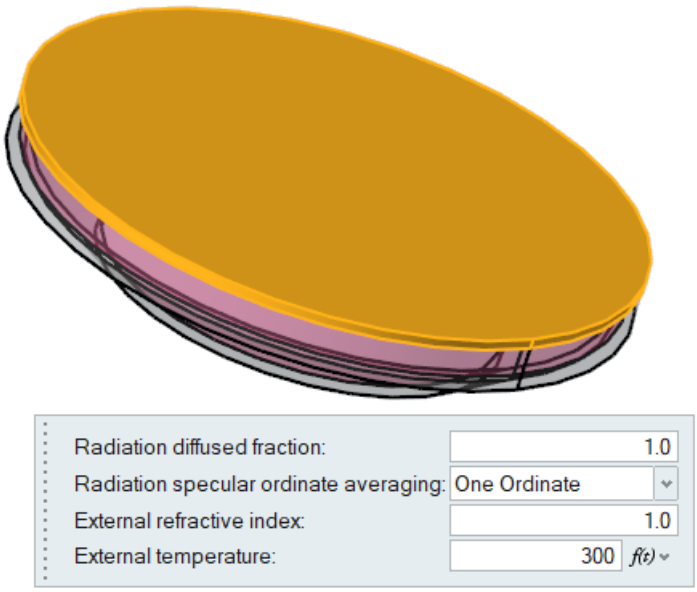


Figure 515:

8. On the guide bar, click ✓ to execute the command and exit the tool.
9. Save the model.

Generate the Mesh

In this step, you will define the mesh controls and then generate the mesh.

Define the Surface Mesh Controls

1. From the Mesh ribbon, click the **Surface** tool.

Figure 516:

2. Using the box selection method, select all the surfaces in the model.
3. In the microdialog, set the Average element size to `0.002`.

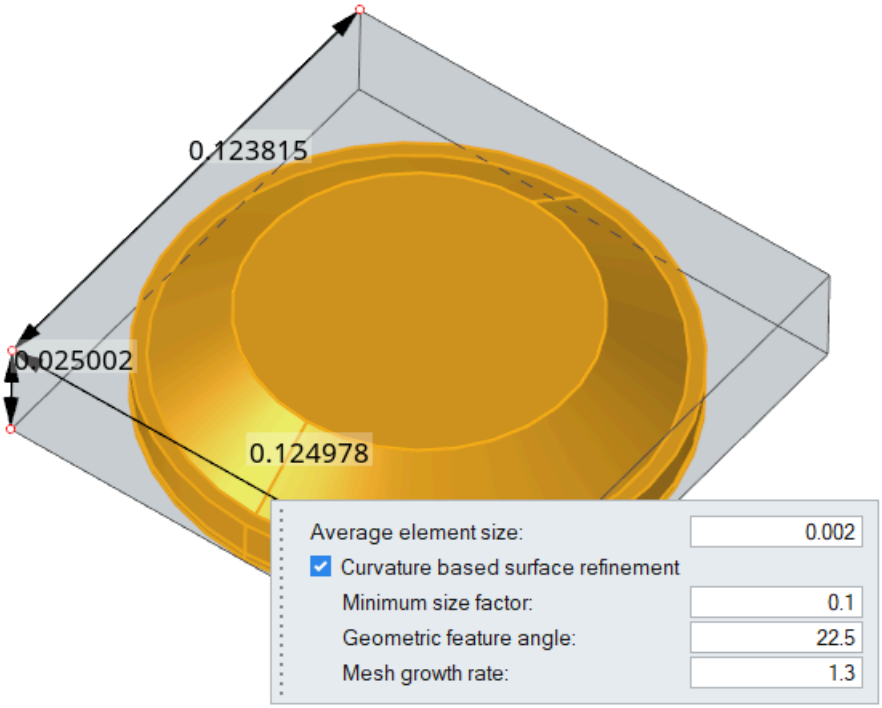


Figure 517:

4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Select the surface highlighted in the figure below and hide it by pressing the **H** key or by right-clicking and select **Hide** from the context menu.

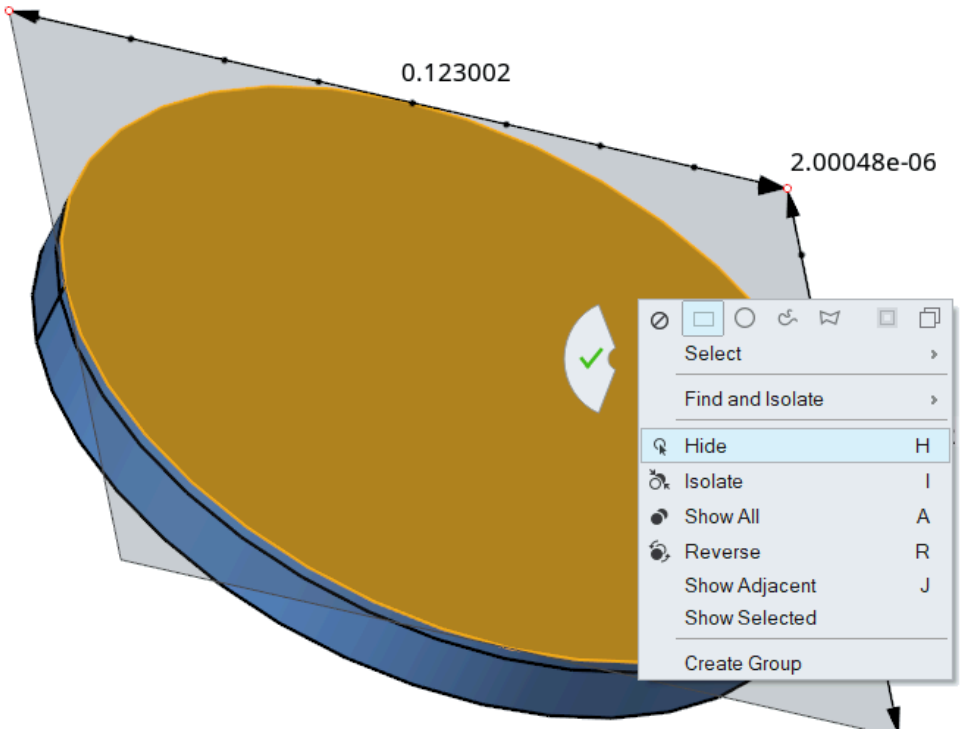


Figure 518:

6. Select the three surfaces highlighted in the figure below.
7. In the microdialog, set the Average element size to `0.001` and the Mesh growth rate to `1`.

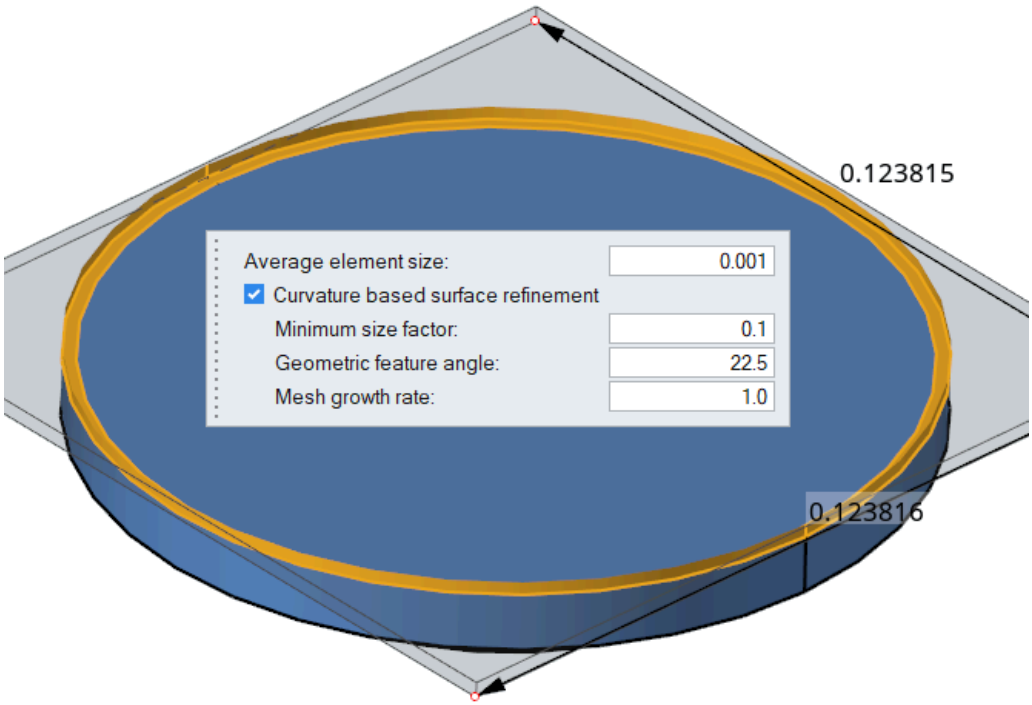


Figure 519:

8. On the guide bar, click ✓ to execute the command and exit the tool.

Define the Boundary Layer Controls

1. From the Mesh ribbon, click the **Boundary Layer** tool.

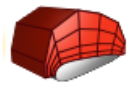


Figure 520:

2. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Material** > **Air_Boussinesq**.
   All the fluid wall surfaces should be selected and a microdialog for BL specification appears.
3. Enter the following values in the microdialog:
   - First layer thickness definition: **Constant**
   - First layer thickness: `0.0005`
   - Total number of layers: `4`
   - Growth method: **Constant**
   - Initial growth rate: `1.3`
   - Termination policy: **Truncate**

- Activate the **Enable surface mesh modification** option

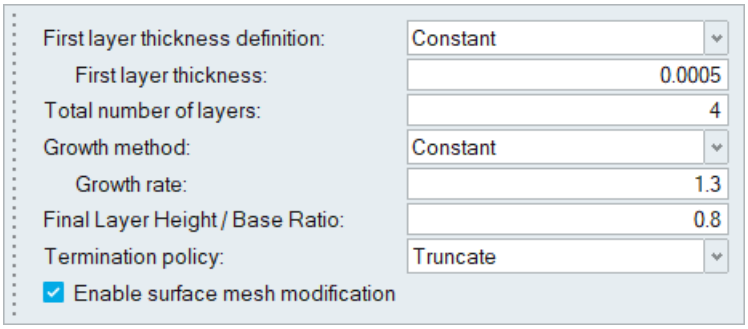


Figure 521:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Turn on the display of all the surfaces.

Define the Volume Mesh Controls

Since the thickness of the housing and the lens solids are small, you will use the thin layer meshing tool so that when the volume mesh is generated, there will be two layers across the thickness of those solids.

1. From the Mesh ribbon, click the **Volume Mesh** tool.

Figure 522:

2. Select the housing and the lens solids.
3. In the microdialog,
   a) Set the Average size to `0.001`.
   b) Set the Growth rate to `1.0`.
   c) Activate the **Thin layer meshing** option and set the Number of layers to `2`.

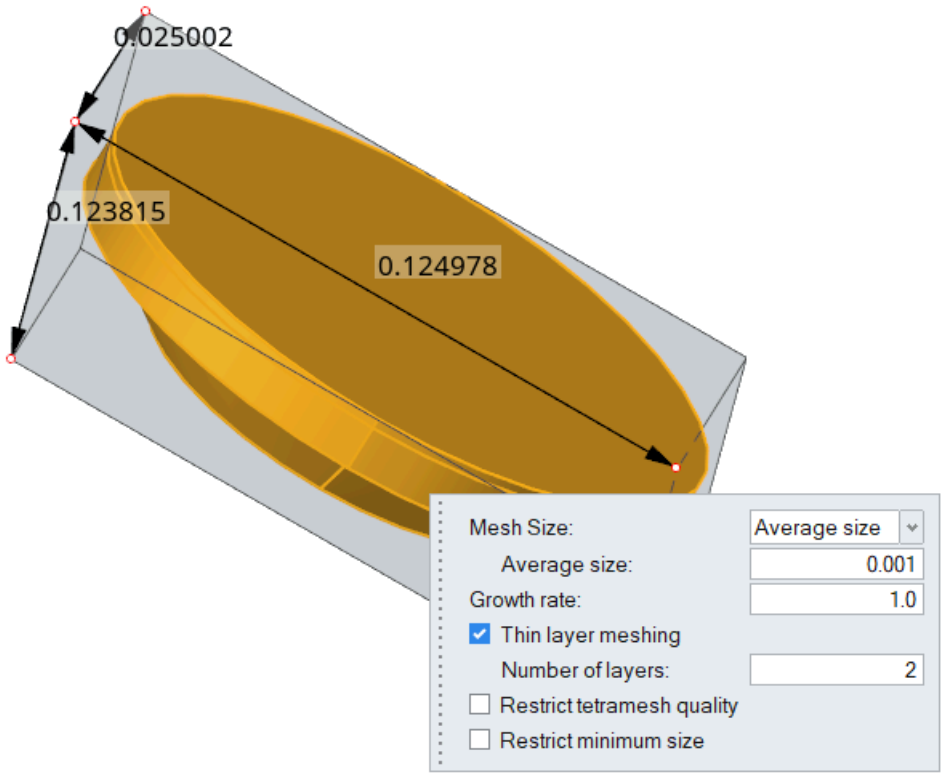


Figure 523:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

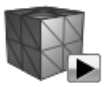


Figure 524:

The **Meshing Operations** dialog opens.

2. Set the Mesh growth rate to `1`.
3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 525:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand the **Default initial conditions** menu and deactivate the **Pre-compute flow** option.
5. Set the x-velocity to `0` and the Temperature to `300`.
6. Leave the remaining options as default and click **Run** to launch AcuSolve.

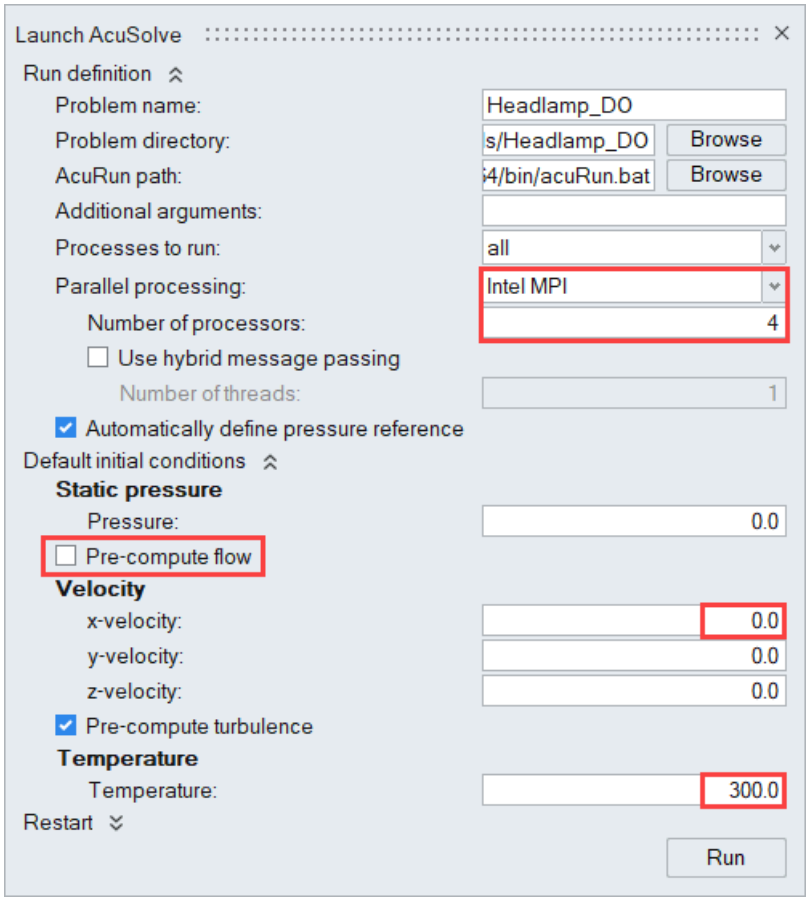


Figure 526:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Slice Planes** tool.

Figure 527:

5. Select the plane perpendicular to the x-axis, as shown in the figure below.

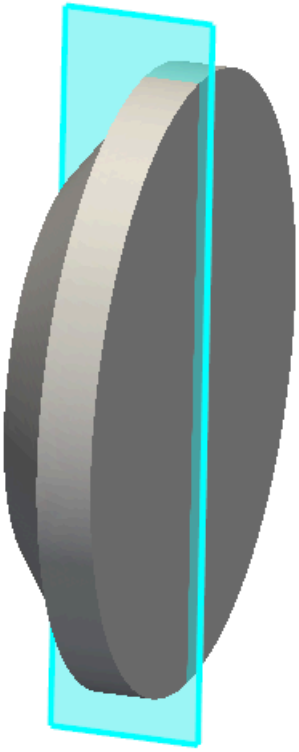


Figure 528:

6. Click the **Front** face on the View Cube to align the model.

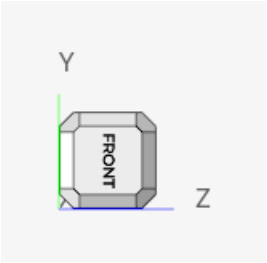


Figure 529:

7. Click in the slice plane microdialog.
8. In the display properties microdialog, set the display to **temperature**.
9. Activate the **Legend** toggle and click to reset the range.
10. Click , set the Legend location to **Upper Right Corner**, and the Colormap Name to **Rainbow Uniform**.

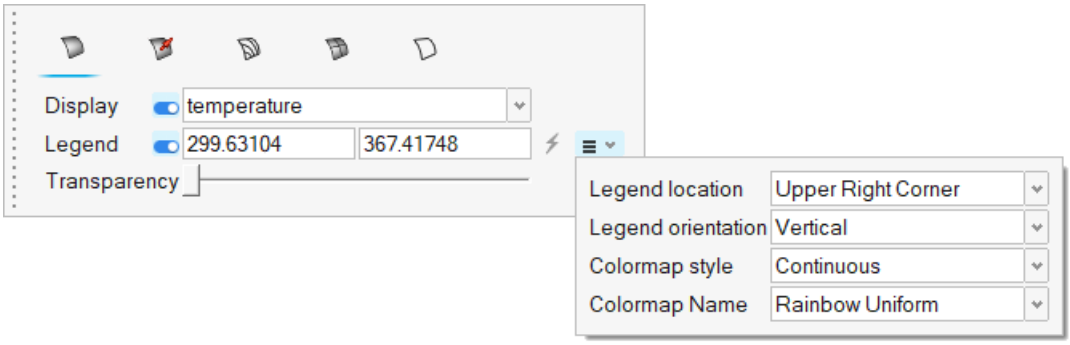


Figure 530:

11. Click ✓ on the guide bar.

12. In the Post Browser, hide the **Boundary Collections**.

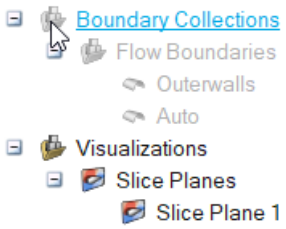


Figure 531:

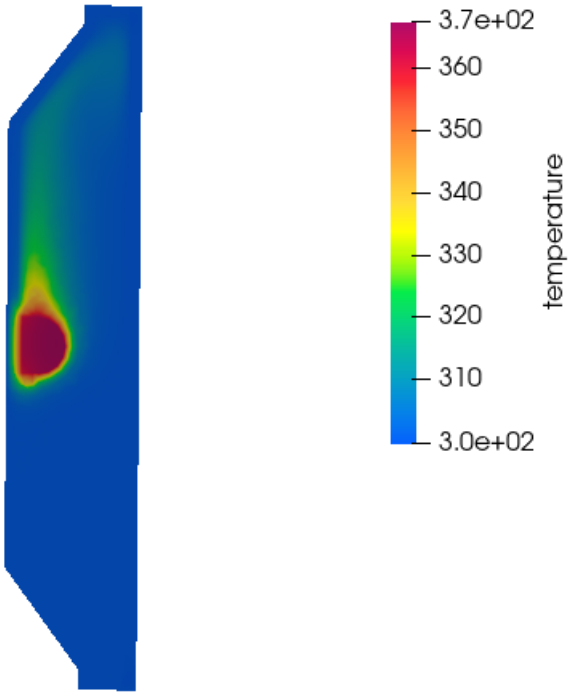


Figure 532:

13. Right-click on **Slice Plane 1** in the browser and select **Edit**.
14. Change the display to **incident radiation**.
15. Click ⚡ to refresh the legend's range.
16. Click ✓ on the guide bar.

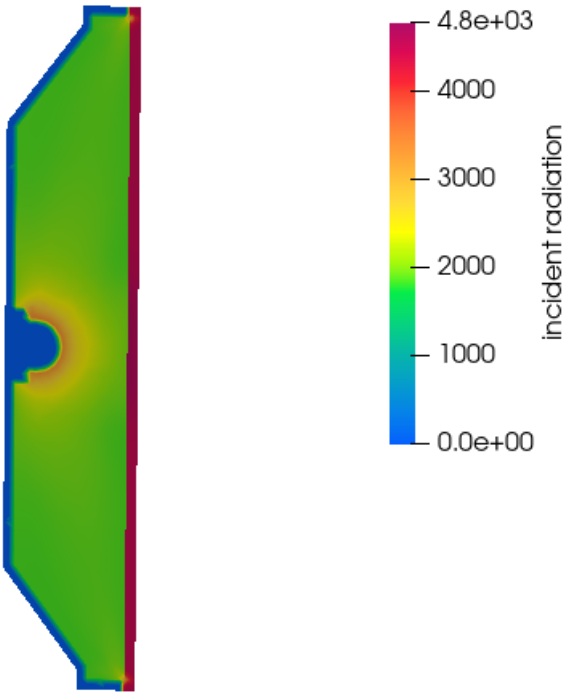


Figure 533:

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer problem in a headlamp using the discrete ordinate radiation model in HyperMesh CFD. You started by importing and cleaning up the geometry and then set up the flow, thermal, and radiation boundary conditions. Once the solution was computed, you processed the results using the Post ribbon where you created contour plots of temperature and incident radiation.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/8ctsmherhq

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3300 Modeling of a Heat Exchanger Component

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial provides instructions for modeling a heat exchanger component using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3300_HeatExchanger.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

Heat exchangers are devices used to facilitate heat exchange between two fluids that are at different temperatures. A typical heat exchanger device has two separate flows, the cross flow and tubular flow. Depending on the temperatures, heat can flow to or from one fluid to the other through the solids separating them. Since the heat exchanger geometry is quite complex with a large number of plates and pipes, it is not realistic to model the actual heat exchanger when performing system level simulations. In such scenarios, a simplified approach can be employed where only the effect of the heat exchanger on the cross flow is considered instead of modeling the actual heat exchanger. This is done using the heat exchanger component available in AcuSolve.

The heat exchanger component has two effects on the cross flow:

1. Pressure drop which is calculated based on the friction parameters provided by the user.
2. Heat addition or removal based on the coolant heat transfer parameters.

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a cylindrical pipe channel with an interior heat exchanger component volume with thickness ‘t’. Air enters the pipe at a velocity of 0.1 m/sec and flows through the heat exchanger volume and then exits through the outlet. As the air flows through the heat exchanger component, heat is added to it from the coolant such that the total coolant heat reject is 200 W.

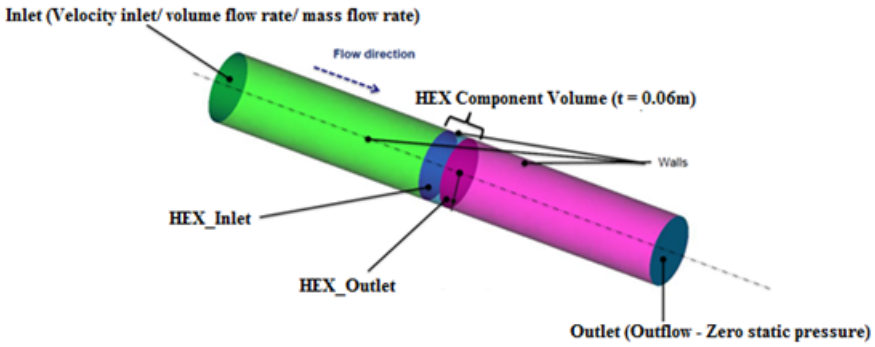


Figure 534:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 535:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3300_HeatExchanger.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `HeatExchanger` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `HeatExchanger` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 536:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

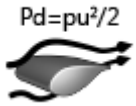


Figure 537:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.
   c) Activate the **Heat transfer** checkbox.

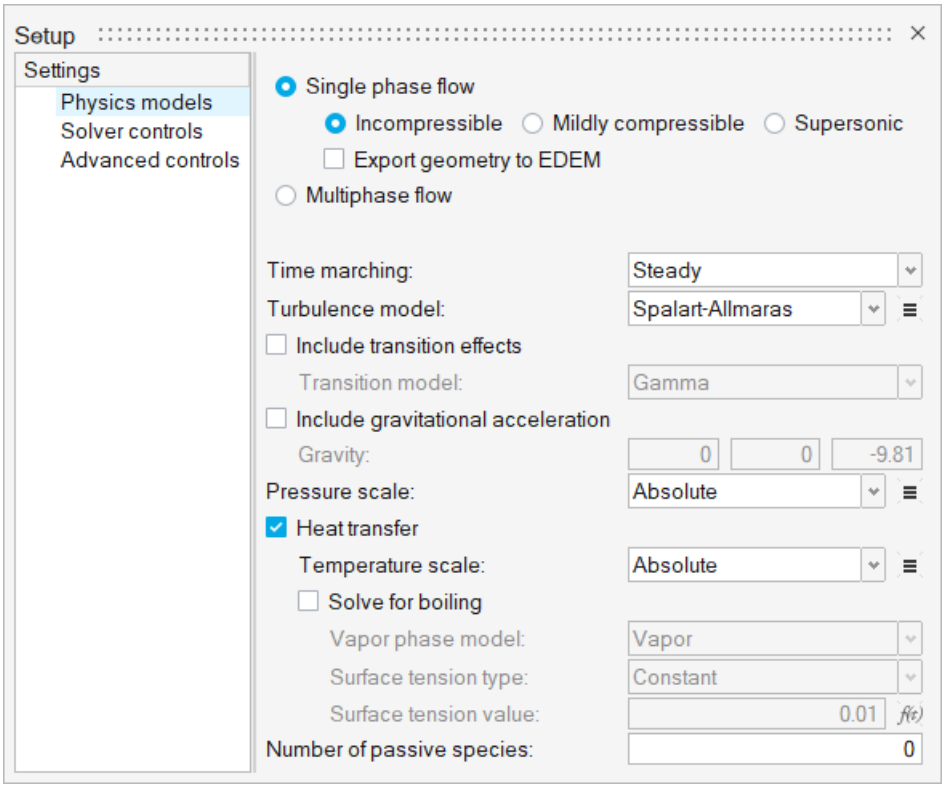


Figure 538:

3. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 539:

2. Verify that the Air material has been assigned to all three volumes.
3. Click ✓ on the guide bar to exit the tool.

Define the Heat Exchanger Component

1. From the Flow ribbon, click the arrow next to the Domain tool set, then select **Heat Exchanger**.

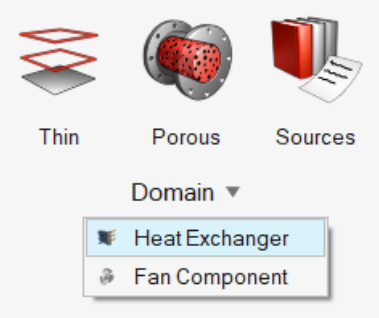


Figure 540:

2. Select the middle solid as the heat exchanger component volume.

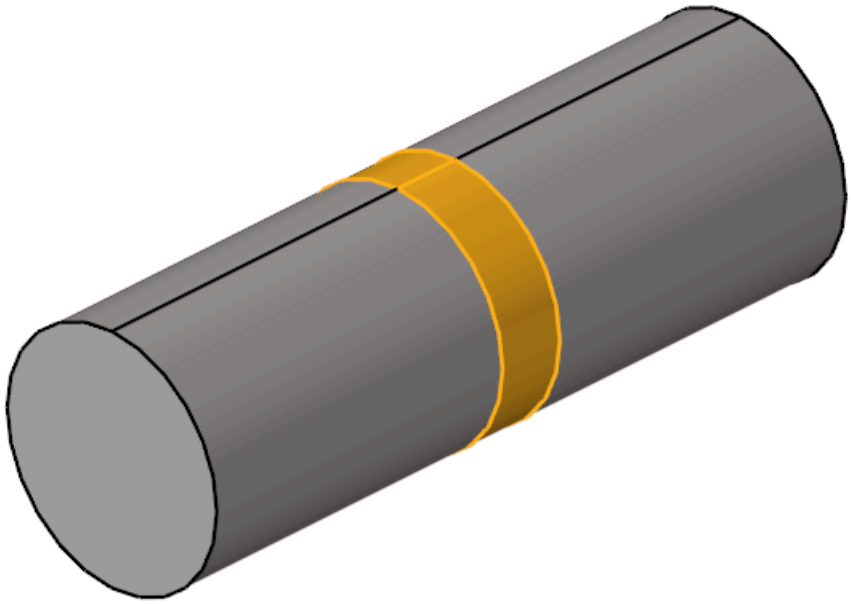


Figure 541:

3. On the guide bar, click **Inlet** then select the face shown below as the inlet of the heat exchanger component.

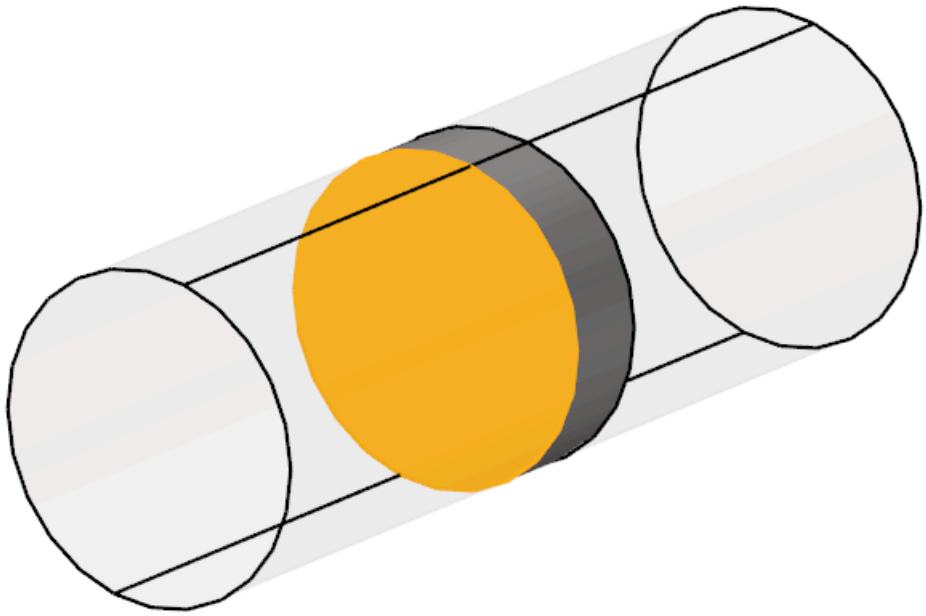


Figure 542:

4. In the microdialog, enter the following parameters:

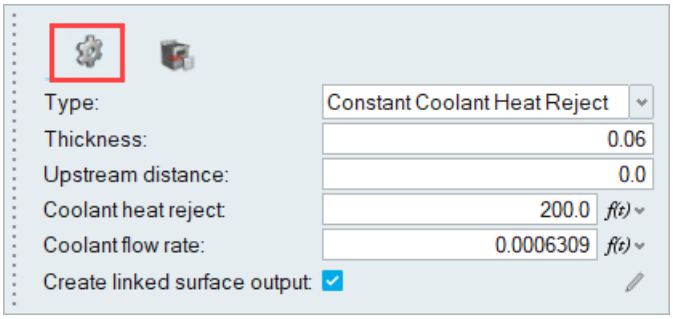


Figure 543:

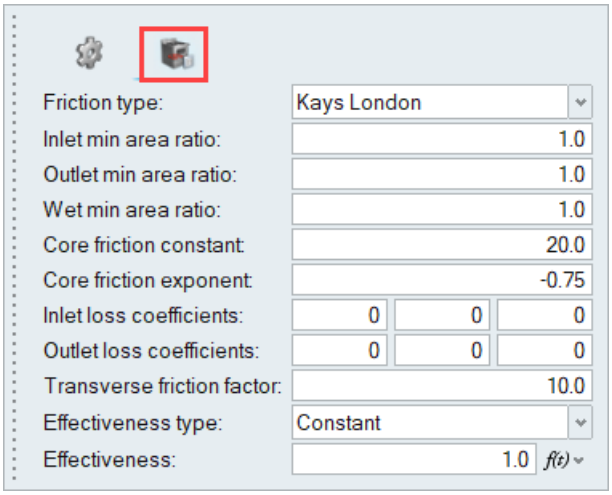


Figure 544:

5. On the guide bar, click ✓ to execute the command and exit the tool.
6. Save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 545:

2. Click the inlet face highlighted in the figure below.

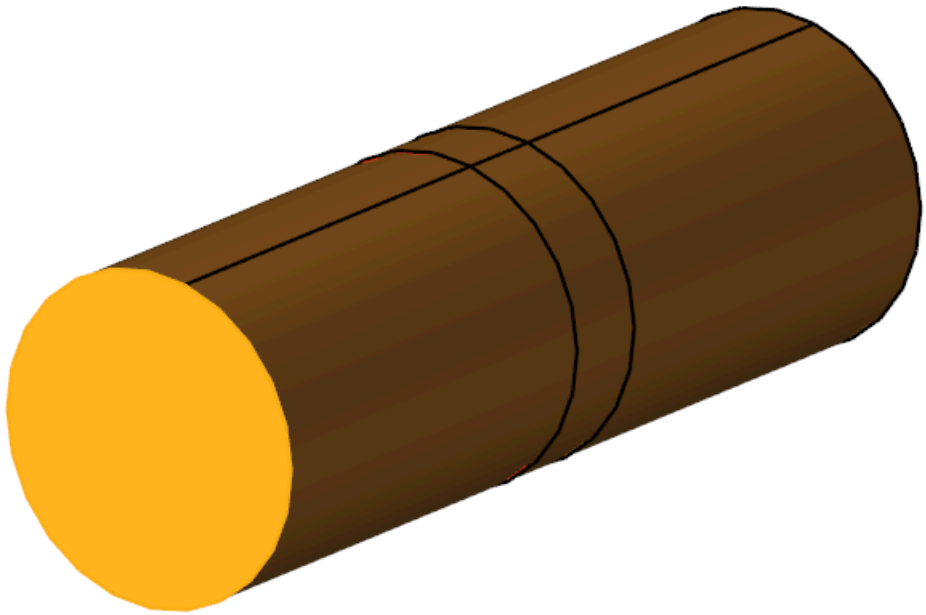


Figure 546:

3. In the microdialog, enter the following values for the Momentum and Turbulence tabs.

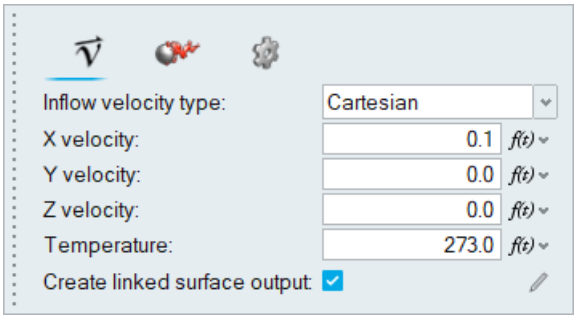


Figure 547:

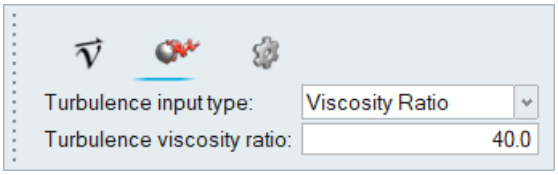


Figure 548:

4. On the guide bar, click ✓ to execute the command and exit the tool.

5. Click the **Outlet** tool.

Figure 549:

6. Select the face highlighted in the figure below and then click ✓ on the guide bar.

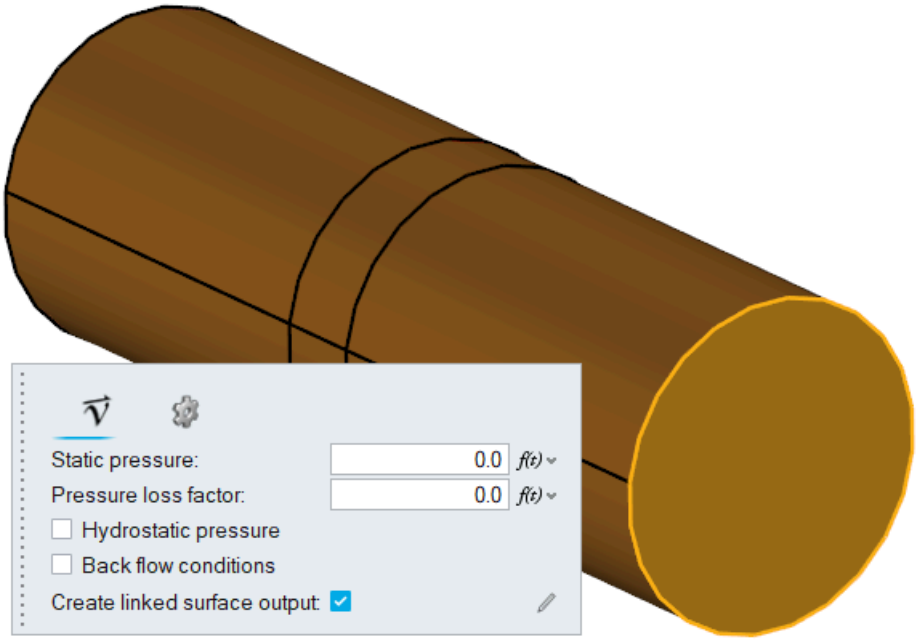


Figure 550:

7. Save the model.

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

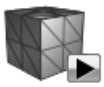


Figure 551:

2. In the **Meshing Operations** dialog, set the Mesh growth rate to `1.1` (if not set already).

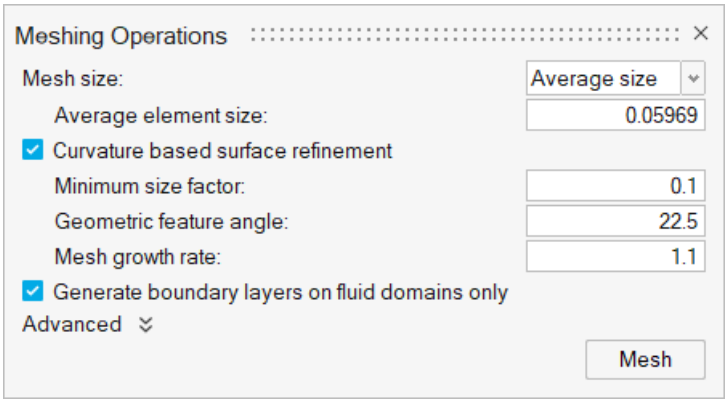


Figure 552:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 553:

   The **Launch AcuSolve** dialog opens.
2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Leave the remaining options as default and click **Run** to launch AcuSolve.

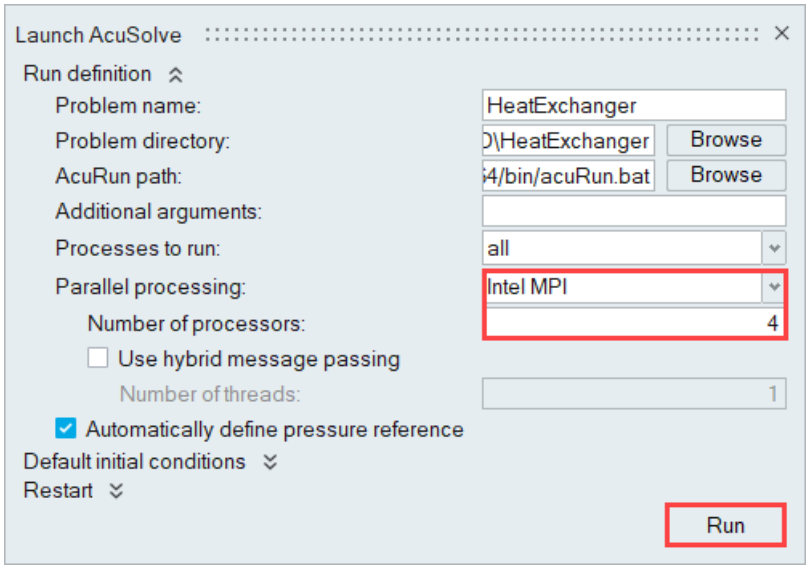


Figure 554:

Post-Process with the Plot Tool

1. In the **Run Status** dialog, right-click on the AcuSolve run.

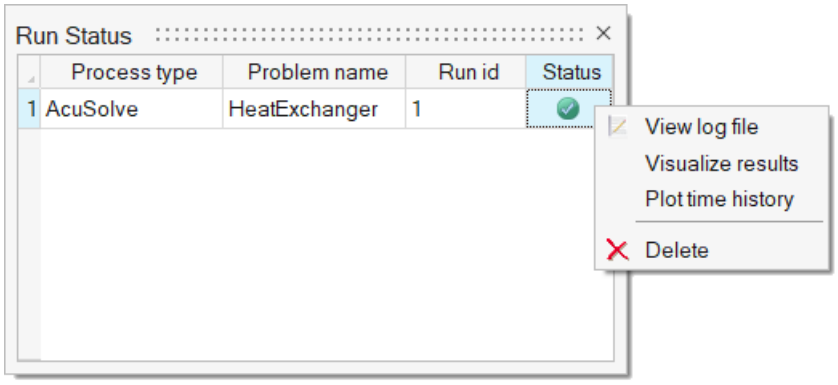


Figure 555:

2. Select **Plot time history** to plot the residual and solution ratios.

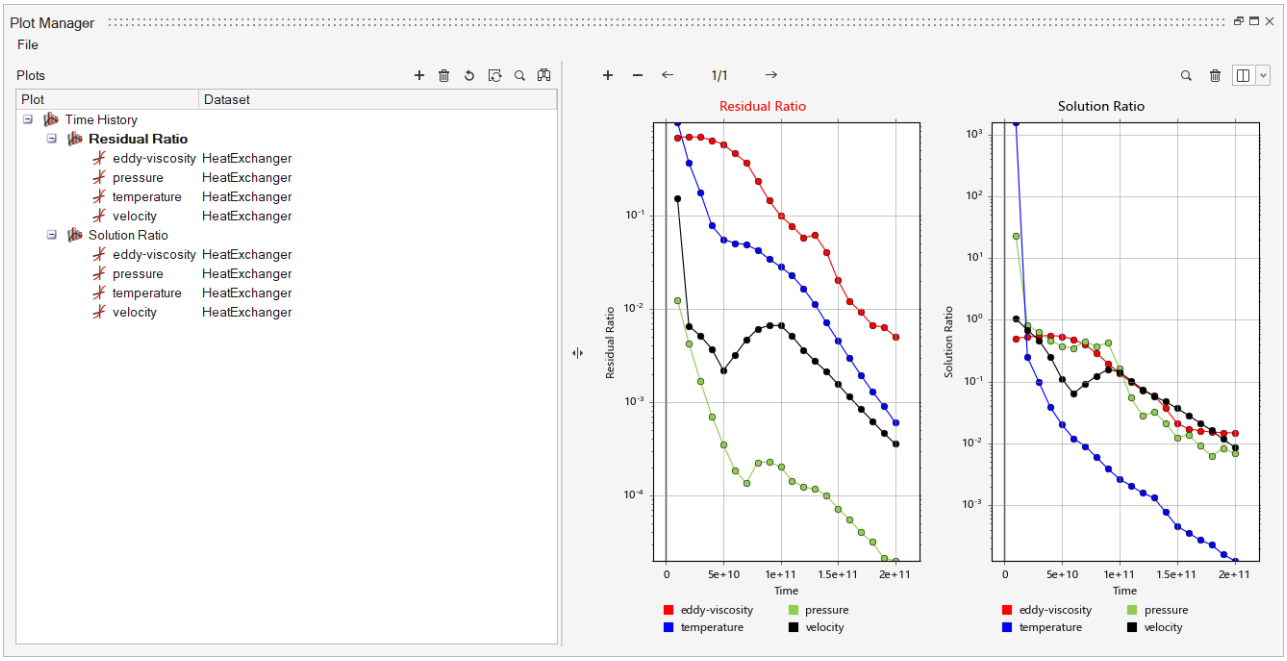


Figure 556:

3. Click + to create a new plot.
4. Right-click **User Defined** under the Model panel to create a user defined function.
5. In the Data Tree, expand **Heat Exchanger Component Output** then select **coolant temperature**.
6. In the Name field, enter `dT`.
7. In the Value field, type `value=`.
8. Under Parts, choose **HX component** then click to insert the field as part of the value, as shown below.

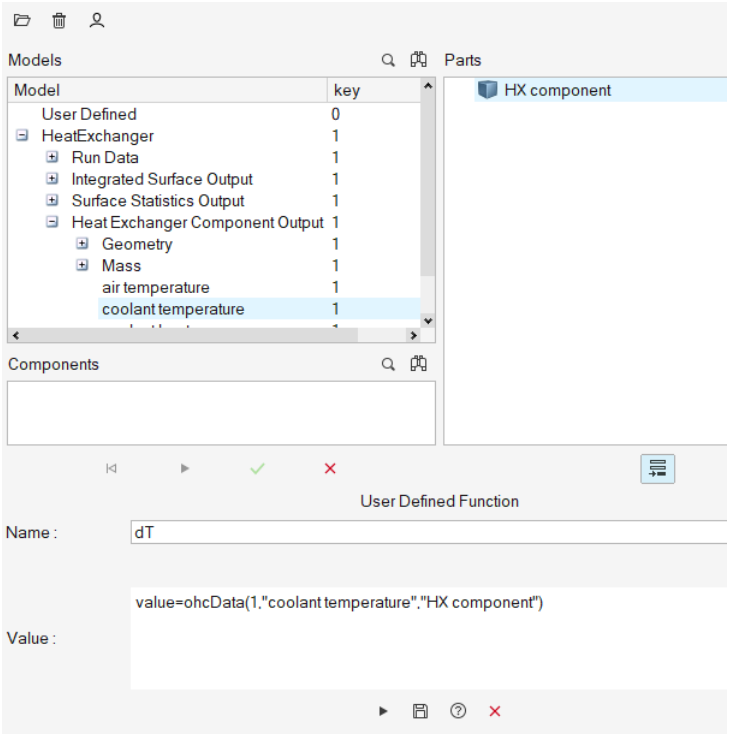


Figure 557:

9. Type – at the end of the user defined function value.
10. In the Data Tree, select **air temperature**, then pick **HX component** and click to insert the field as part of the value.
11. Click ▸ to add the user defined function.

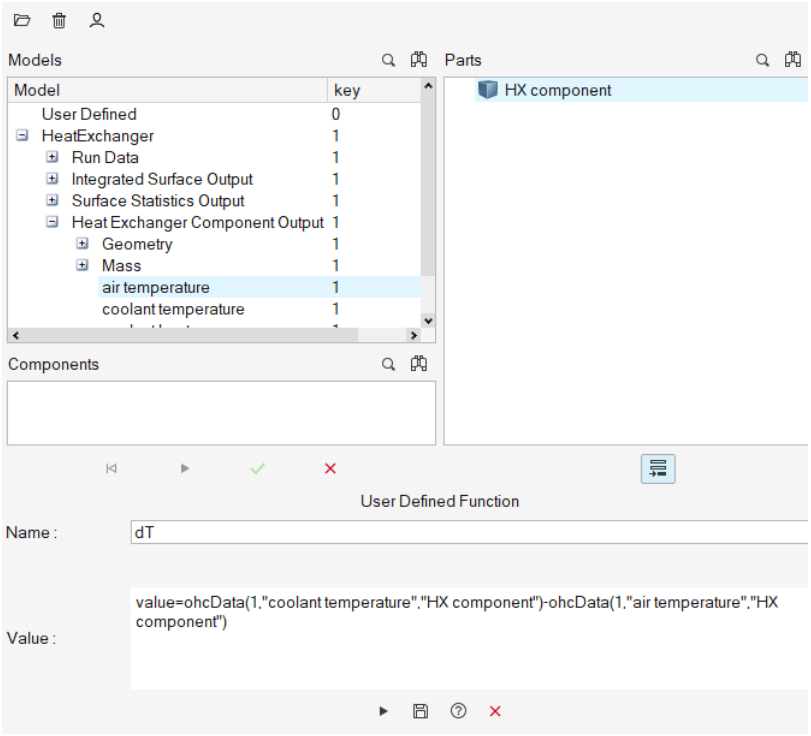


Figure 558:

12. Click the **x axis** to switch to Time Steps.

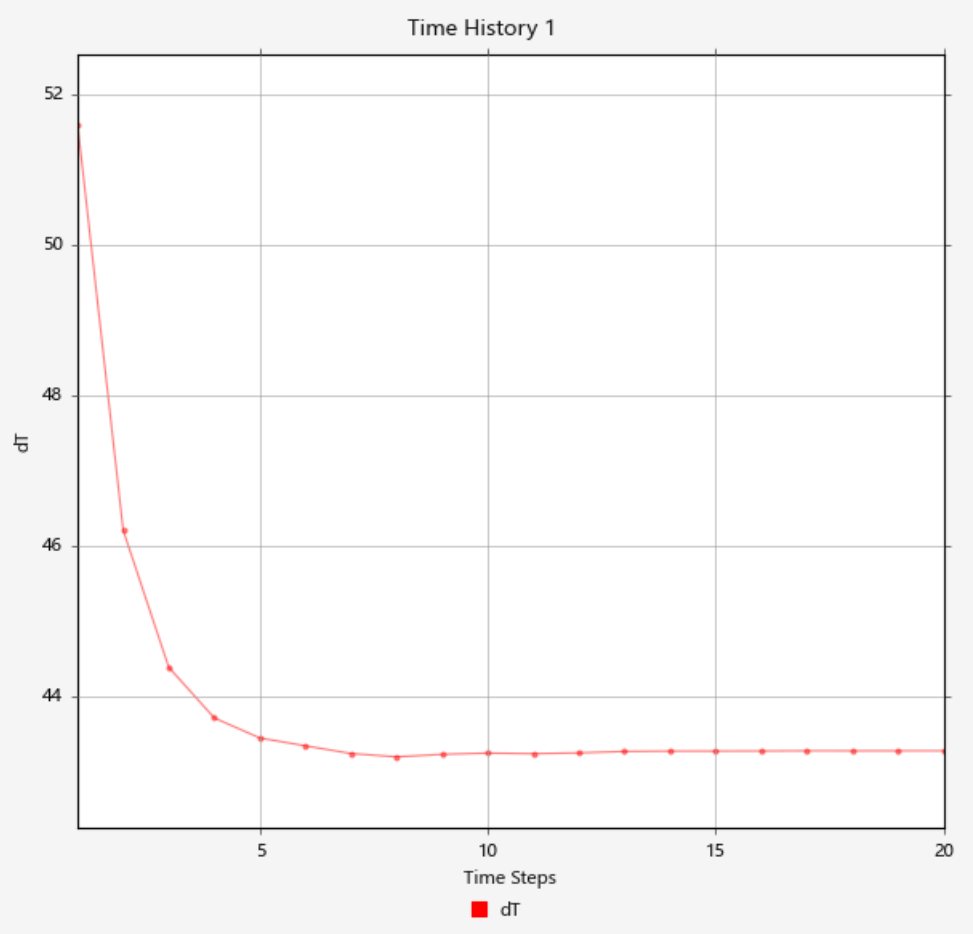


Figure 559:

As shown in the plot above, the temperature rise is 43.3K.

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a Heat Exchanger component using HyperMesh CFD. You started by opening the HyperMesh input file with the geometry and then defined the simulation parameters, the heat exchanger component, and the flow boundary conditions. Once the solution was computed, you defined a user-function in the Plot Manager in order to create a plot of the temperature rise across the heat exchanger volume.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/dsfvhoy05z

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3310 Single Phase Nucleate Boiling

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial provides instructions for modeling a single-phase nucleate boiling using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3310_NB1_Steiner.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 560. It is based on the popular wall heat transfer model for sub-cooled boiling (Steiner Model). It consists of a channel with a heated wall at the bottom. The temperature of the wall is selected to onset the nucleate boiling at the heated wall.

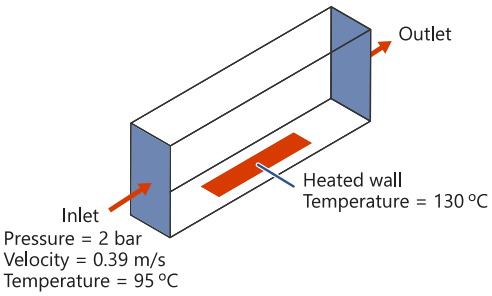


Figure 560: Schematic of Channel

The dimensions of the inlet are 0.03 x 0.04 m; the inlet velocity (v) is 0.39 m/s and the temperature (T) of the fluid entering the inlets is 368.15 K (95 C).

The preheated air enters the inlets and heat is transferred to the fluid from the walls. The heat causes sub-cooled boiling to occur in the region close to the wall and leads to formation of bubbles at nucleation sites.

The heat transfer in this regime is basically dominated by two effects, the macro convection due to the motion of the bulk liquid and the latent heat transport associated with the evaporation of the liquid micro-layer between the bubble and the heated wall.

The fluid in this problem is water, which has temperature dependent material properties: density, viscosity, enthalpy and conductivity. There are also surface tension and vapor phase models specified for this material.

Water vapor which also has temperature dependent material properties is specified as the vapor phase model.

The AcuSolve simulation will be set up to model steady state heat transfer to determine the temperature and heat flux on the heated walls of the manifold.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 561:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3310_NB1_Steiner.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `NB1` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `NB1_Steiner` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 562:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

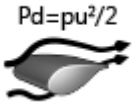
   

   Figure 563:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.
   c) Activate the **Heat transfer** checkbox.
   d) Click the **Solve for boiling** checkbox to activate Nucleate Boiling.
   e) Select **Vapor_Therm** for the Vapor phase model.
   f) Check that the Surface tension type is set to **Constant** and set the Surface tension value to `0.01`.

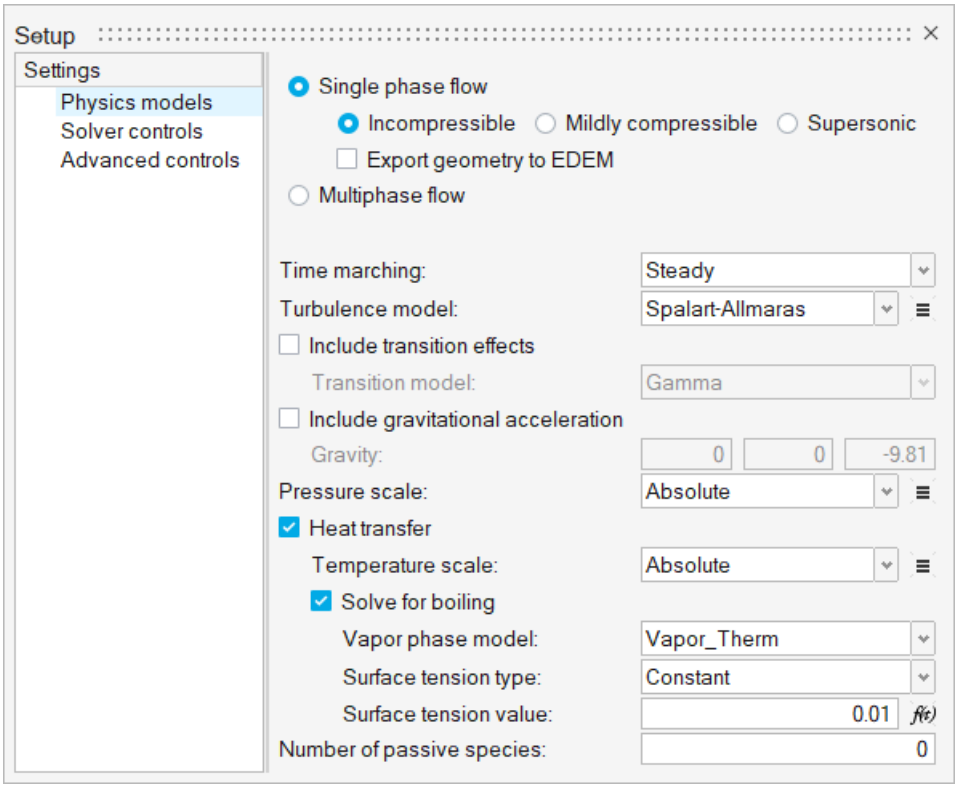


Figure 564:

3. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 565:

2. Select the model solid.
3. Select **Water_Therm** from the Material drop-down menu.

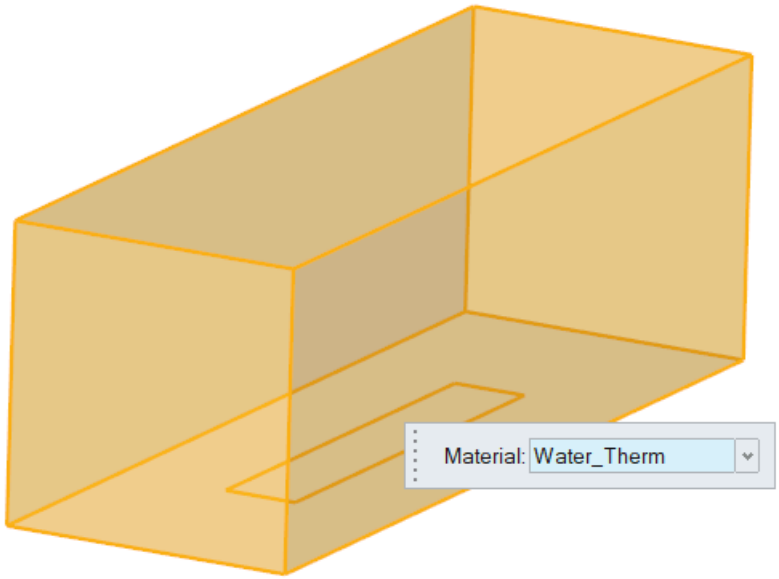


Figure 566:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 567:

2. Click on the inlet face, highlighted in the figure below.
3. In the microdialog, enter `0.39` for the Average velocity and `368.15` for the Temperature.

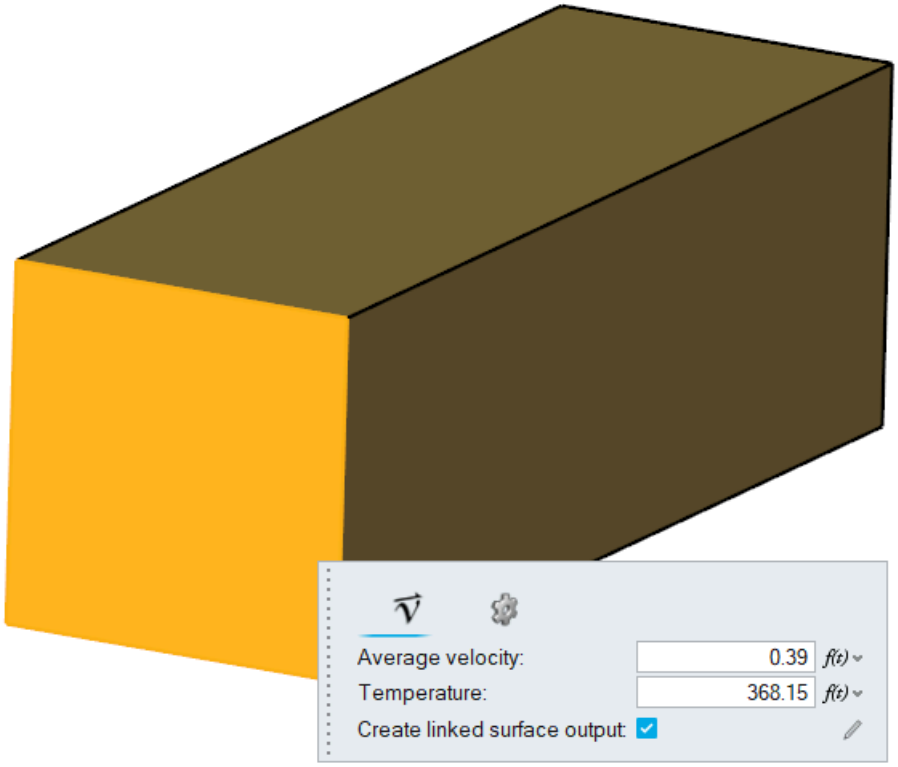


Figure 568:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 569:

6. Select the face highlighted in the figure below, set the Static pressure to `200000`, and activate the **Back flow conditions**.

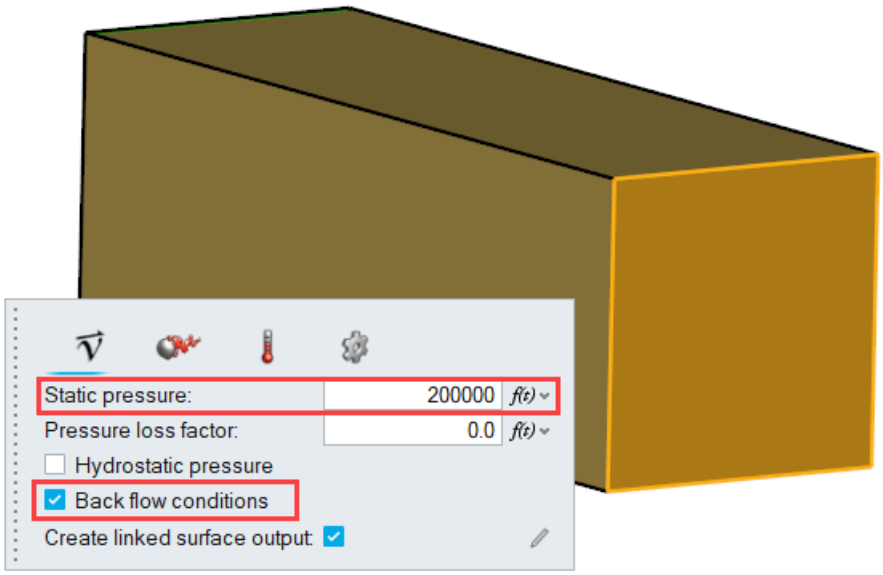


Figure 570:

7. In the Turbulence and Temperature tabs, set the back flow type to **Exiting Mass Flux Average**

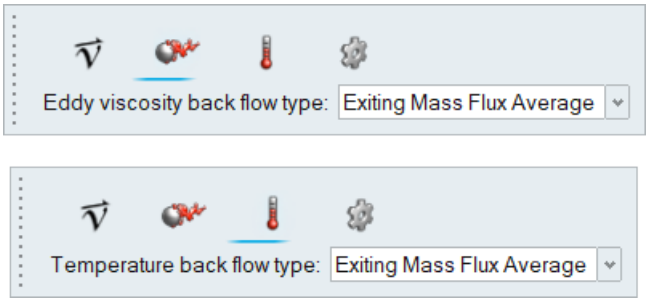


Figure 571:

8. On the guide bar, click ✓ to execute the command and exit the tool.
9. Click the **No Slip** tool.

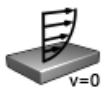


Figure 572:

10. Select the face highlighted in the figure below then click ▸ on the guide bar.

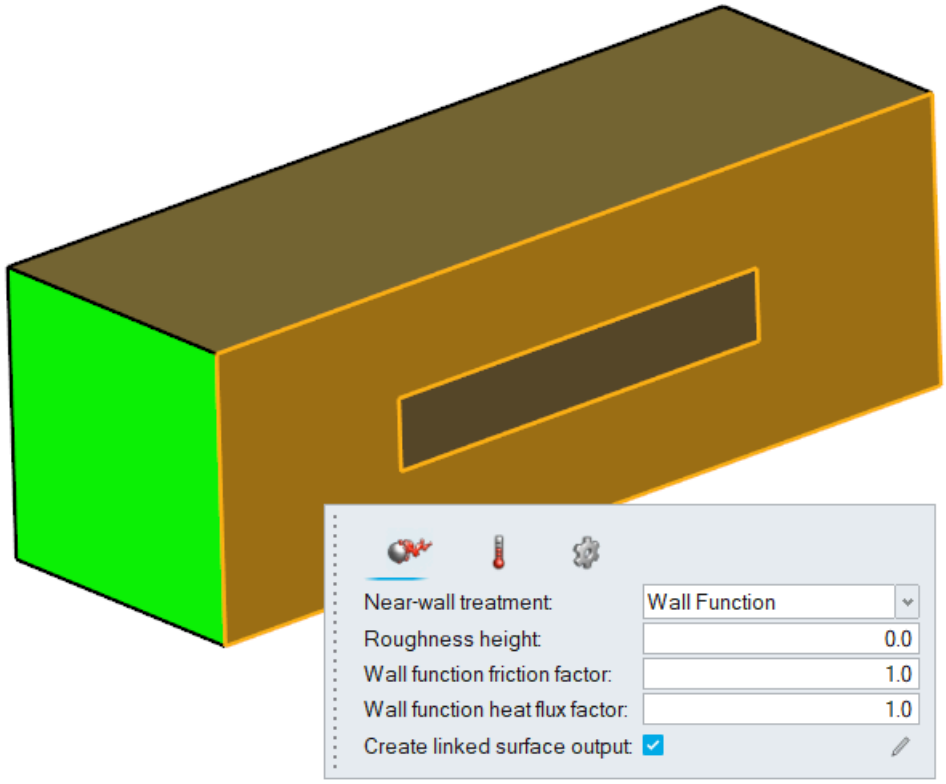


Figure 573:

11. Select the next face highlighted in the figure below.

12. In the microdialog, click the **Temperature** tab, set the Thermal boundary condition to **Temperature** and set the temperature value to `403.15`.

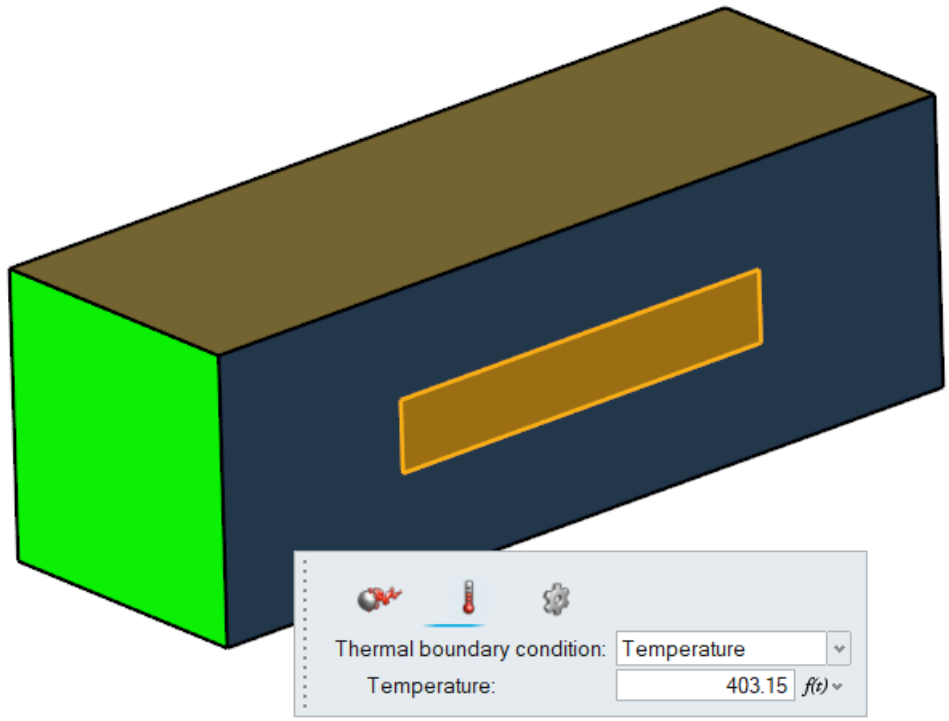


Figure 574:

13. Click ▸ on the guide bar.
14. In the Boundaries legend, double-click on **Wall 1** and rename it to `Heated_Wall`.
15. Double-click on **Wall** and rename it to `Bottom`.

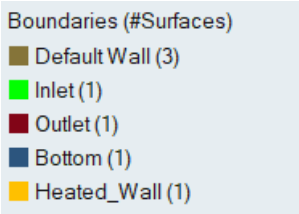


Figure 575:

16. Click ✓ on the guide bar.
17. Save the model.

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

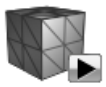


Figure 576:

2. In the **Meshing Operations** dialog, check that the Mesh growth rate is set to `1.3`.

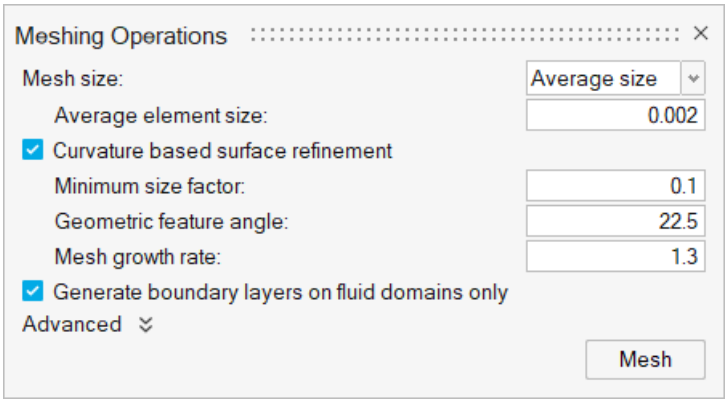


Figure 577:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   > **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 578:

   The **Launch AcuSolve** dialog opens.
2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial conditions** and enter the values as shown below to define the initial conditions.

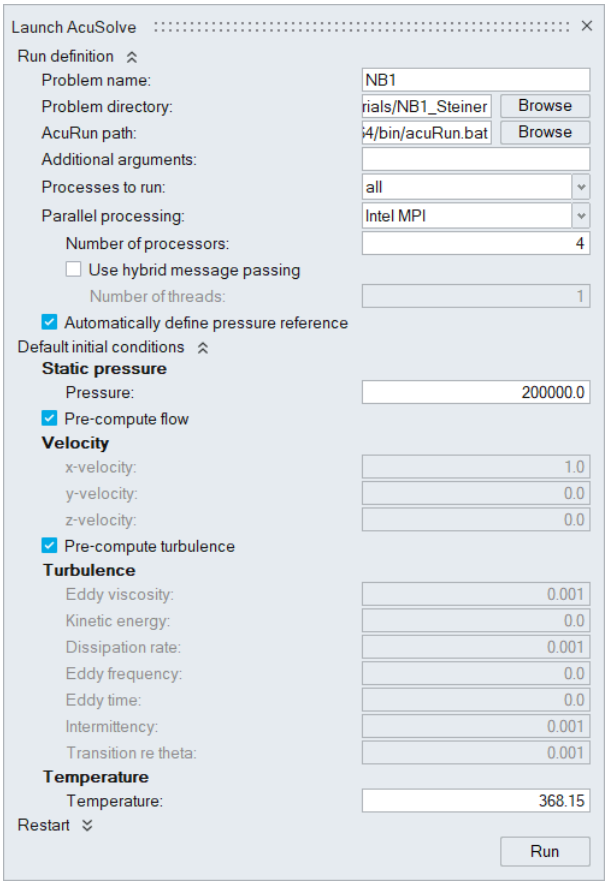


Figure 579:

5. Click **Run** to launch AcuSolve.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 580:

3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

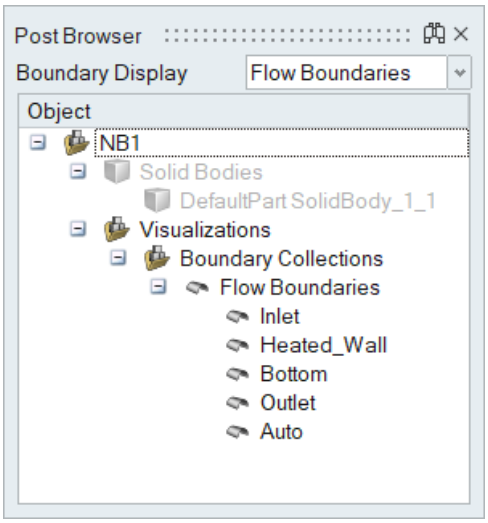


Figure 581:

4. To check the temperature contours on the **Heated_Wall** and **Bottom** surfaces, right-click on those surfaces in the Post Browser and select **Isolate**.
5. In the Post Browser, right-click on **Heated_Wall** and **Bottom** again and select **Edit**. A surface coloring microdialog opens.
6. Set the Display option to **Temperature**.

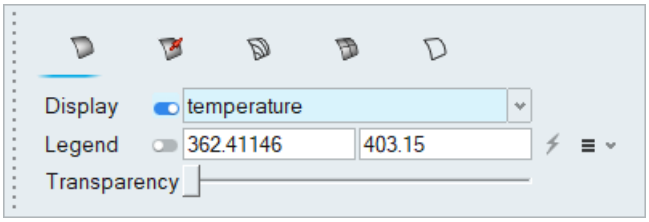


Figure 582:

7. Activate the **Legend** radio button then click ≡ and set the legend properties as shown below.

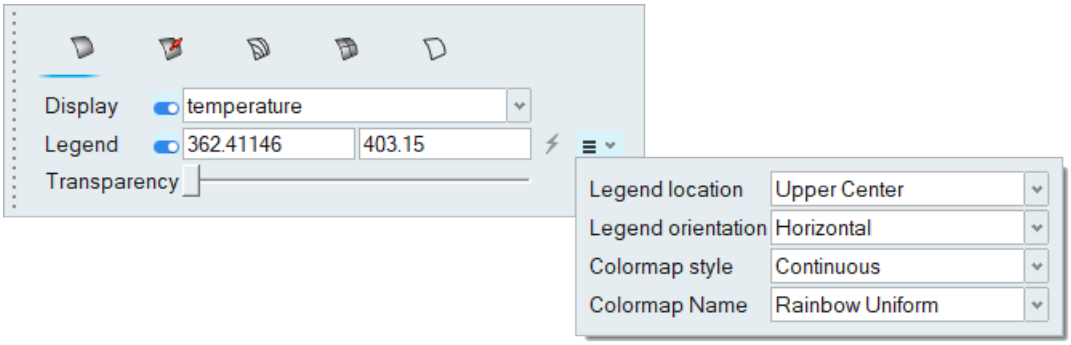


Figure 583:

The temperature contours are displayed on the model.

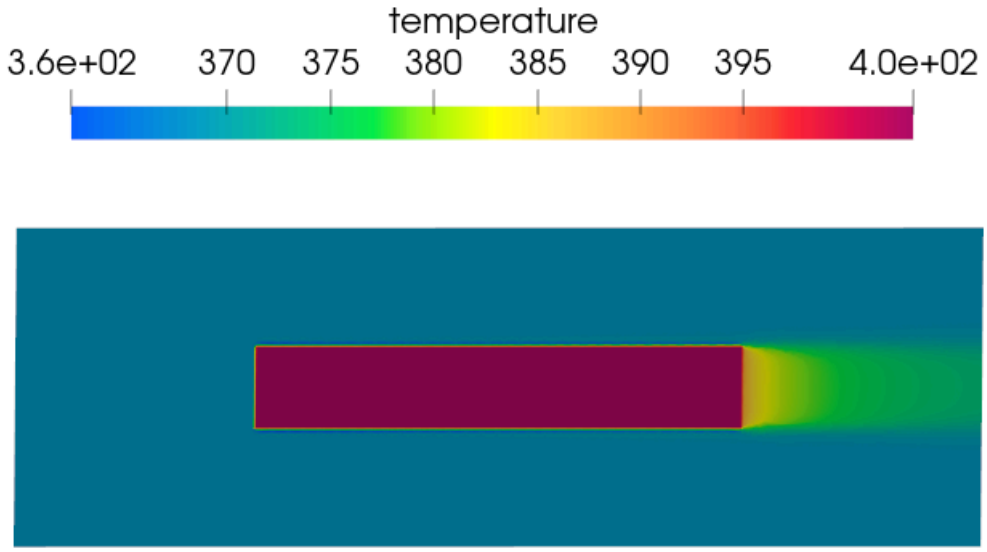


Figure 584:

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a single-phase nucleate boiling using HyperMesh CFD. You started by opening the HyperMesh input file with the geometry and then defined the simulation parameters and flow boundary conditions. Once the solution was computed, you used HyperMesh CFD Post to create the contours of temperature.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/351zla083n

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

This tutorial provides instructions for modeling a two-phase nucleate boiling in a pipe using the Algebraic Eulerian model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3311_TwoPhaseNB.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 585. It consists of a channel with a heated wall at the bottom. The temperature of the wall is selected to onset the nucleate boiling at the heated wall.

Water at 2 bar pressure and 95 ℃ temperature enters the inlet at an average velocity of 0.39 m/sec and passes through the heated wall which is maintained at 130 ℃.

The pre-heated air enters the inlets and heat is transferred to the fluid from the walls. The heat causes sub-cooled boiling to occur in the region close to the wall and leads to formation of bubbles at nucleation sites.

The heat transfer in this regime is basically dominated by two effects, the macro convection due to the motion of the bulk liquid and the latent heat transport associated with the evaporation of the liquid micro-layer between the bubble and the heated wall.

The Algebraic Eulerian (AE) model is used to simulate the heat transfer and momentum exchange between a carrier field and a dispersed field.

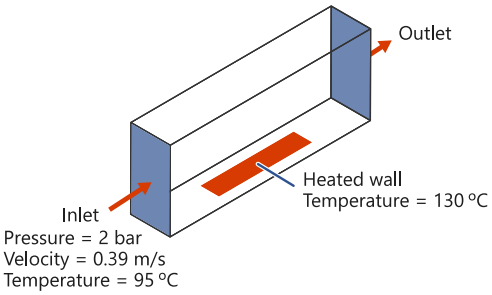


Figure 585: Schematic of Channel

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 586:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3311_TwoPhaseNB.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `NB2` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `NB2_Steiner` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate

icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 587:

Set Up Flow

Verify and Create Materials

1. From the Flow ribbon, click the **Material Library** tool.

   Figure 588:

   The **Material Library** dialog opens.
2. Under Settings, click **Fluid**, then click the **My Materials** tab.
3. Verify that the material Water_NB has the properties shown below.

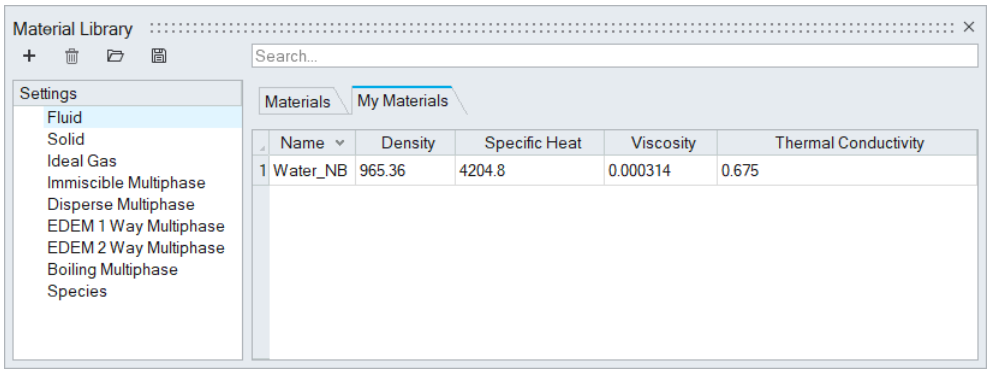


Figure 589:

4. Under Settings, click **Boiling Multiphase**, then click the **My Materials** tab.
5. Click **+** to create a new material.
6. Name the material `Boiling_NB` and verify the following properties in the microdialog.

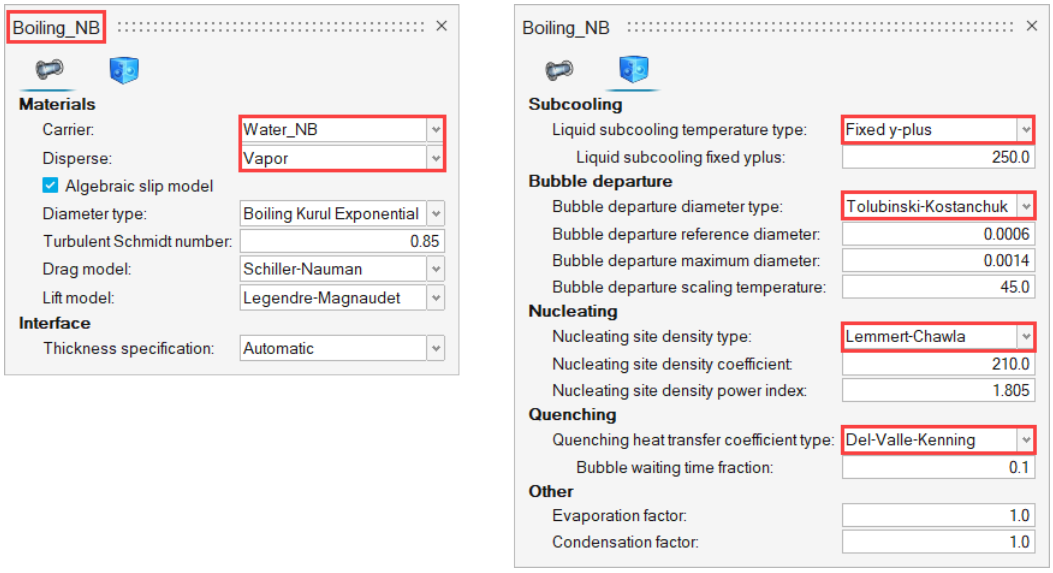


Figure 590:

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

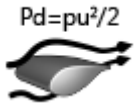


Figure 591:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Boiling** and the Boiling material to **Boiling_NB**.
   c) Ensure that the Time step size is `0.01`.
   d) Select **Spalart-Allmaras** as the Turbulence model.
   e) Set the Gravity value to `-9.81` in the z direction.
   f) Set the Pressure scale to **Gauge** then click ≡ besides the drop-down and set the Absolute pressure offset to `200000`.

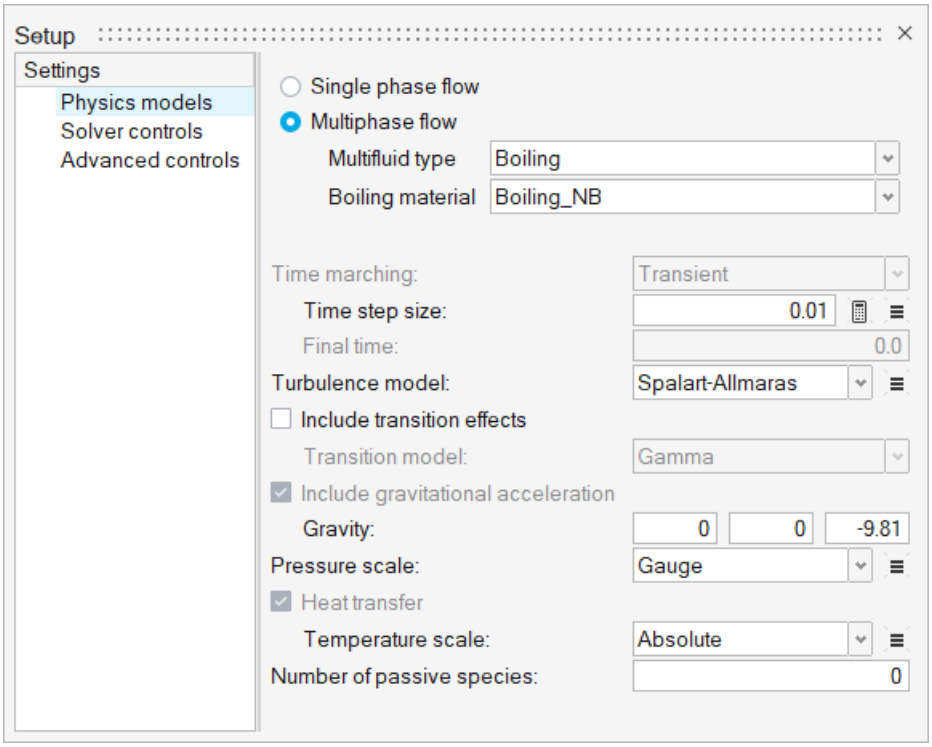


Figure 592:

3. Click the **Solver controls** setting and verify that:

a) Minimum stagger iteration is set to `2`.

b) Maximum stagger iteration is set to `4`.

c) Transient maximum step is set to `40`.

d) The **Flow**, **Temperature**, **Multifluid**, and **Turbulence** checkboxes are activated.

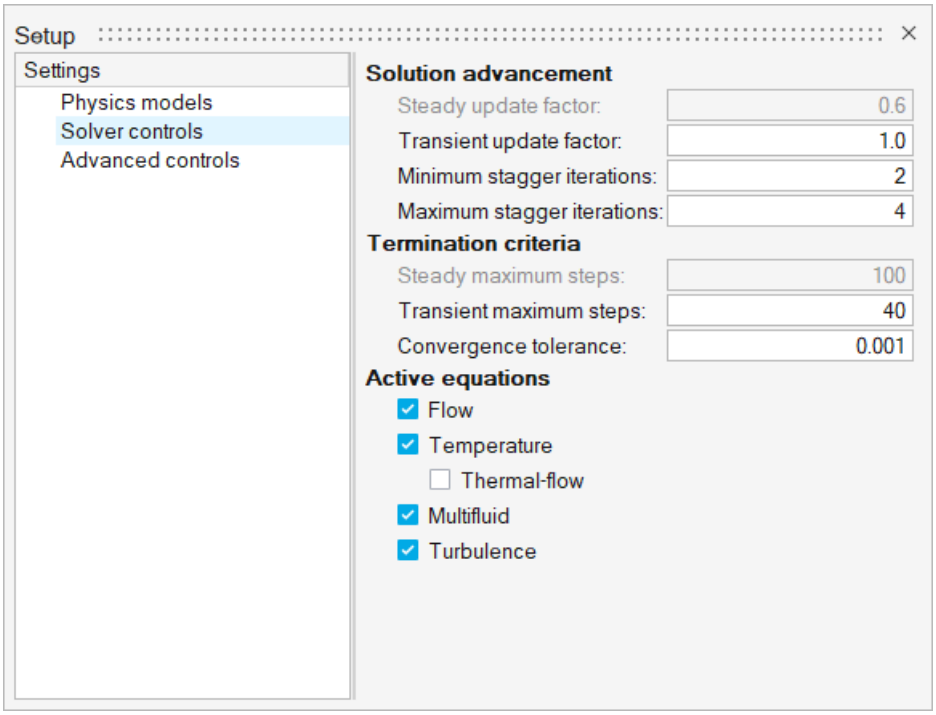


Figure 593:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 594:

2. Select the model solid.
3. Select **Boiling_NB** from the Material drop-down menu.

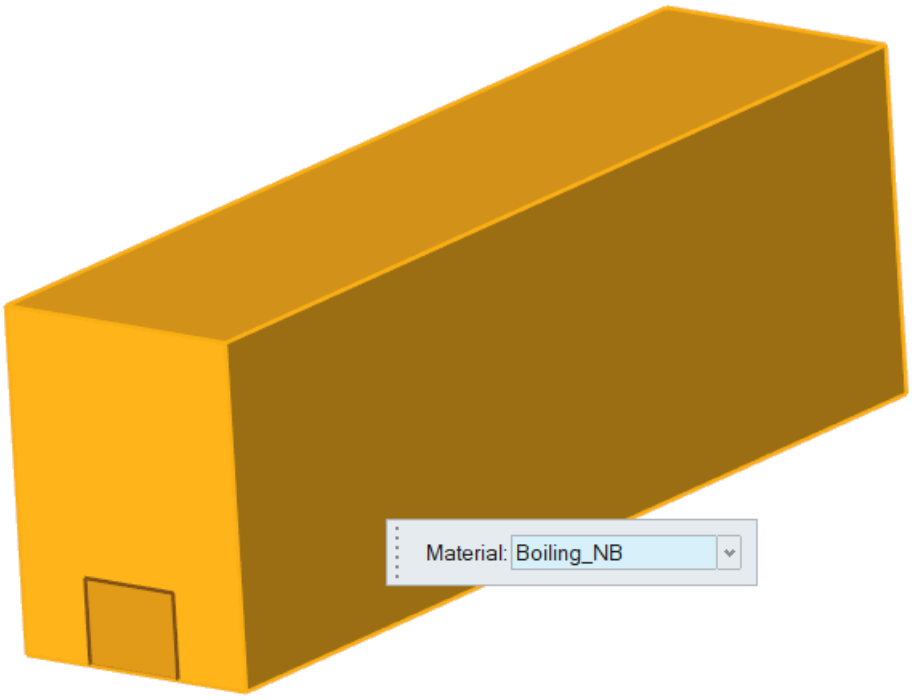


Figure 595:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 596:

2. Click on the inlet face, highlighted in the figure below.
3. In the microdialog, enter `0.39` for the Average velocity and `368.15` for the Temperature.

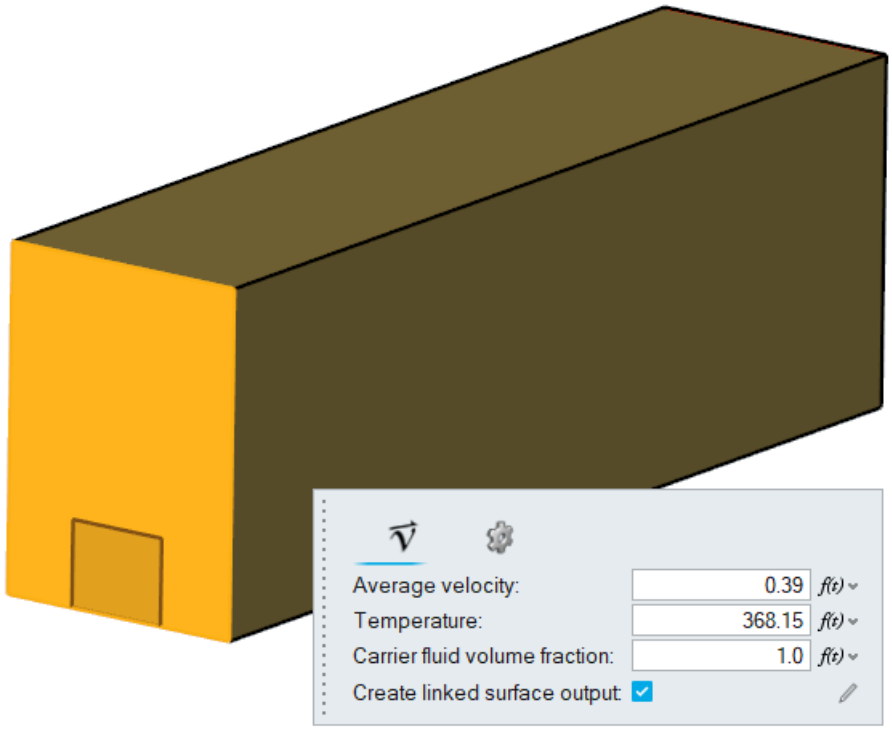


Figure 597:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 598:

6. Select the face highlighted in the figure below then click ✓ on the guide bar.

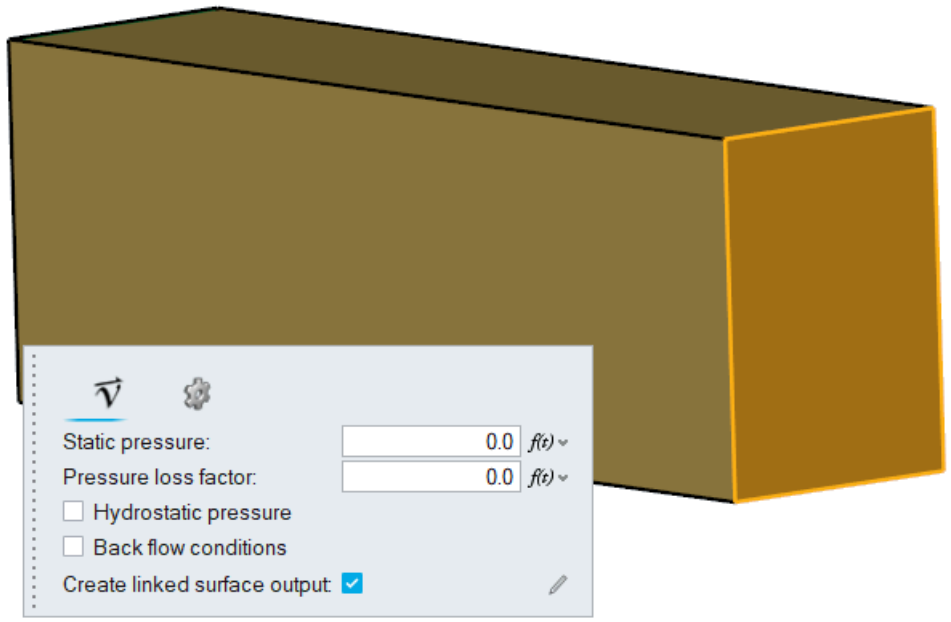


Figure 599:

7. Click the **No Slip** tool.

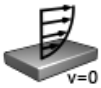


Figure 600:

8. Select the face highlighted in the figure below to create the Heated_wall face.

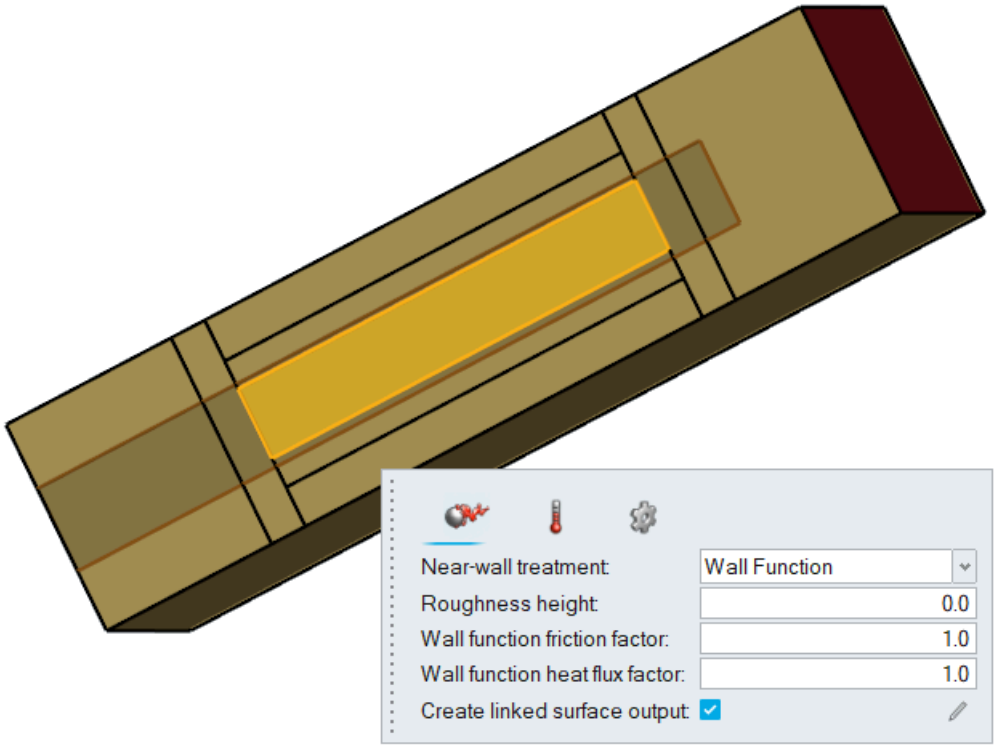


Figure 601:

9. Click the Temperature tab in the microdialog, set the Thermal Boundary condition to **Temperature**, and set the Temperature value to `403.15`.

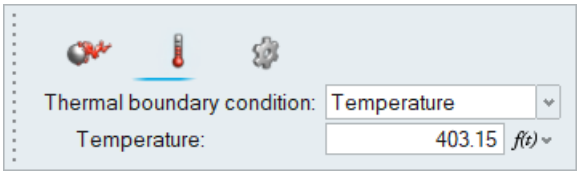


Figure 602:

10. On the guide bar, click ▸ to execute the command and remain in the tool.

11. In the Boundaries legend, right-click on **Wall**, select **Rename**, and enter `Heated_wall`.

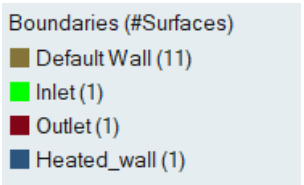


Figure 603:

12. Select the 8 highlighted surfaces shown in the figure below to create the Bottom_wall faces then click ▸ on the guide bar.

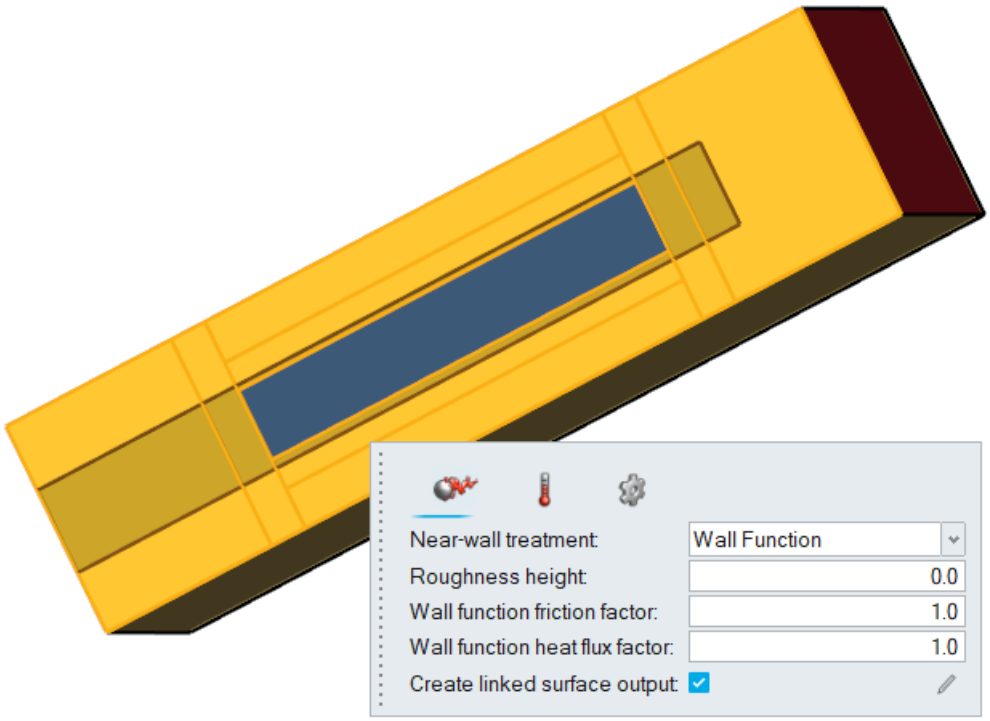


Figure 604:

13. In the Boundaries legend, rename Wall to `Bottom_wall`.

14. From the Solution ribbon, click the **Field** tool.

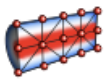


Figure 605:

The **Field Output** dialog opens.

15. Set the Time step interval to `10`.

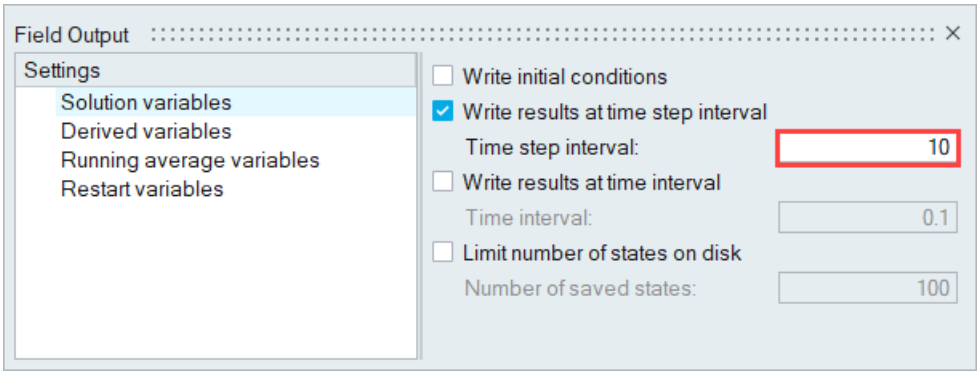


Figure 606:

16. Save the model.

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

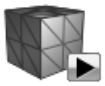


Figure 607:

2. In the **Meshing Operations** dialog, set the Average Element size to `0.004` (if not set already).

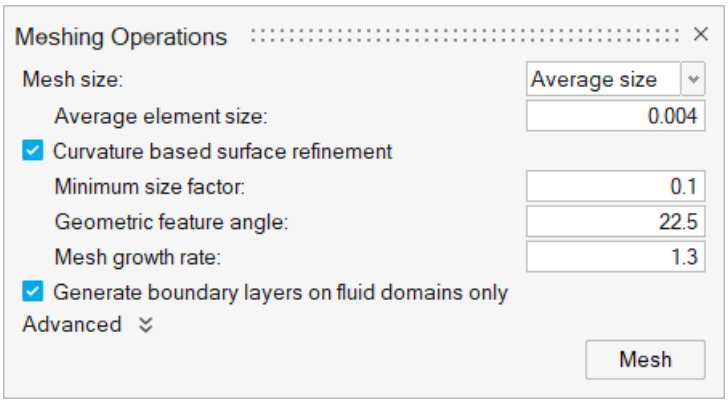


Figure 608:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 609:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Check the box for **Use hybrid message passing** and set the number of threads to the same as the number of processors.
5. Expand **Default initial conditions** and enter the values as shown below to define the initial conditions.

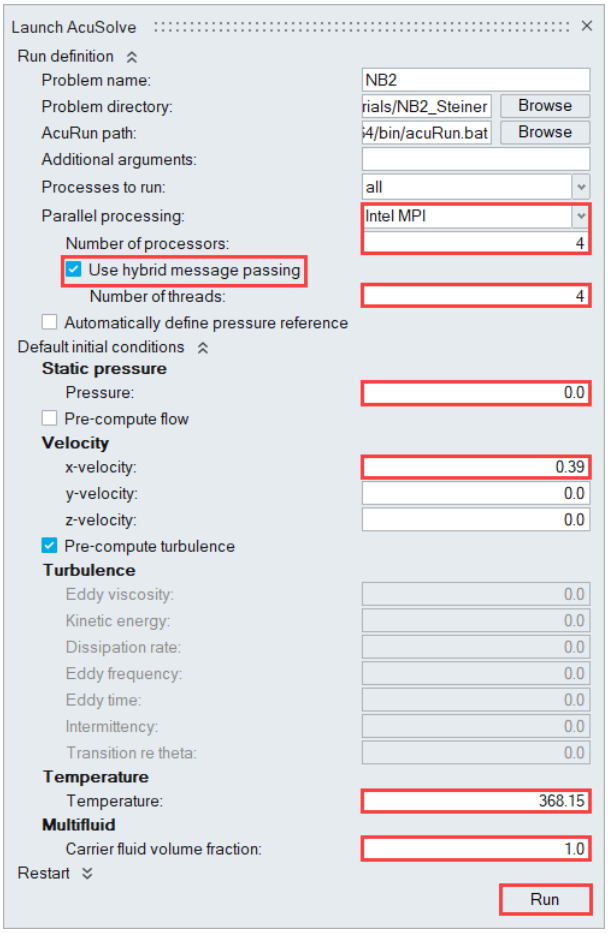


Figure 610:

6. Click **Run** to launch AcuSolve.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing.
   The solid and all the surfaces are loaded in the Post Browser.

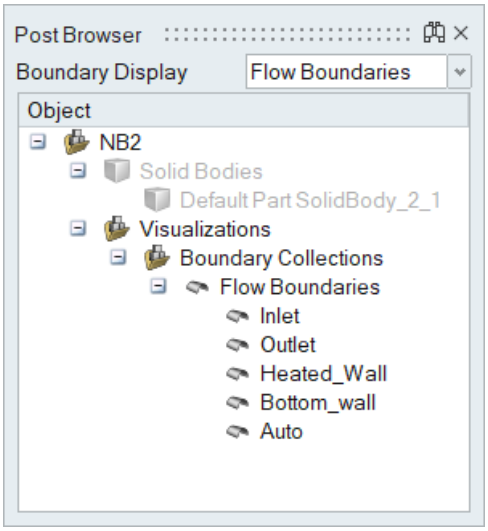


Figure 611:

4. To check the contours of volume fraction of vapor on the **Heated_Wall** and **Bottom_wall** surfaces, right-click on those surfaces in the Post Browser and select **Isolate**.
5. From the Post ribbon, click the Boundary Groups tool.

Figure 612:

6. Select all the surfaces on the model.
7. In the microdialog, set the Display option to **volume fraction vapor**.
8. Activate the **Legend** radio button then click ≡ and set the legend properties as shown below.

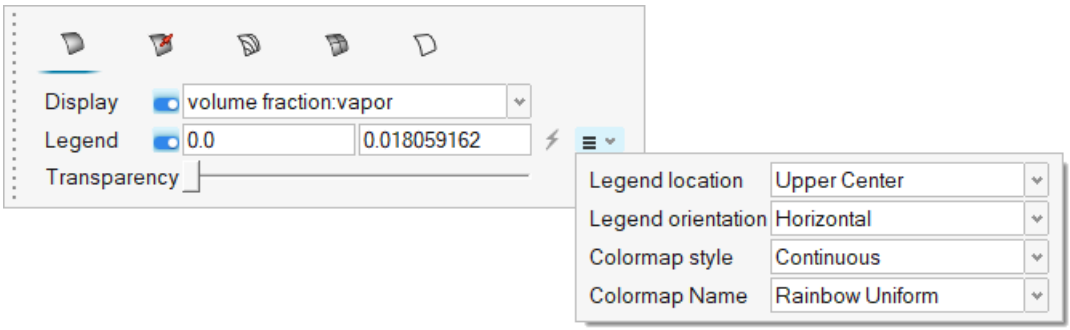


Figure 613:

9. Click ✓ on the guide bar.
10. Move the slider at the bottom of the modeling window to 4/4.
    The volume fraction of vapor contours are displayed on the model.

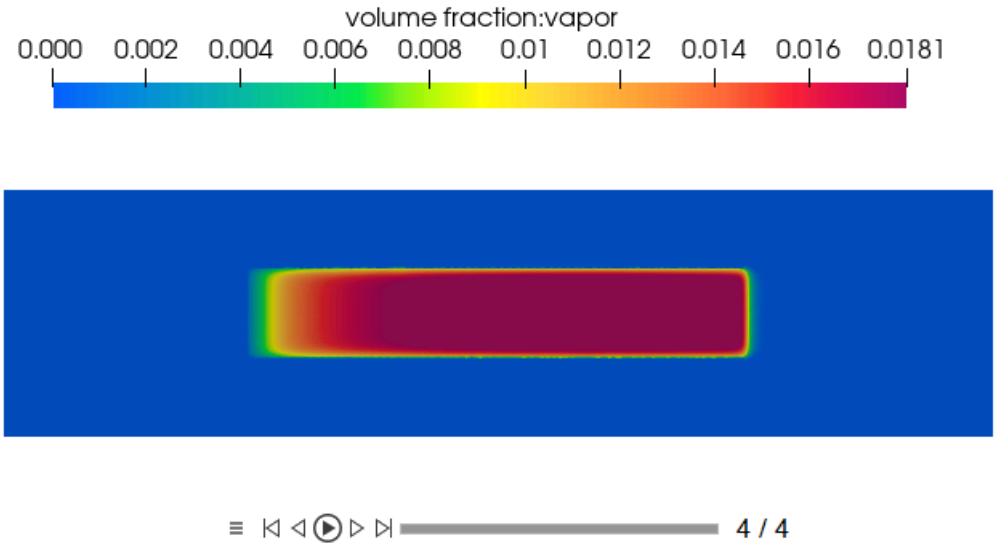


Figure 614:

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a two-phase nucleate boiling using HyperMesh CFD. You started by opening the HyperMesh input file with the geometry and then defined the simulation parameters and flow boundary conditions. Once the solution was computed, you used HyperMesh CFD Post to create the contours of volume fraction of vapor.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/bm192xw8fe

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3400 AcuSolve -Flux Integration

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/2jdgan01zj

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T:3401 Electric Motor Analysis using AcuSolve - Flux One-Way Coupling

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

This tutorial works directly from the solution setup. Before you begin download the SimLab database

Tutorial Video

https://fast.wistia.net/embed/iframe/gykjydrs5o

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3500 Electric Potential - Automotive Fuse

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/lvyhq8j05g

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3510 Battery Thermo – Electric: 2S2P Module

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/od4dhjudij

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3511 Battery Thermo-Electric Pouch Cell

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/ov106ttwj8

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3512 Battery Thermal Runaway

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/6oarm1s8z4

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3513 ARC-Based Thermal Runaway: 1S1P Module

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/xloxqvd3to

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 3600 Melting of Diesel Exhaust Additive within an Enclosed Tank

Perform this tutorial using the following user interface(s):

HyperMesh CFD

HyperMesh CFD

Tutorial Level: Advanced

This tutorial provides the directions for setting up, solving, and post-processing results for a simulation that models the approximated melting of the common diesel fuel exhaust additive that is contained within a notional tank. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T3600_UreaTankMelting_TemperatureBasedEffectiveness.hm`

Problem Description

In this tutorial, a diesel exhaust additive, commonly known as Urea, is initially subjected to freezing conditions dictating that the fluid is frozen, T(Initial)=263.15 K. The fluid is then heated above its defined melting point by imposing a fixed thermal condition on the heating element. As the temperature within the enclosure exceeds the melting temperature of the material, the porosity effectiveness of the fluid decreases to 0.0, allowing the fluid to move under natural convection currents. Eventually the entire fluid volume becomes melted, reaching the temperature of the heating element. The time to melt is computed for the specified heating element temperature of 313.15 K. Note that the interface between the solid/fluid material is captured because of the material definition as defined with a linear temperature gradient spanning 2 degrees at the melting front.

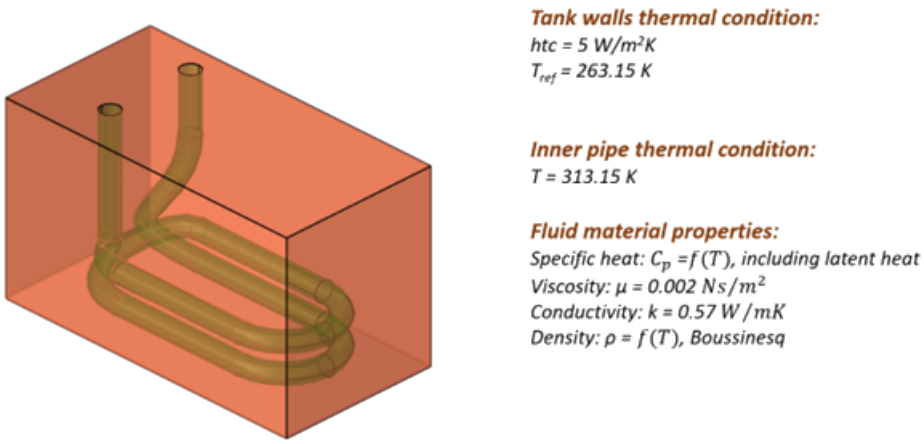


Figure 615:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 616:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3600_UreaTankMelting_TemperatureBasedEffectiveness.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `MeltingTank` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `MeltingTank` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate

icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 617:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

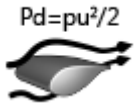
   

   Figure 618:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.5` seconds.
   c) Click ≡ beside the time step field and select **Modify time integration settings**. Set the Time integration order to **First**.

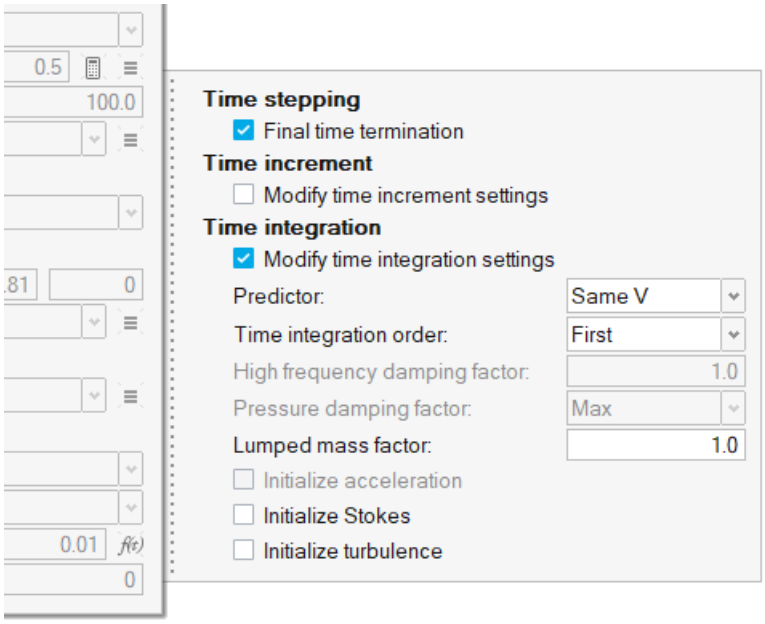


Figure 619:

d) Set the Final time to `100.0` seconds.
e) Select **Spalart-Allmaras** as the Turbulence model.
f) Activate **Include Gravitational Acceleration** and set it to `[0.0, -9.81, 0.0]` m/s^2.
g) Set the Pressure scale to **Absolute**.
h) Active **Heat transfer**.
i) Set the Temperature scale to **Absolute**.

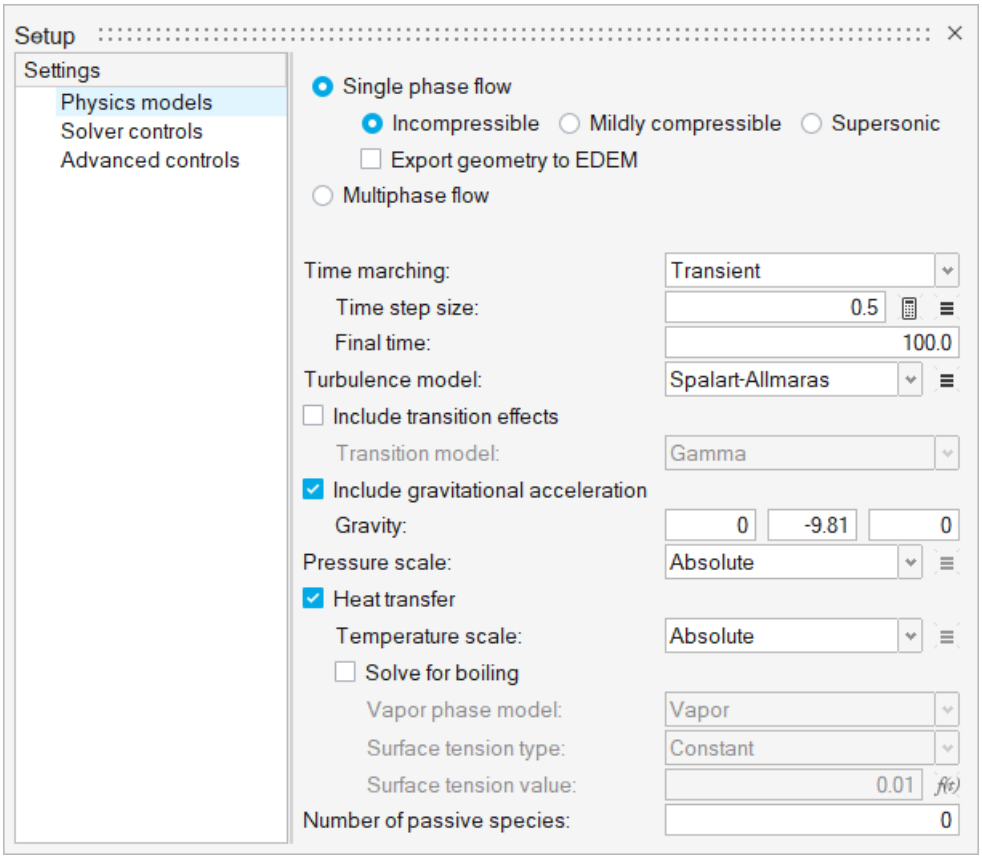


Figure 620:

3. Under the Advanced controls setting, set the minimum stagger iterations to `1` and the maximum to `4`.

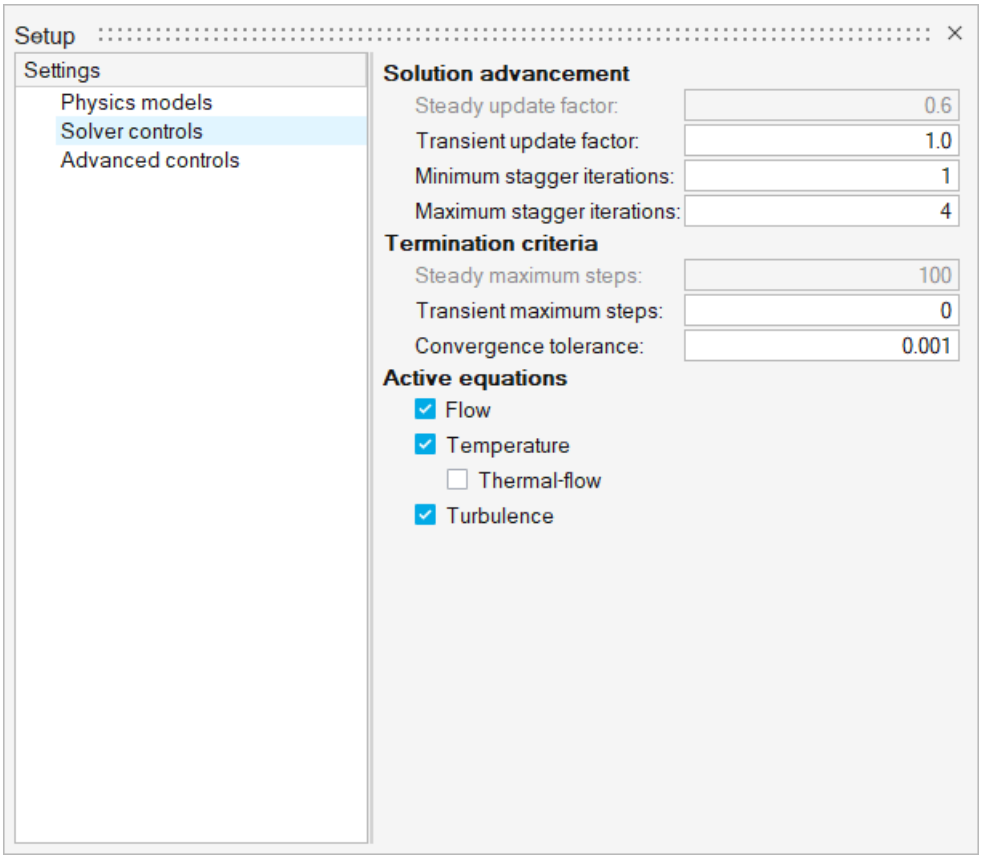


Figure 621:

Assign Material Properties

1. From the Flow ribbon, click the **Material Library** tool.

Figure 622:

2. Under Settings, click **Fluid**, then click the **My Materials** tab.
3. Click **+** to create a new material.
4. Name the material `Urea`, set the density (ρ) type to **Boussinesq** and enter Density (kg/m^3) and Reference temperature (K) values as shown below.

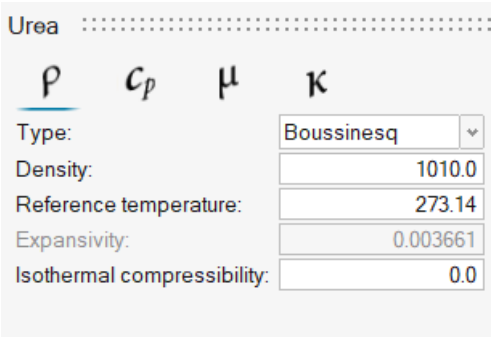


Figure 623:

5. Click the **Specific Heat** tab and set the Type to **Piecewise Linear Enthalpy**.
6. Verify that the Curve fit variable is set to **Temperature**.
7. Set the Latent heat type to **Constant**.
8. Enter the values for Latent heat (J), Latent heat temperature (K), and Latent heat temperature interval (K) by referring to the figure below.
9. Click three times to add three rows to the table and then enter the table values according to the figure below.

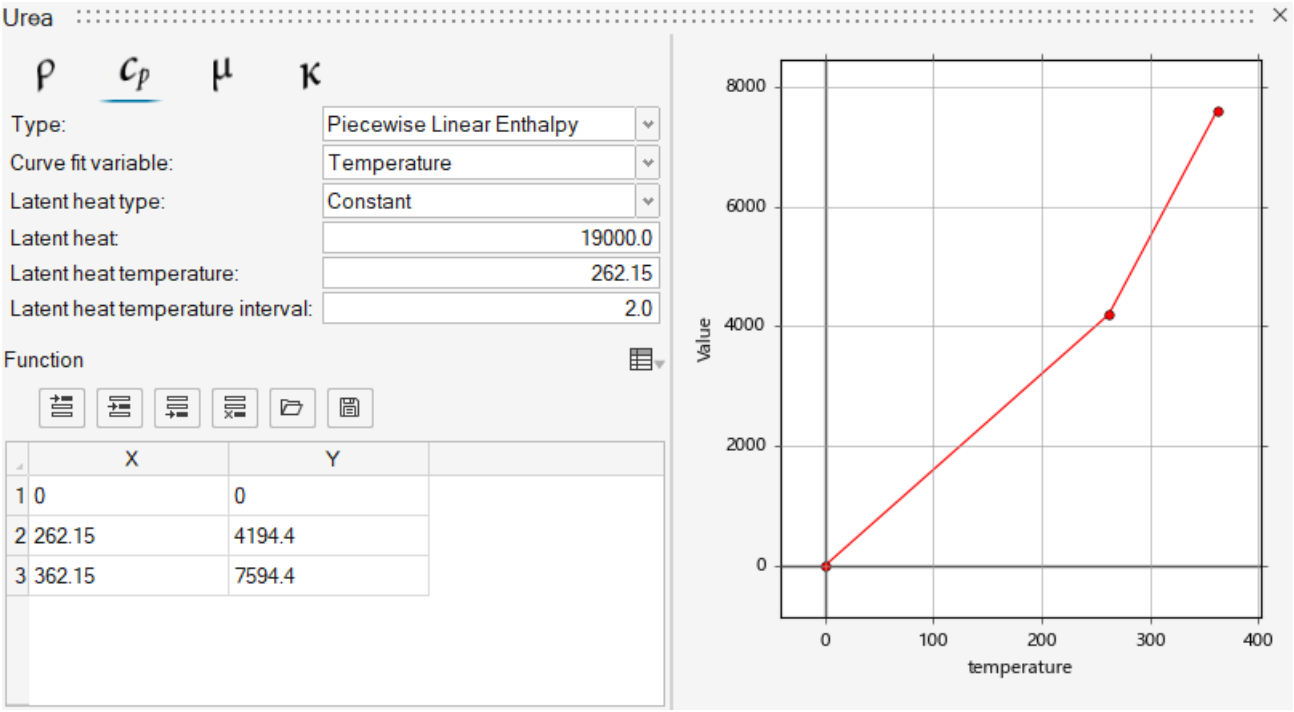


Figure 624:

10. Click the **Viscosity** tab and set μ (viscosity) to `0.002` kg/m-sec.
11. Click the **Conductivity** tab and set k (conductivity) to `0.57` W/m-K.
12. Close the material creation dialog.

13. From the Flow ribbon, click the **Material** tool.

Figure 625:

14. Assign the Fluid material – Urea – to the tank volume , as shown in the figure below.

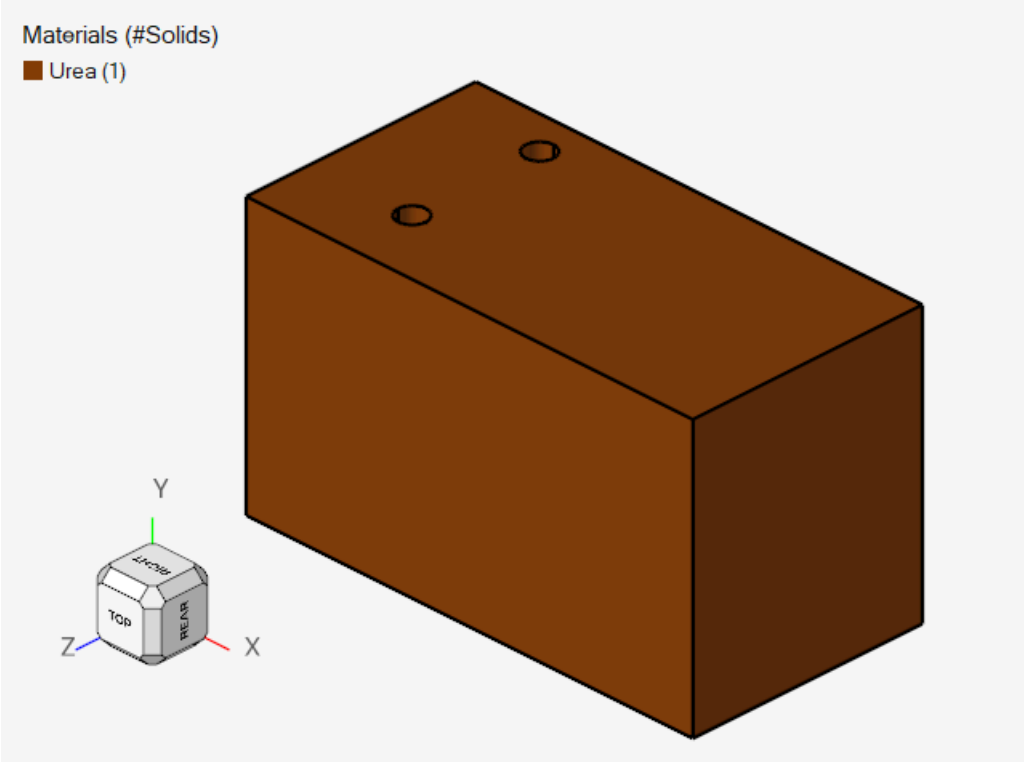


Figure 626:

Define the Porous Medium

1. From the Flow ribbon, Porous tool group, click the **Cartesian Porous Media** tool.

Figure 627:

2. Select the tank volume (urea fluid body)

3. Click **Orientation** on the guide bar then choose an arbitrary location to define the Porous model orientation

Any location is acceptable for this model.

4. Assign the Porous Media values according to the following figure.

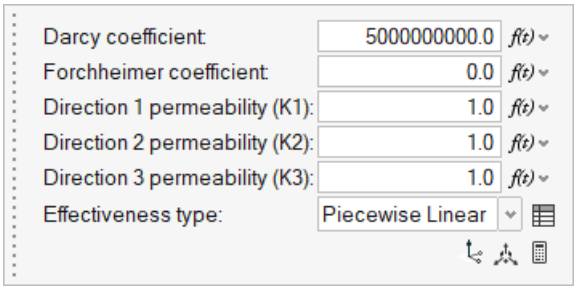


Figure 628:

5. Click besides Effectiveness type.
6. In the new dialog, click four time to add four empty rows to the table then enter values according to the figure below.

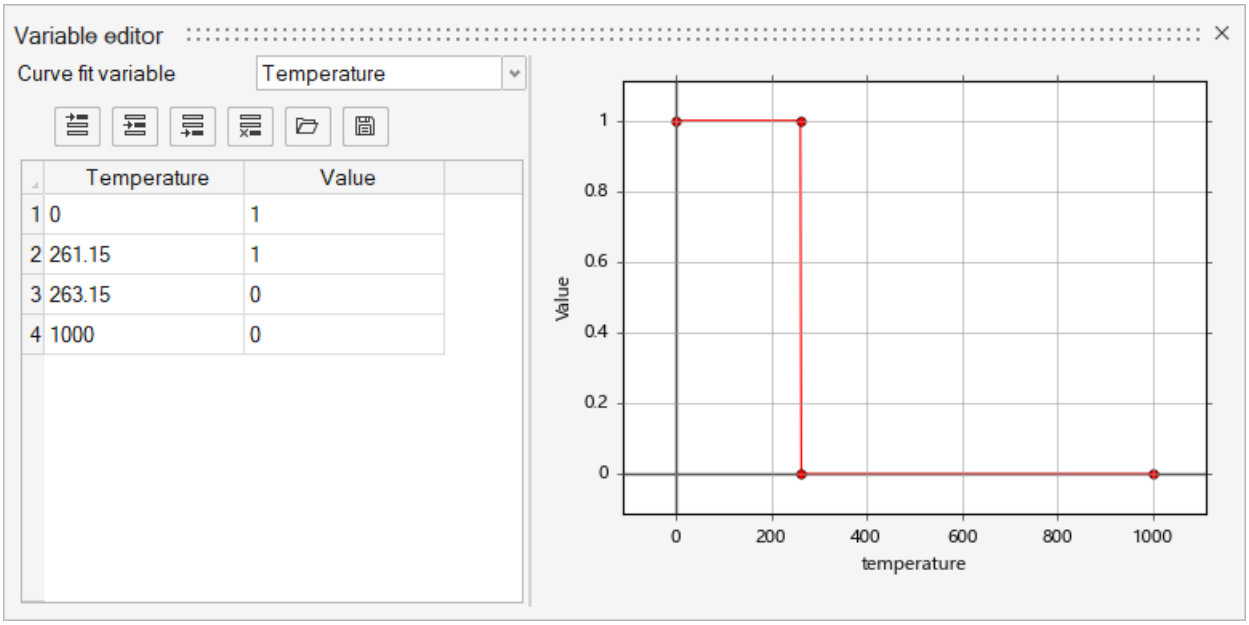


Figure 629:

7. Rename the Porous model.
   a) Double-click on **Porous** in the legend on the left side of the modeling window.
   b) Type `Additive` and press **Enter**.
8. On the guide bar, click ✓ to execute the command and exit the tool.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **No Slip** tool.

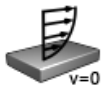


Figure 630:

2. Select the exterior faces of the plastic tank.

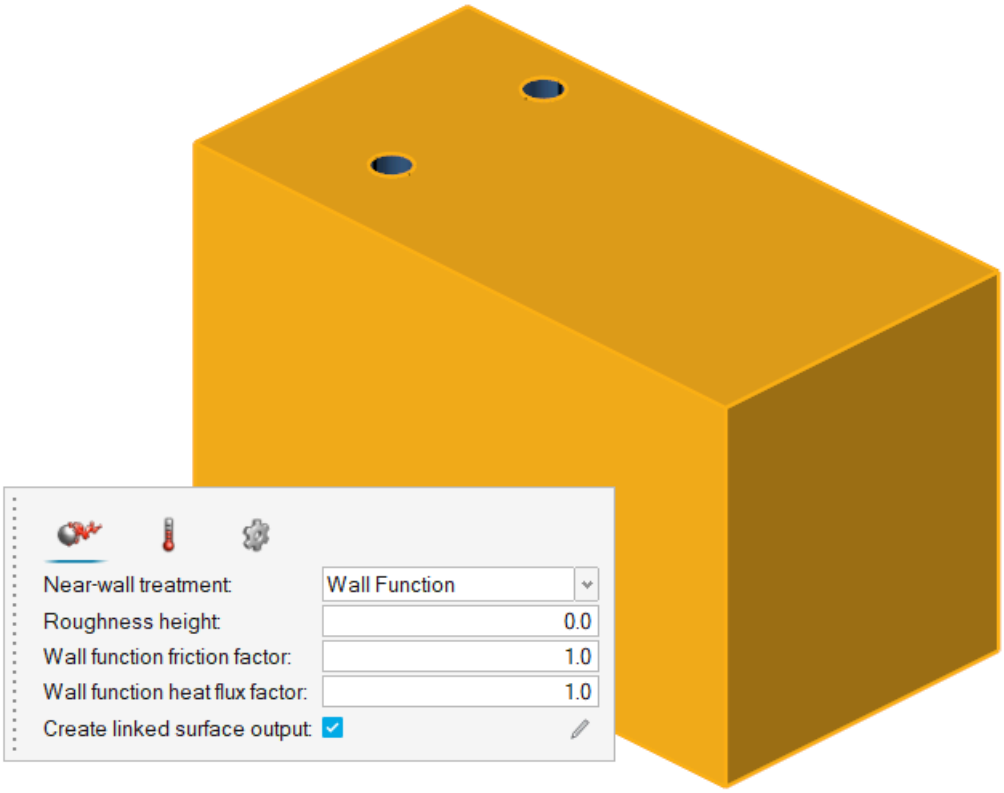


Figure 631:

3. In the microdialog, click the **Temperature** tab.
4. Verify that the Thermal boundary condition is set to **Flux**.
5. Set the Convective heat coefficient to `5.0` W/m^2K.
6. Set the Convective heat reference temperature to `263.15` K.

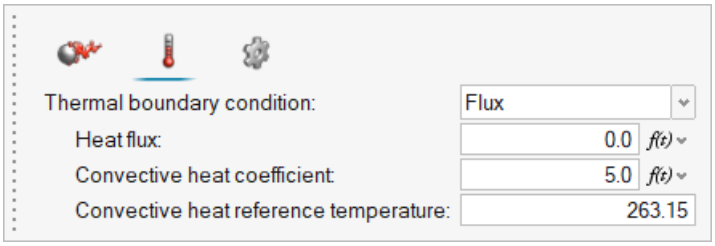


Figure 632:

7. Rename the wall boundary.
   a) Double-click on **Wall** in the legend on the left side of the modeling window.
   b) Type `Exterior_Wall` and press **Enter**.
8. On the guide bar, click ▸ to execute the command and remain in the tool.
9. Hide Exterior_Wall and select the interior faces of the heating element.
10. In the microdialog, click the **Temperature** tab.
11. Change the Thermal boundary condition to **Temperature** and set the Temperature (K) to `313.15`.

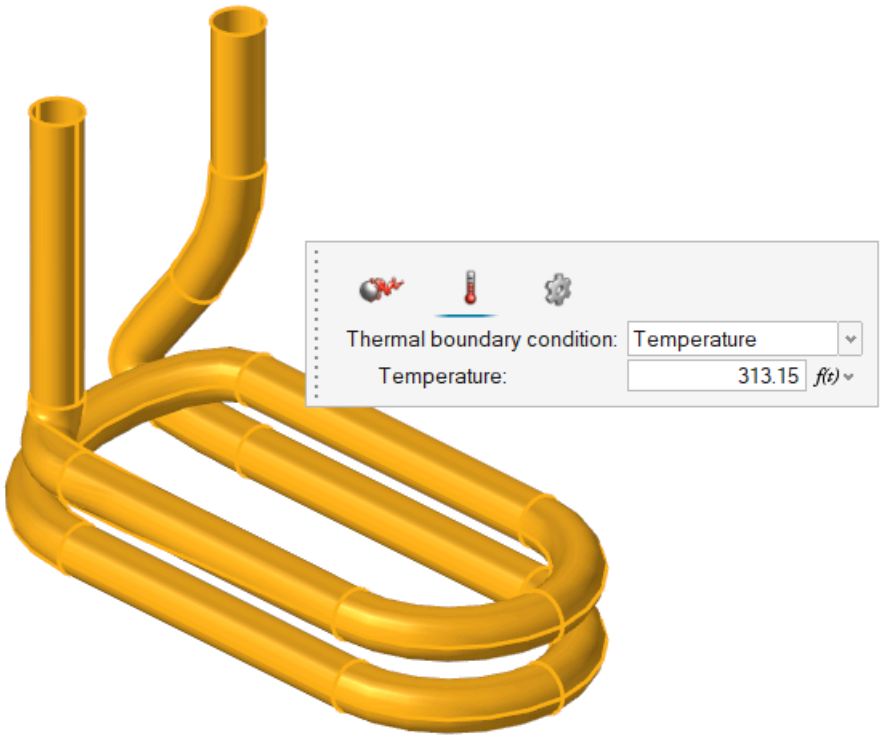


Figure 633:

12. Double-click on **Wall** in the legend and rename it to `Pipe`.
13. Click ✓ on the guide bar.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

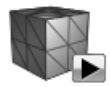


Figure 634:

2. Set the Maximum element size (m) to `0.02` and verify the geometric feature angle (degree) is set to `12.5`.

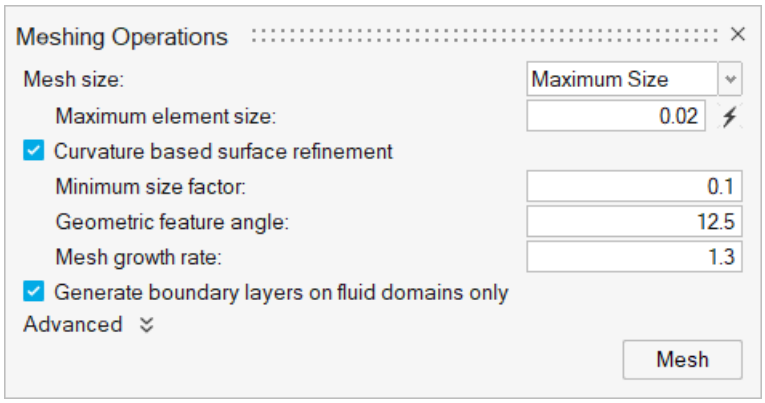


Figure 635:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Set the Field Outputs

1. Form the Solution ribbon, click the **Field** tool.

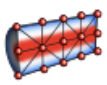


Figure 636:

2. Uncheck **Write results at time step interval**.
3. Check **Write results at time interval**.

4. Set the Time interval to `1` sec.

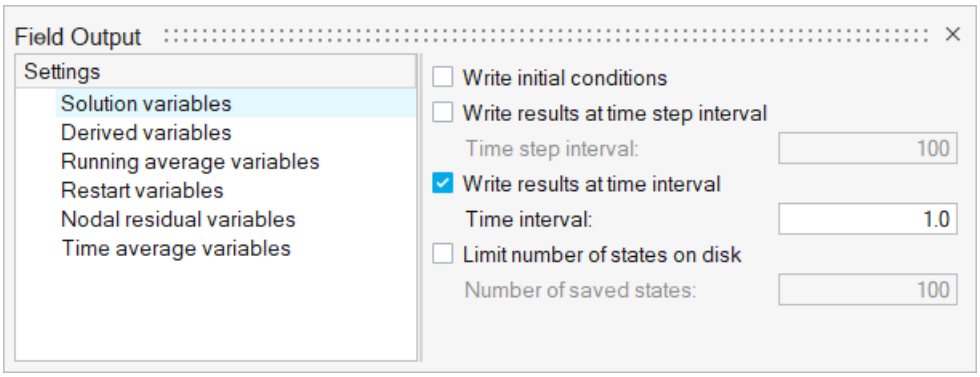


Figure 637:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 638:

   The **Launch AcuSolve** dialog opens.
2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Set the Default initial conditions as in the following figure.

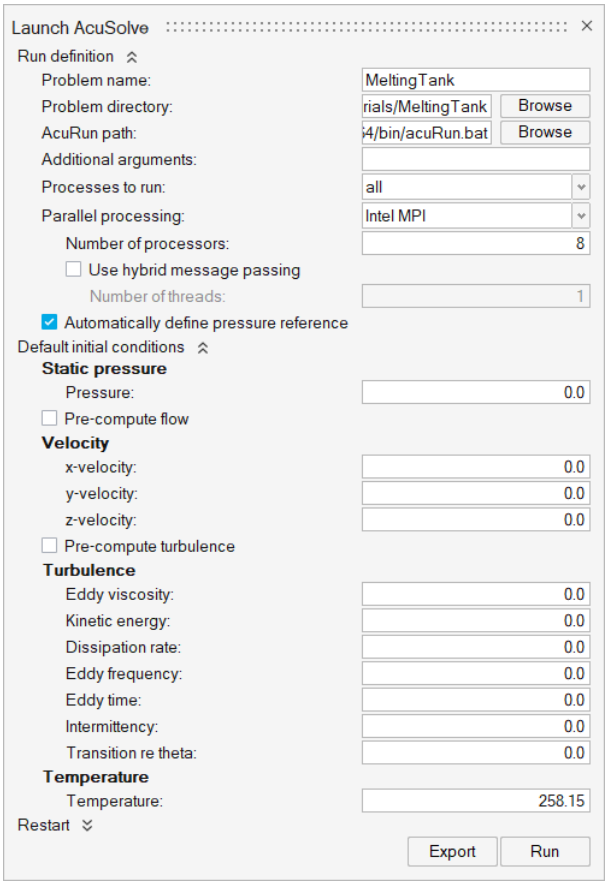


Figure 639:

5. Leave the remaining default options then click **Run** to launch AcuSolve.

Post-Process the Results

Post-Process with the Plot Tool

1. From the Solution ribbon, click the **Manage** tool.

Figure 640:

2. In the **Run Status** dialog, right-click on the AcuSolve run and select **Plot time history** to launch the **Plot Manager**.
   The residual and solution ratios for pressure, velocity, eddy viscosity, and temperature are shown.

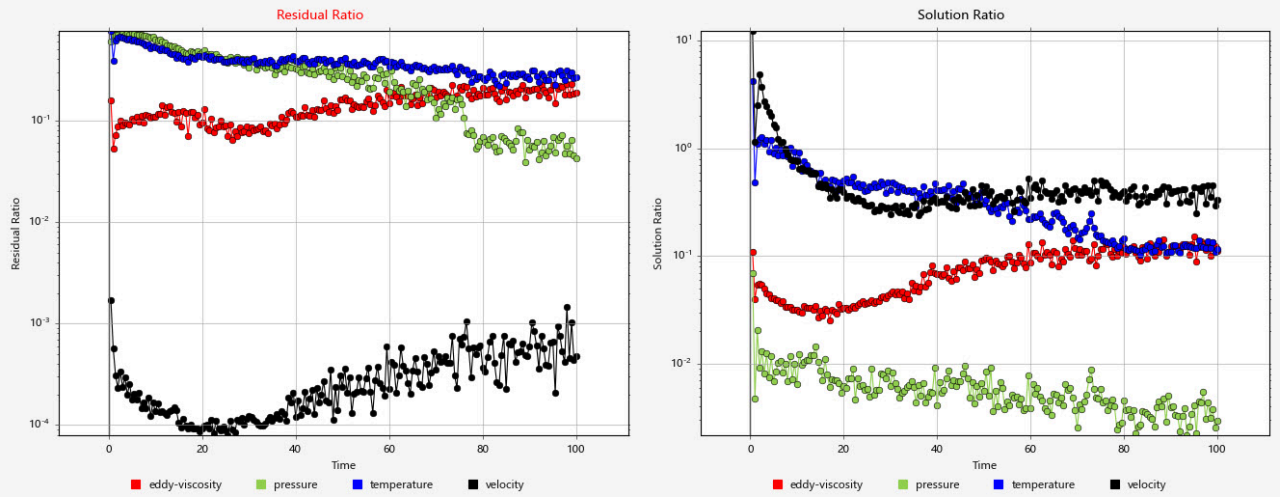


Figure 641:

3. Click + to define a new plot.
4. Expand **Integrated Surface Output** > **Temperature** and select **heat flux**.
5. Select **Pipe – Output** to plot the heat flux entering the domain from the heating element.

> **Note:** The value stabilizes near the end of the simulation, indicating that the model is approaching the prescribed thermal boundary conditions.

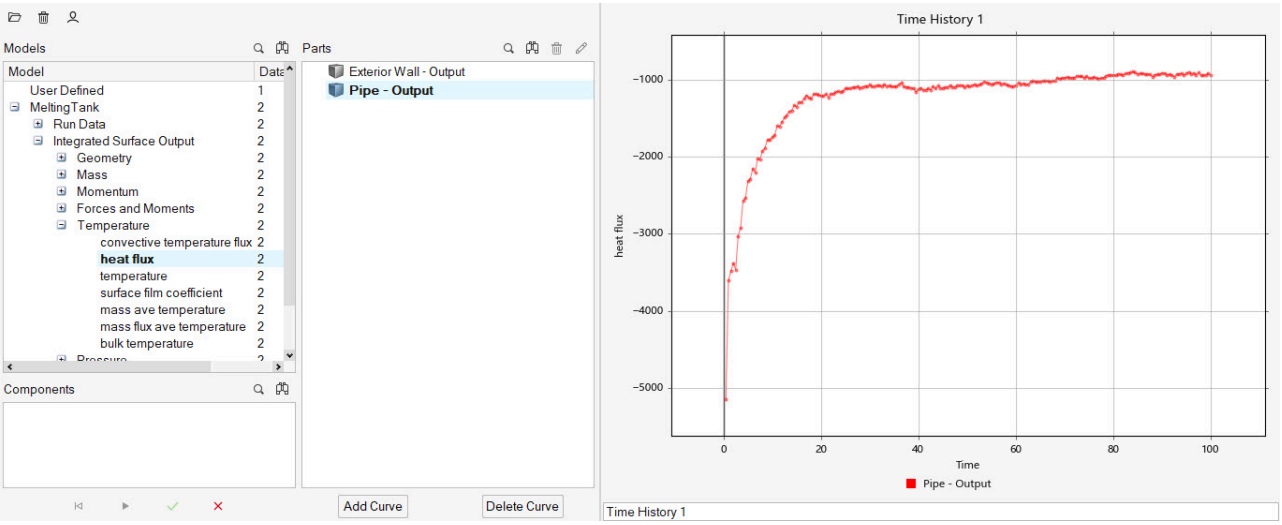


Figure 642: Integrated heat flux entering the domain from the heating element surface

6. Expand the **Integrated Surface Output** > **Temperature** > **temperature**.
7. Select the **Exterior Wall – Output** to plot the temperature increase of the inside wall of the tank.

Note: The value is not stabilized by the end of the simulation, indicating that the model has not reached the final temperature to balance the prescribed thermal boundary conditions.

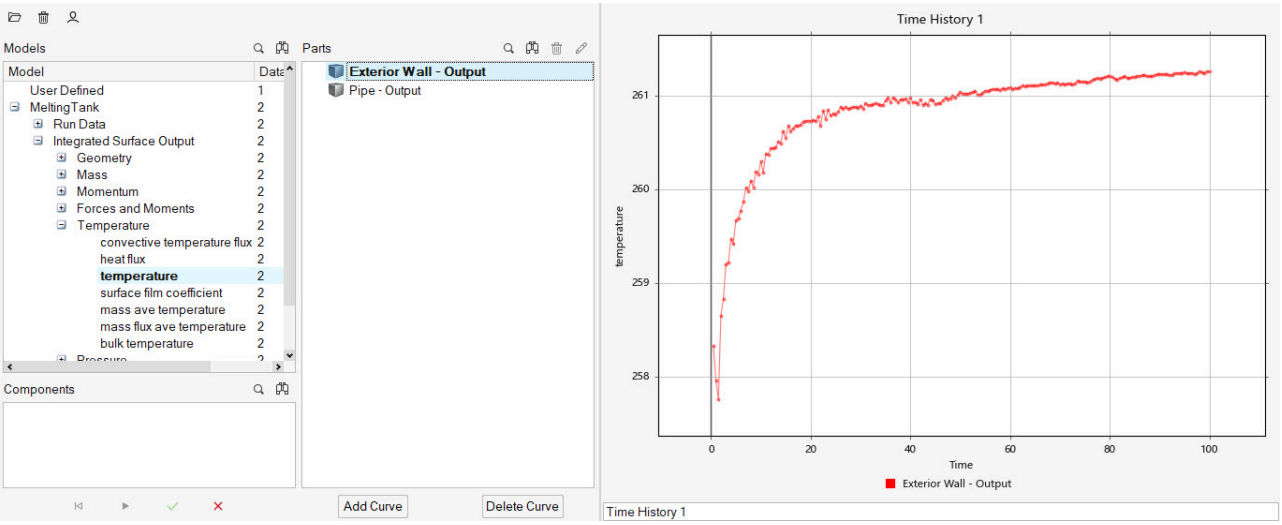


Figure 643: Integrated temperature on the interior surface of the tank wall

Post-Process with HyperMesh CFD Post

1. In the **Run Status** dialog, right-click on the AcuSolve run and select **Visualize results** to launch HyperMesh CFD Post.
2. From the Post ribbon, click the **Boundary Groups** tool.

Figure 644:

3. Click ··· on the guide bar to open the **Advanced Selection** dialog.
4. Select **Exterior_Wall - Output** then close the dialog.

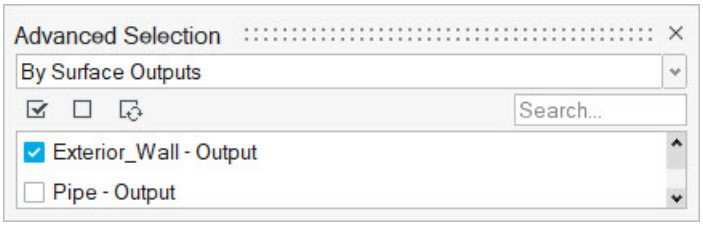


Figure 645:

5. Set the transparency of the Boundary Group to approximately `75`%.
6. On the guide bar, click ✓ to execute the command and exit the tool.
7. From the Post ribbon, click the **Iso-Surfaces** tool.

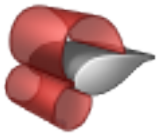


Figure 646:

8. Select **temperature** as the Iso Variable and set the Iso Value of temperature (K) to `261.15`.

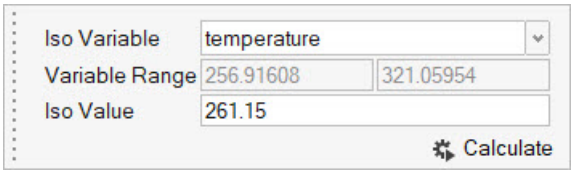


Figure 647:

9. Click **Calculate**.
10. In the display properties microdialog, define the Display as **constant** and select a color of choice.

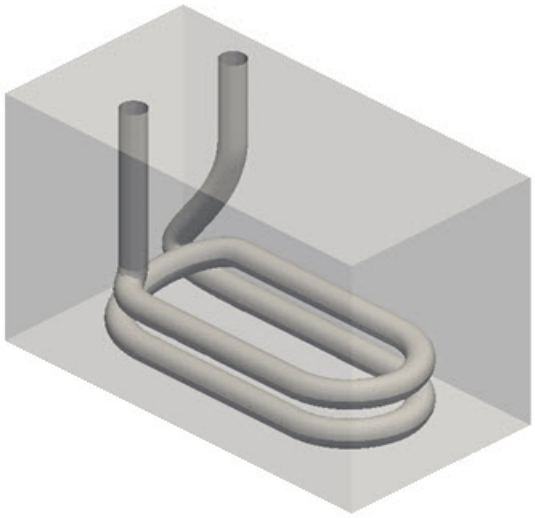


Figure 648: Transparent boundary of the tank, showing an Iso-Surface of temperature T=261.15 K, Time=0.0 sec

11. Using the results animation toolbar, skip to the end of the simulation results.

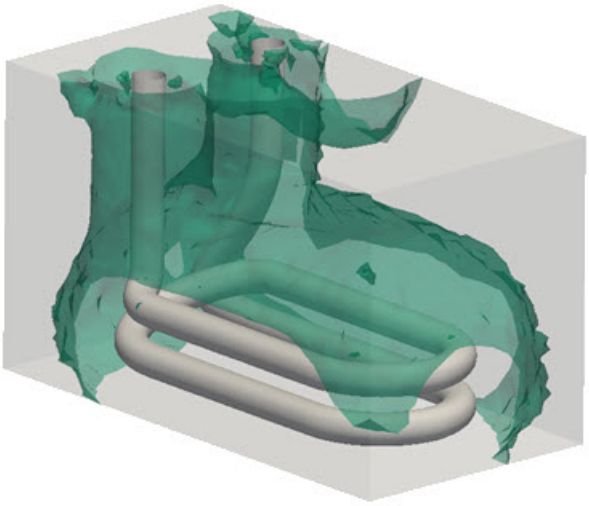


Figure 649: Transparent boundary of the tank, showing an Iso-Surface of temperature T=261.15 K, Time=100.0 sec

Summary

In this tutorial, you learned how to set up and solve a flow and thermal simulation with temperature dependent porous media effectiveness. Utilizing the proposed implementation allowed you to specify the melting point of a fluid and compute the melted region of the fluid. You started by importing the

HyperMesh CFD input database and then you defined the porous medium to control the melting point of the fluid. Next, you assigned the thermal boundary conditions and generated the mesh. Once the solution was computed, you created a plot of the heat flux and temperature for the critical surfaces with the model using HyperMesh CFD Plot Manager.

Multiphase

4

This chapter covers the following:

- ACU-T: 4000 Dam Break Simulation (p. 379)
- ACU-T: 4001 Water Filling in a Tank (p. 393)
- ACU-T: 4002 Sloshing of Water in a Tank (p. 409)
- ACU-T: 4003 Freely Falling Water Droplet (p. 423)
- ACU-T: 4100 Multiphase Flow using Algebraic Eulerian Model (p. 442)
- ACU-T: 4101 T-Junction Flow using the Eulerian Multiphase Model (p. 455)
- ACU-T: 4102 Fluidized Bed using the Granular Multiphase Model (p. 470)
- ACU-T: 4200 Humidity – Pipe Junction (p. 487)
- ACU-T: 4201 Condensation & Evaporation - Air Box (p. 502)
- ACU-T: 4300 Species Transport Modeling (p. 523)
- ACU-T: 4400 Mixing Flows Using the Gas Kinetics Model (p. 540)

ACU-T: 4000 Dam Break Simulation

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial introduces you to the workflow for setting up an AcuSolve transient dam break simulation using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4000_dambreak2D.x_t`

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a square water column held in place by the reservoir walls. At time t=0, the walls are removed and the water column is now free to flow out. The simulation can be used to visualize and study the surge patterns as the column of water rushes out, as in a dam wall break.

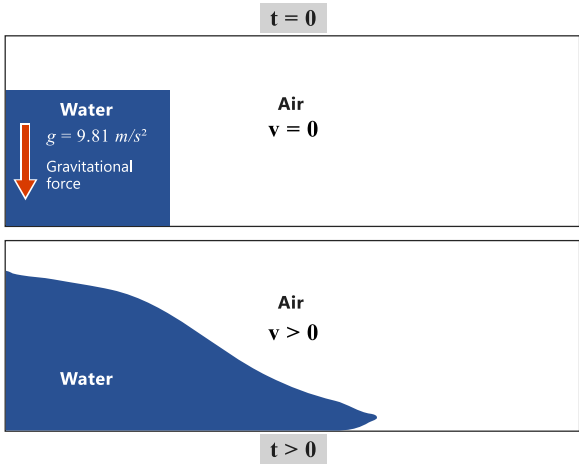


Figure 650:

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.
   - From the Home tools, Files tool group, click the **Save As** tool.

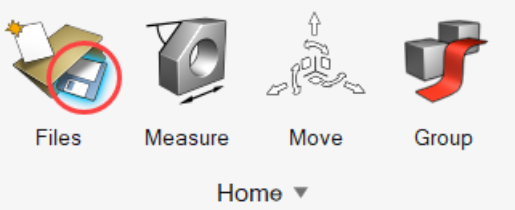


Figure 651:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `DamBreak` as the name for the database then click **Save**.

   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T4000_dambreak2D.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

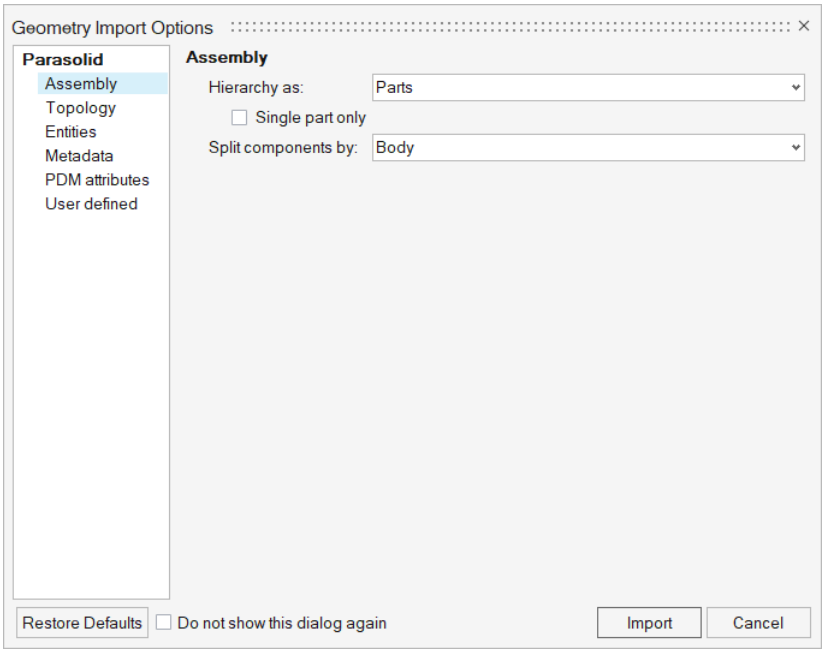


Figure 652:

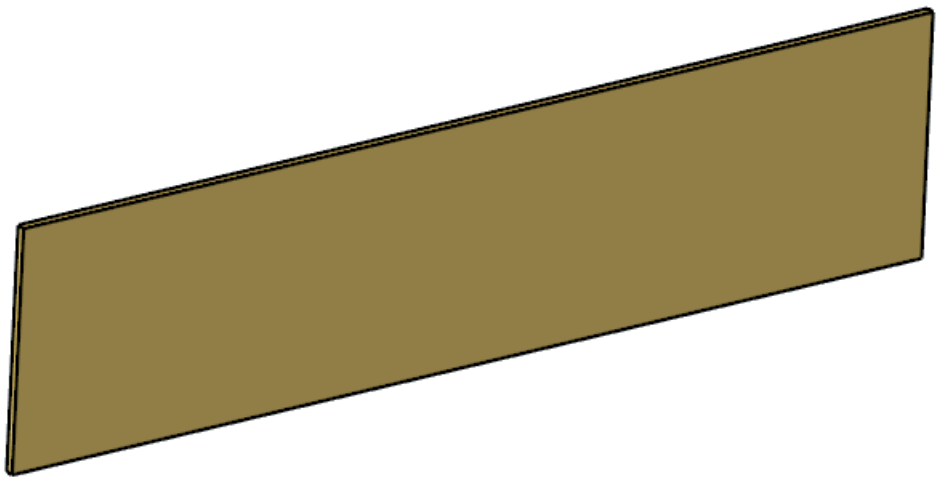


Figure 653:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 654:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 655:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

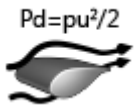


Figure 656:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Immiscible** and the Immiscible material to **Air-Water**
   c) Set the Time step size to `0.002498` and the Final time to `1.0`
   d) Select **Laminar** as the Turbulence model.
   e) Set the gravity to `-9.81` m/sec^2 in the y direction.

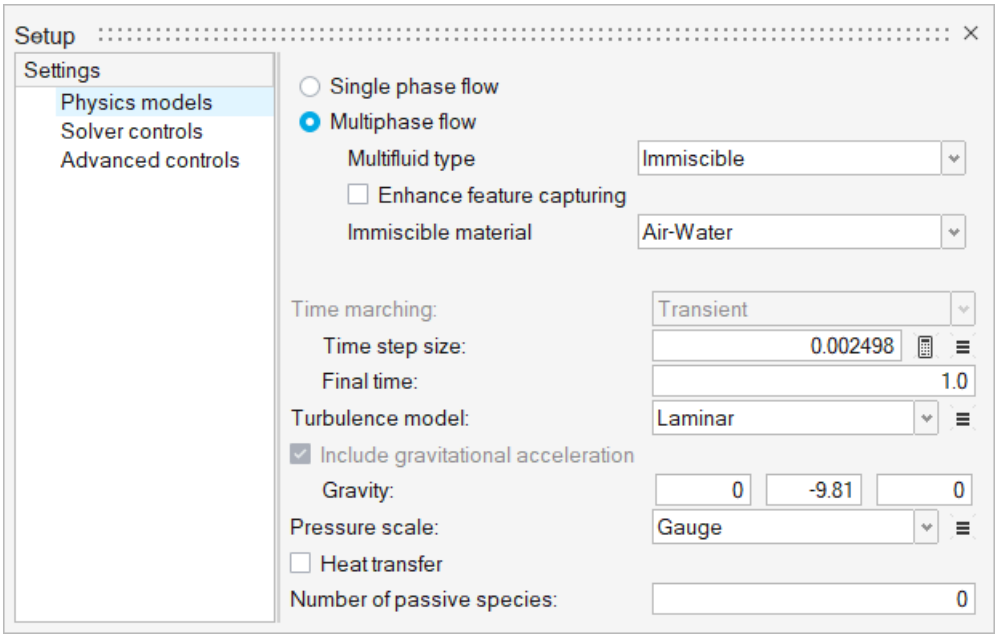


Figure 657:

3. Click the **Solver Controls** setting and set the Maximum stagger iterations to `5`.

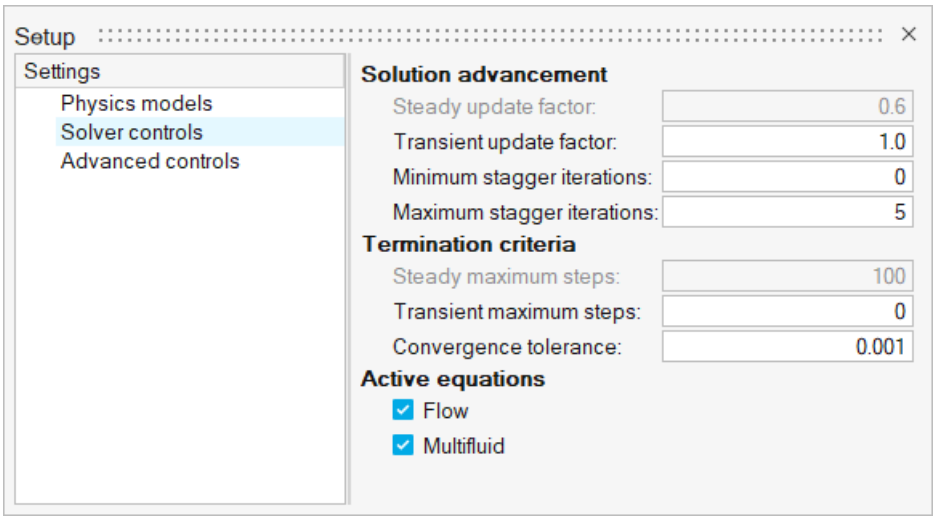


Figure 658:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 659:

2. In the Materials legend, verify that **Air-Water** is assigned.
3. Click ✓ on the guide bar.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Symmetry** tool.

Figure 660:

2. Select the right most face on the positive z-axis, as shown in the figure below.

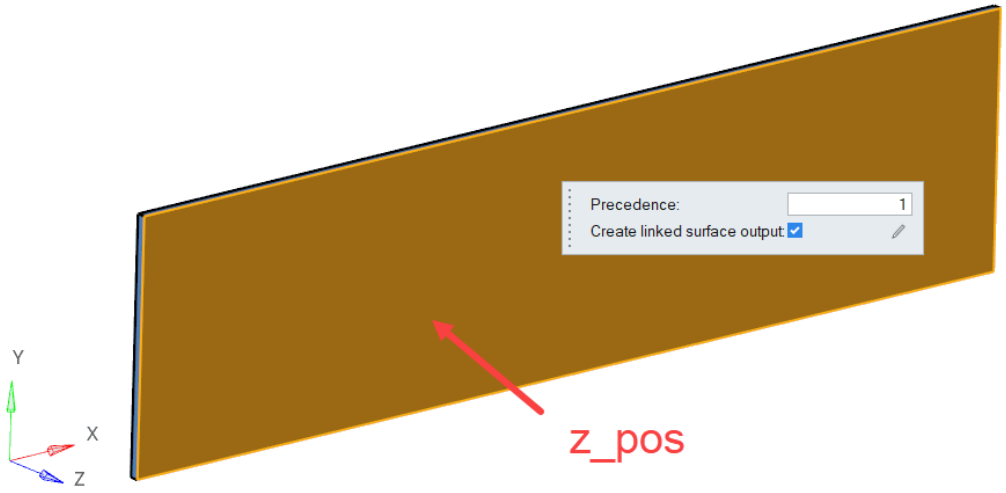


Figure 661:

3. In the Boundaries legend, double-click on **Symmetry**, rename it to `z_pos` and press **Enter**.
4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Rotate the model and select the opposite face.
6. In the Boundaries legend, rename Symmetry to `z_neg`.
7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Save the database.

Generate the Mesh

In this step, first you will create a surface mesh using the Interactive meshing tool; then, you will specify a global mesh size and growth rate for the model and generate the volume mesh using the Batch tool in the Mesh ribbon.

Create Surface Mesh

1. From the Mesh ribbon, click the **Interactive** tool.

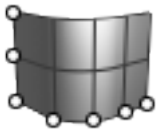


Figure 662:

By default, the Create should be selected from the secondary ribbon.

2. Click ≡ on the guide bar to open the options menu, then make the following changes:
   a) Set the Element size to `0.00254`.
   b) Set the Mesh type to **Mapped**.
   c) Set the Map method to **Map as triangle**.

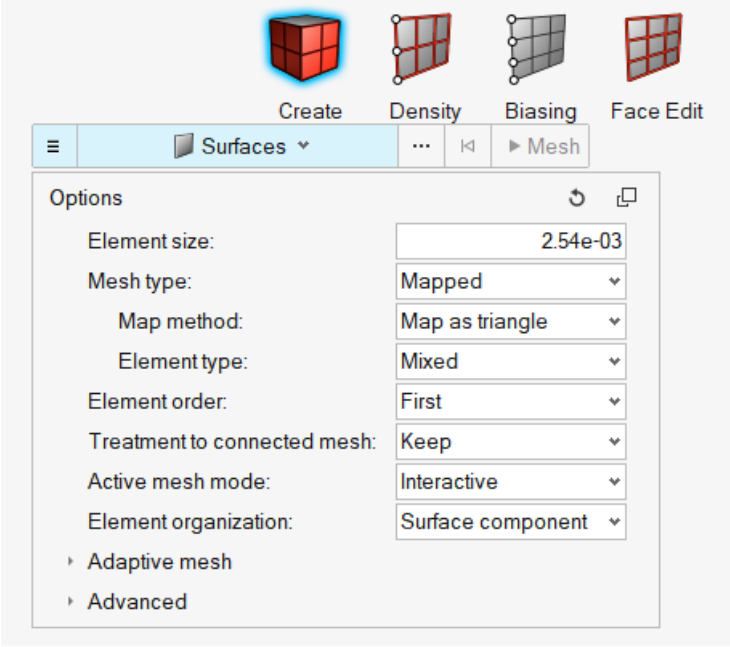


Figure 663:

3. On the guide bar, change the entity selector to **Solids** then select the solid in the modeling window.

4. Click **Mesh** in either the microdialog or on the guide bar to generate the surface mesh.
5. Once the surface mesh is created, press **Esc** to exit out of the tool.

Generate Volume Mesh

1. From the Mesh ribbon, click the **Volume** tool.

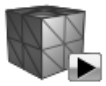


Figure 664:

The **Meshing Operations** dialog opens.

2. Verify that the Mesh size option is set to **Average size**.
3. Set the Average element size to `0.006`.

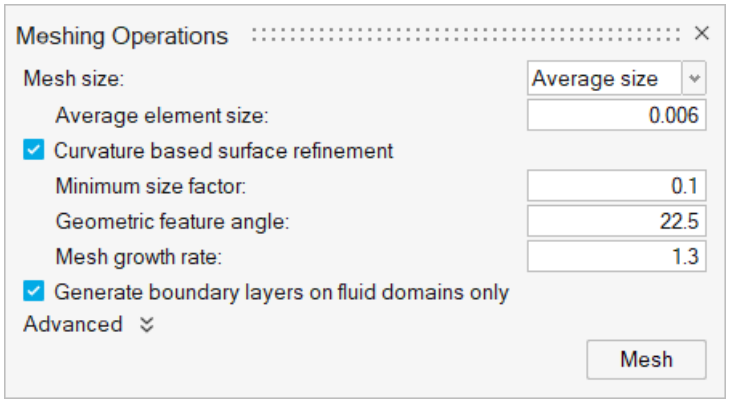


Figure 665:

4. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define Nodal Outputs and Nodal Initial Conditions

In this step, you will define the nodal output frequency and then specify the nodal initial conditions for the water column.

Define Nodal Output Frequency

1. From the Solution ribbon, click the **Field** tool.

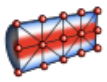


Figure 666:

The **Field Output** dialog opens.

2. Click the **Solution variables** setting.
3. Activate the **Write initial conditions** option.
4. Verify that the **Write results at time step interval** option is active.
5. Set the Time step interval to `5`.

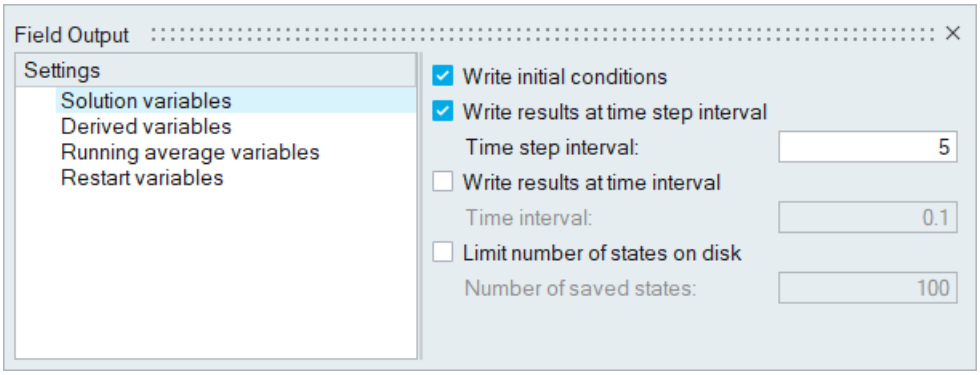


Figure 667:

Define the Nodal Initial Conditions

1. From the Solution ribbon, Zones tool group, click the **Box** tool.

Figure 668:

2. Hover the cursor over the dam geometry. When the zone is aligned with the surfaces of the model, double-click on the geometry.
An initialization zone with arbitrary dimensions should be visible on the model.

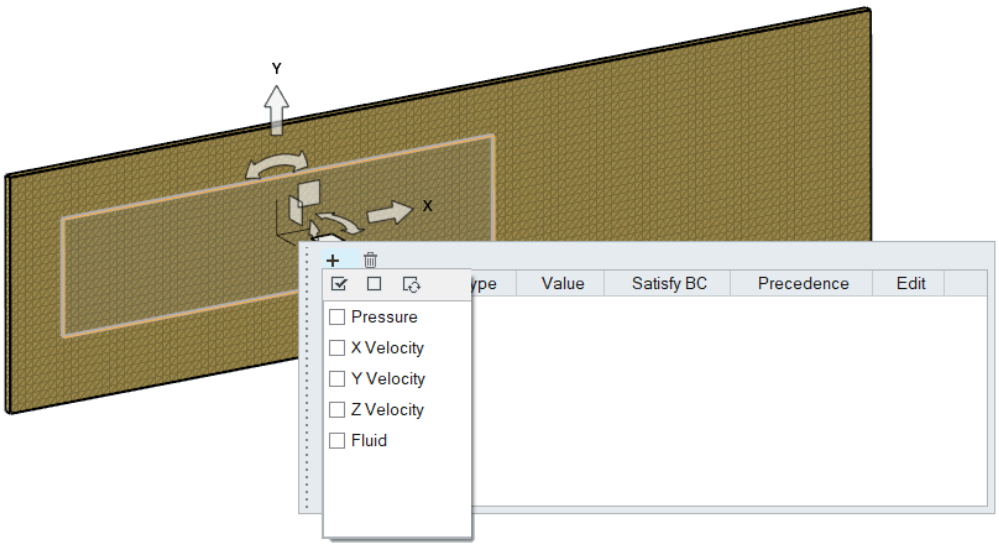


Figure 669:

3. In the microdialog, select **Fluid** then click in empty space in the dialog. Change the Value field to **Water**.

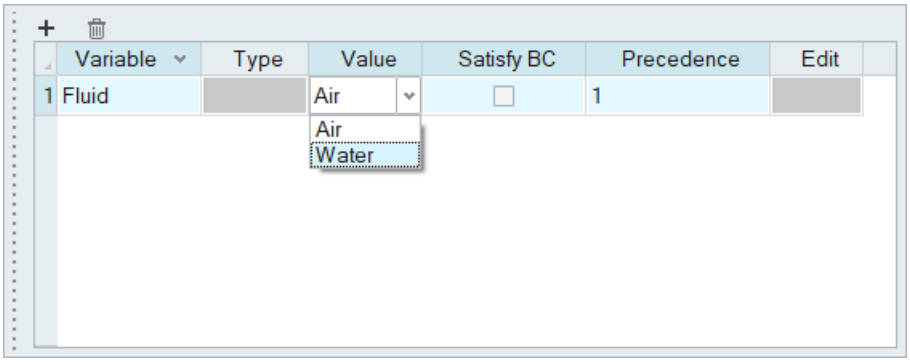


Figure 670:

4. In the zone initialization microdialog, expand the drop-down and set the dimensions of the zone as shown in the figure below.

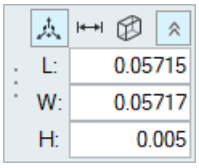


Figure 671:

Note: If you do not see the microdialog for entering the dimensions, move the NIC variable microdialog.

5. Click the center of the manipulator on the model, as shown in the figure below, then enter the following coordinates for the center: `0.028575, 0.028575, 0.003`.

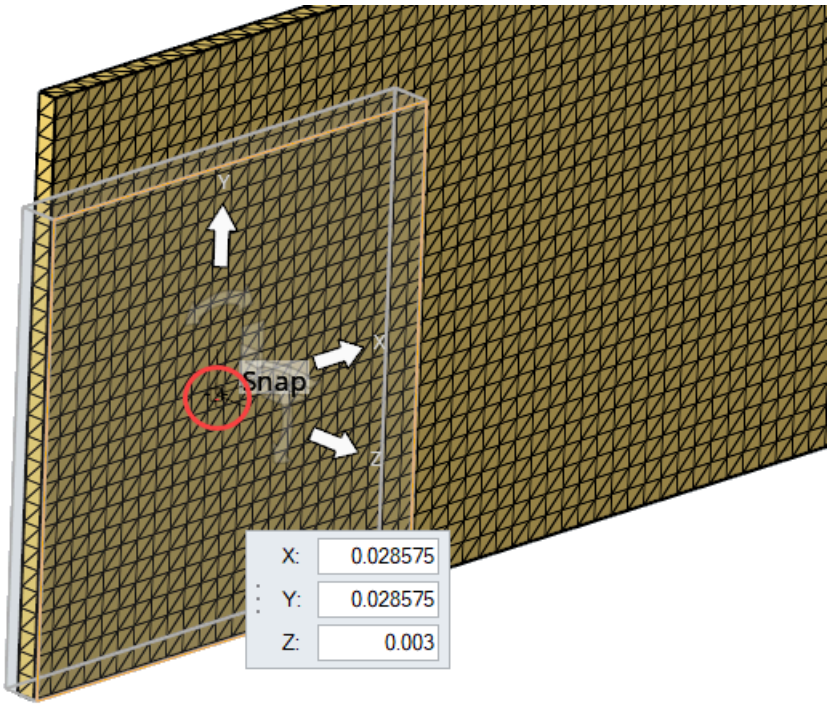


Figure 672:

This will ensure that the fluid domain is initialized with a water column at the corner of the dam.

6. Right-click in the modeling window and select the check mark to exit the tool.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 673:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Verify that the **Automatically define pressure reference** option is active.
5. Expand the **Default initial conditions** tab and deactivate the **Pre-compute flow** option.
6. Set all the velocity components to `0` and set the Immiscible fluid to **Air** (if not already set).
7. Leave the remaining options as default and click **Run** to launch AcuSolve.

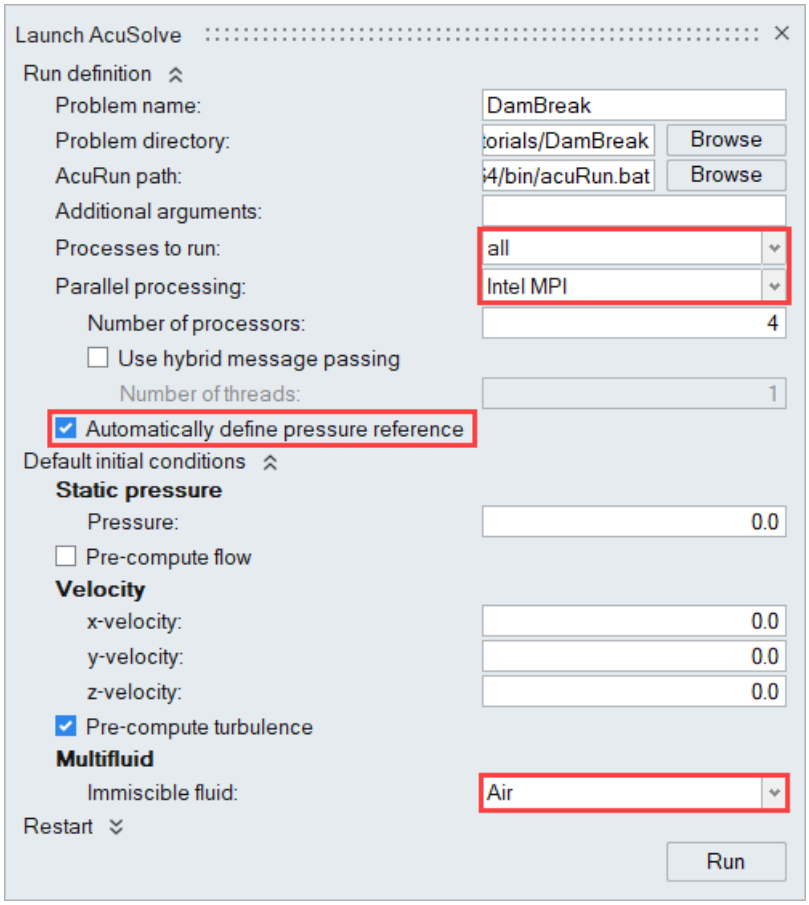


Figure 674:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Top** face on the View Cube to align the model.

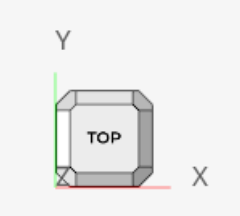


Figure 675:

5. Right-click on the **z_pos** boundary in the Post Browser and select **Edit**.
6. In the display properties microdialog, set the display to **volume fraction water**.
7. Activate the **Legend** toggle and click ⚡ to refresh the range.
8. Click ≡, set the Colormap style to **Filled**, the Number of colors to 2, and the Colormap Name to **Rainbow Desaturated**.

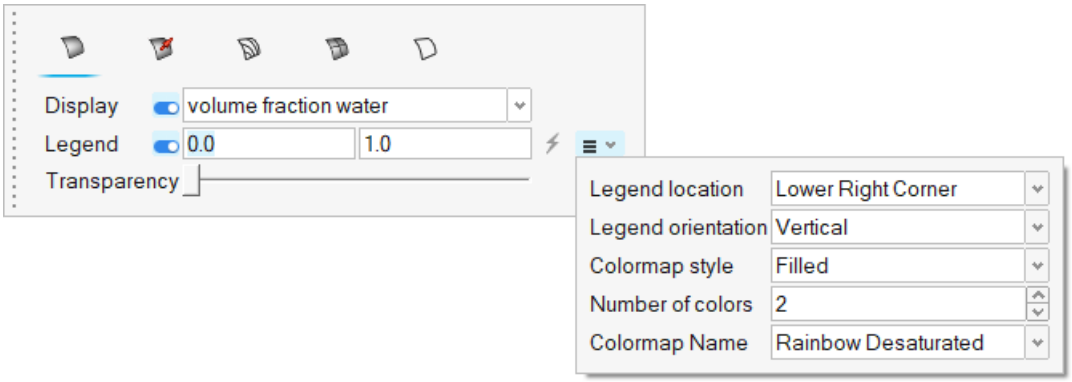


Figure 676:

9. Click ✓ on the guide bar.
10. Click ▶ at the bottom of the modeling window to view a live animation of the flow.

Figure 677:

11. Save the animation.
    a) Go to **File** > **Screen Capture** > **Advanced Capture**.
    b) Click ≡ on the toolbar.
    c) Uncheck **Include mouse cursor**.
    d) Set the frame rate to 30.
    e) Click ⌖ on the toolbar then drag over the area you want to record.
    f) Click ◉ to start recording and the same button to stop recording.
    g) Name the file and save it.

Summary

In this tutorial, you successfully learned how to set up and solve a multiphase flow problem using HyperMesh CFD and AcuSolve. You started by importing the geometry and then completed the flow set up. Once the volume meshing was done, you specified the field initial conditions for the water column using the zone initialization tool. Once the solution was computed, you post-processed the results using the Post ribbon. Here, you generated an animation of the water flow as it surged once the dam walls were removed.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/s9uijo9ak8

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4001 Water Filling in a Tank

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial provides instructions for setting up, solving, and viewing results for a transient dam break simulation using the Level Set method. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4001_tank2D.x_t`

Problem Description

The problem to be solved is shown schematically in the figure below. It consists of a half-filled water tank at time t=0. Water is injected through the Inlet at t=0 and as the water fills in through the inlet, the water-air interface can be visualized in a transient simulation.

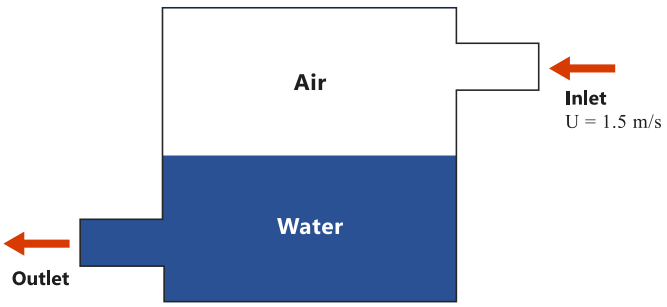


Figure 678:

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.

- From the Home tools, Files tool group, click the **Save As** tool.

Figure 679:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `FillingTank` as the name for the database then click **Save**.
   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T4001_tank2D.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

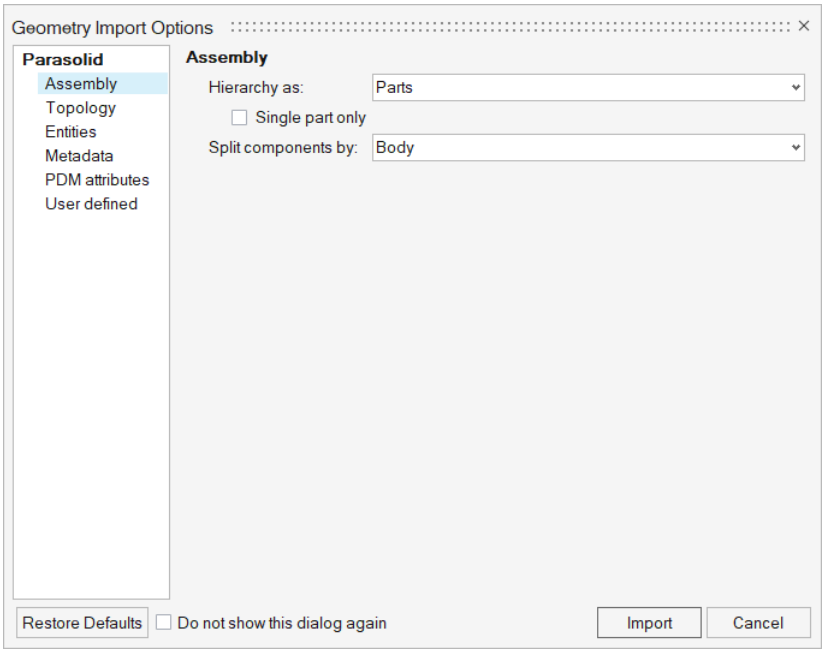


Figure 680:

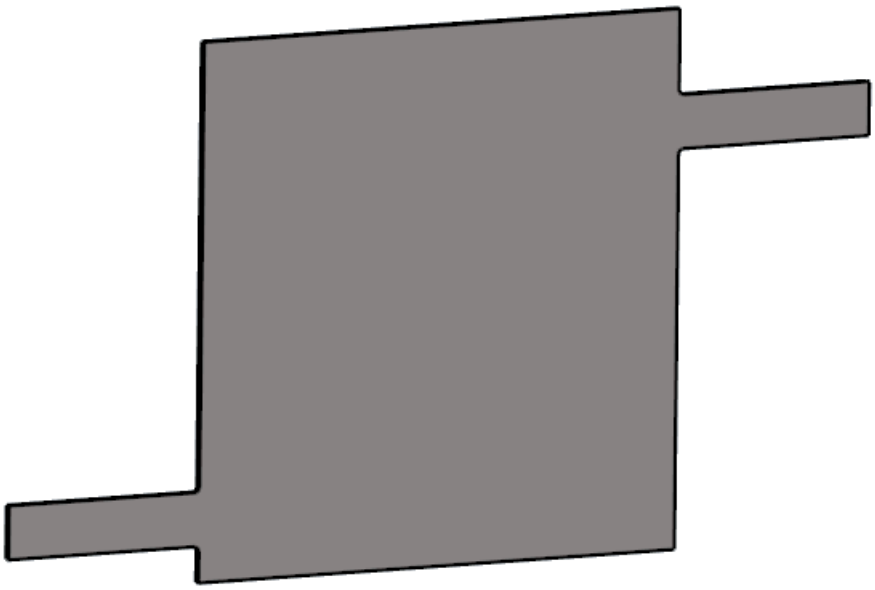


Figure 681:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 682:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 683:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

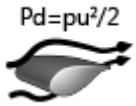


Figure 684:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Immiscible** and the Immiscible material to **Air-Water**
   c) Set the Time step size to `0.01` and the Final time to `3.0`
   d) Select **Laminar** as the Turbulence model.
   e) Set the gravity to `-9.81` m/sec^2 in the y direction.

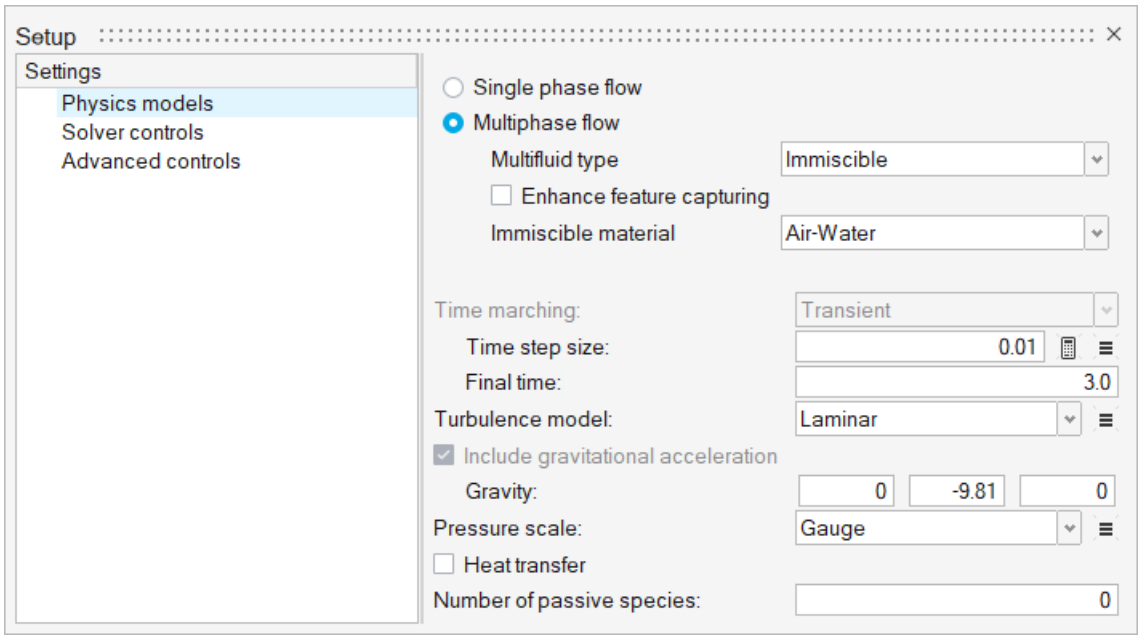


Figure 685:

3. Click the **Solver Controls** setting and set the Maximum stagger iterations to `4`.

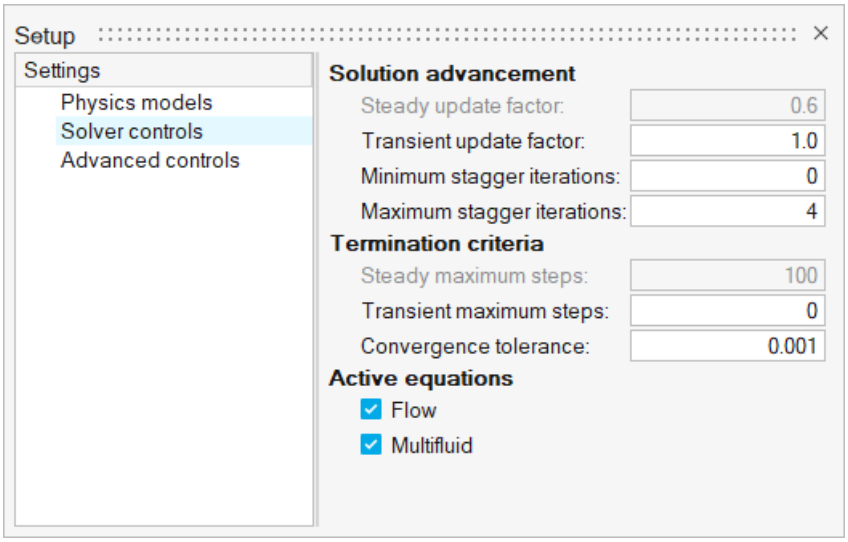


Figure 686:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 687:

2. In the Materials legend, verify that **Air-Water** is assigned.
3. Click ✓ on the guide bar.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Slip** tool.

Figure 688:

2. Select the right most face on the positive z-axis, as shown in the figure below.

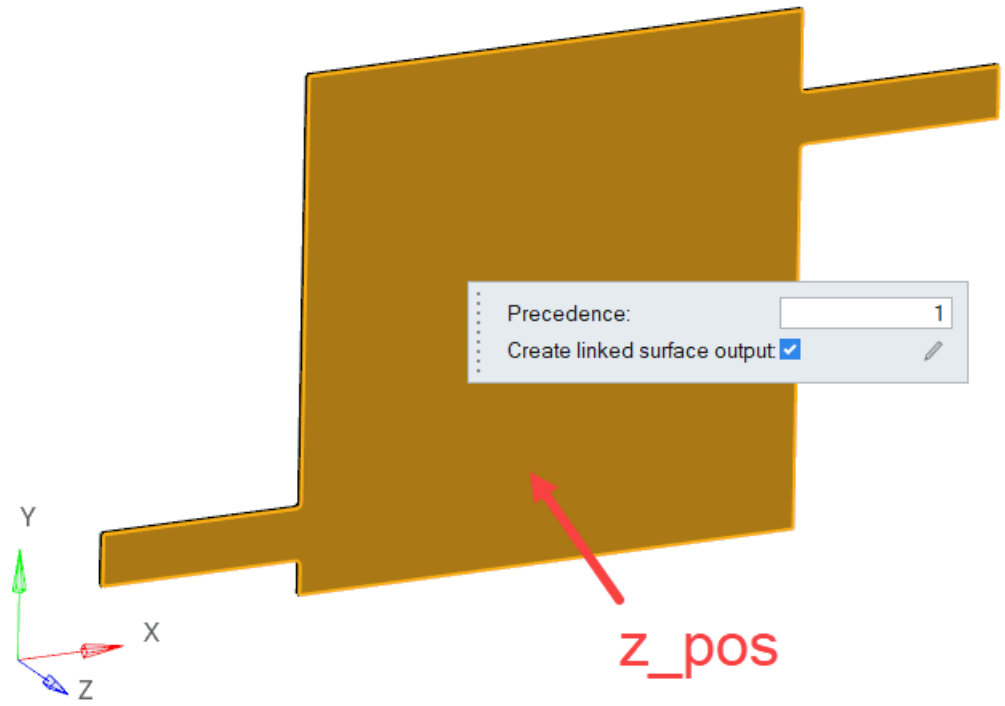


Figure 689:

3. In the Boundaries legend, double-click on **Slip**, rename it to `z_pos` and press **Enter**.
4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Rotate the model and select the opposite face.
6. In the Boundaries legend, rename Slip to `z_neg`.
7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Click the **Constant** tool.

Figure 690:

9. Select the inlet surface shown in the figure below.

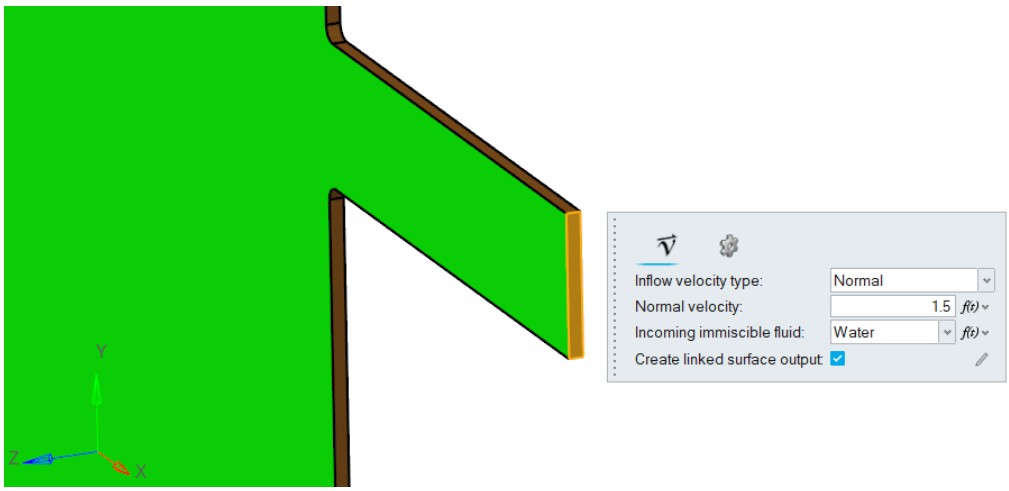


Figure 691:

10. In the microdialog,

a) Set the Inflow velocity type to **Normal**.

b) Set the Normal velocity to `1.5` m/s.

c) Select **Water** as the incoming immiscible fluid.

11. On the guide bar, click ✓ to execute the command and exit the tool.

12. Click the **Outlet** tool.

Figure 692:

13. Select the outlet surface shown in the figure below.

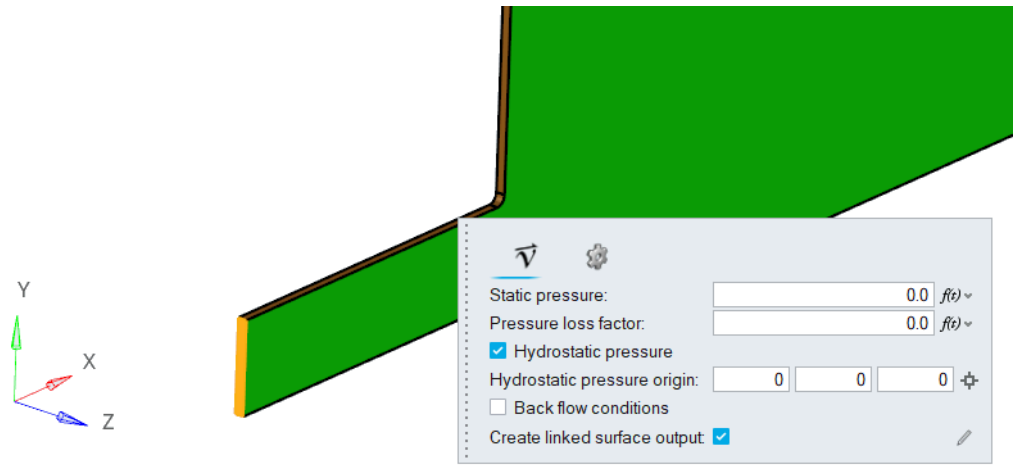


Figure 693:

14. In the microdialog, activate **Hydrostatic pressure**.
15. On the guide bar, click ✓ to execute the command and exit the tool.
16. Save the database.

Generate the Mesh

In this step, first you will create a surface mesh using the Interactive meshing tool; then, you will specify a global mesh size and growth rate for the model and generate the volume mesh using the Batch tool in the Mesh ribbon.

Create Surface Mesh

1. From the Mesh ribbon, click the **Interactive** tool.

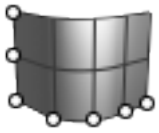


Figure 694:

By default, the Create should be selected from the secondary ribbon.

2. Click ≡ on the guide bar to open the options menu, then make the following changes:
   a) Set the Element size to `0.01`.
   b) Set the Element type to **R-Trias**.
   c) Expand **Adaptive mesh** and activate the **Curvature based refinement** option.

Leave the remaining settings unchanged.

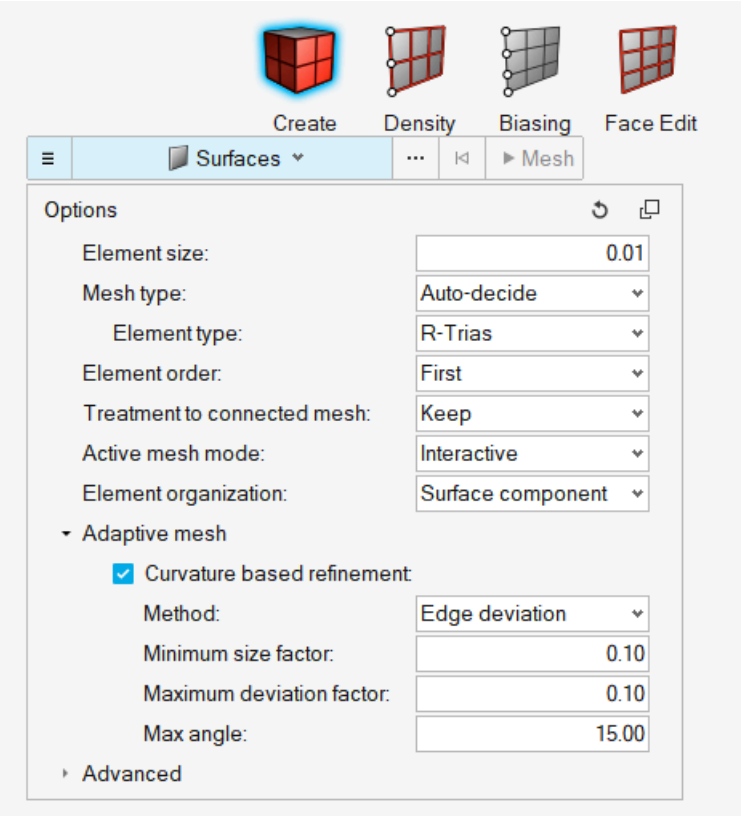


Figure 695:

3. On the guide bar, change the entity selector to **Solids** then select the solid in the modeling window.
4. Click **Mesh** in either the microdialog or on the guide bar to generate the surface mesh.
5. Once the surface mesh is created, press **Esc** to exit out of the tool.

Generate Volume Mesh

1. From the Mesh ribbon, click the **Volume** tool.

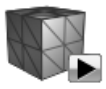


Figure 696:

The **Meshing Operations** dialog opens.

2. Verify that the Mesh size option is set to **Average size**.
3. Set the Average element size to `0.02`.
4. Set the Mesh growth rate to `1.0`.

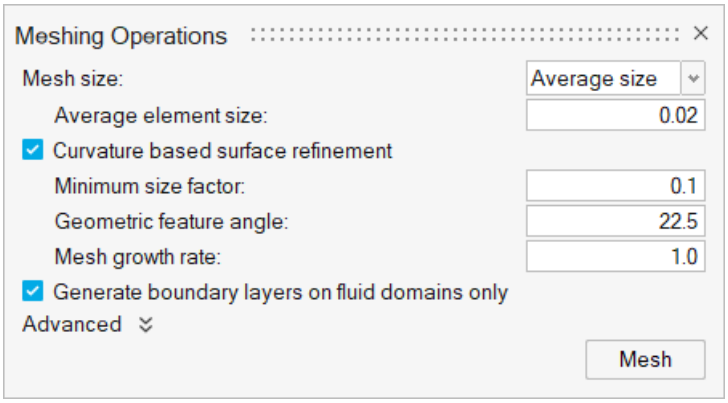


Figure 697:

5. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define Nodal Outputs and Nodal Initial Conditions

In this step, you will define the nodal output frequency and then specify the nodal initial conditions for the water column.

Define Nodal Output Frequency

1. From the Solution ribbon, click the **Field** tool.

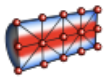


Figure 698:

   The **Field Output** dialog opens.
2. Click the **Solution variables** setting.
3. Activate the **Write initial conditions** option.
4. Verify that the **Write results at time step interval** option is active.
5. Set the Time step interval to `1`.

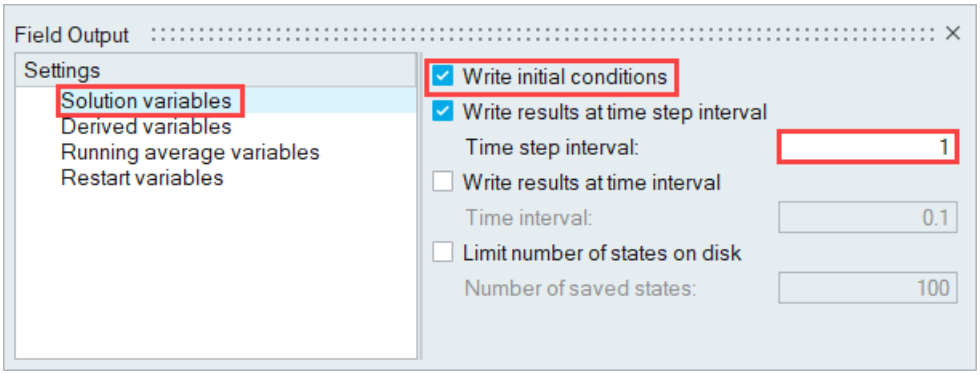


Figure 699:

Define the Nodal Initial Conditions

1. From the Solution ribbon, click the **Plane** tool.

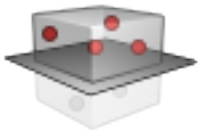


Figure 700:

2. Click on the tank solid and verify the selection on the guide bar.

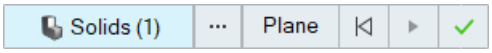


Figure 701:

3. Click **Plane** on the guide bar then click anywhere near the center of the solid.
4. In the microdialog, click + in the top-left corner, select **Fluid**, then click in empty space in the dialog.
5. Change the Value field to **Water**.

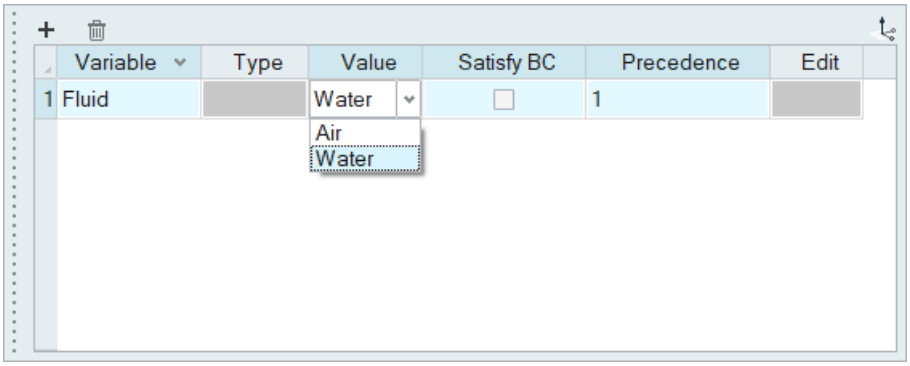


Figure 702:

6. Click in the top-right corner to orient the plane with the Vector tool.
7. In the plane definition microdialog, verify that the normal orientation is along the negative y-axis (i.e. 0, -1, 0).

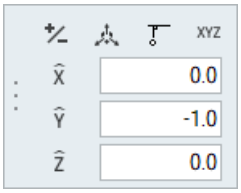


Figure 703:

8. Click XYZ, set the coordinates to `(0, 0, 0)`, then press **Enter**.

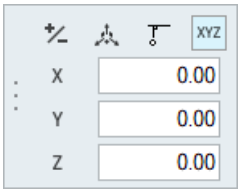


Figure 704:

9. On the guide bar, click ✓ to execute the command and exit the tool.
10. Save the database.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 705:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand the **Default initial conditions** tab and deactivate the **Pre-compute flow** option.
6. Set the x-velocity to `0`.
7. Set the Immiscible fluid to **Air** (if not already set).
8. Leave the remaining options as default and click **Run** to launch AcuSolve.

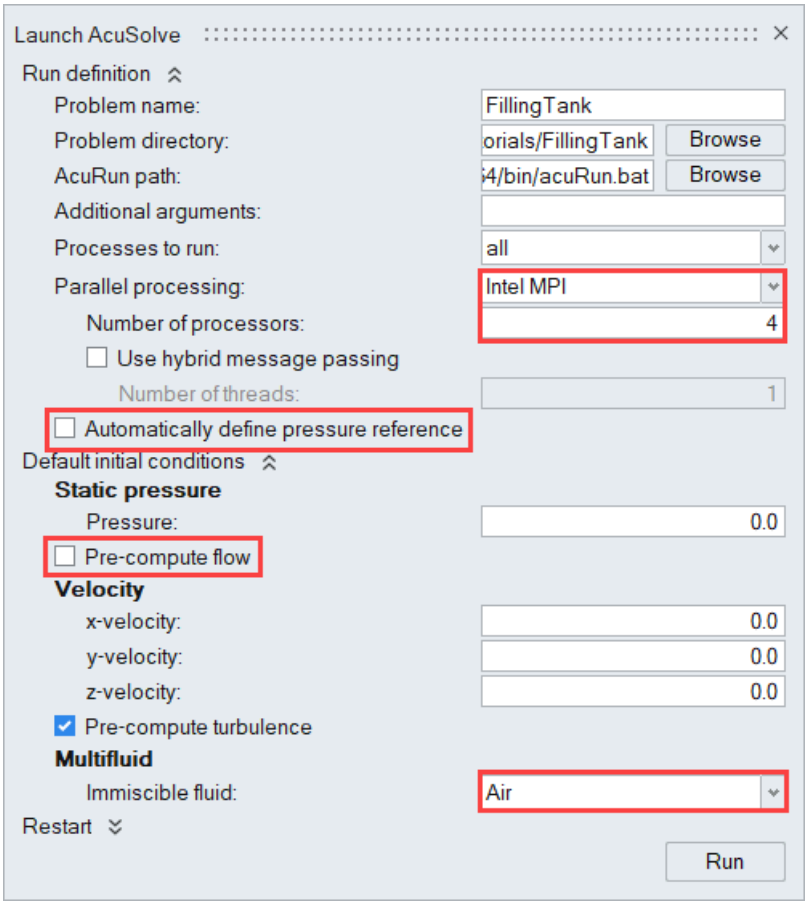


Figure 706:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Top** face on the View Cube to align the model.

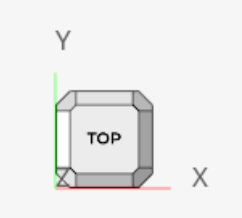


Figure 707:

5. Right-click on the **z_pos** boundary in the Post Browser and select **Edit**.
6. In the display properties microdialog, set the display to **volume fraction water**.
7. Activate the **Legend** toggle and click ⚡ to refresh the range.
8. Click ≡ ˅, set the Colormap style to **Filled**, the Number of colors to 2, and the Colormap Name to **Rainbow Desaturated**.

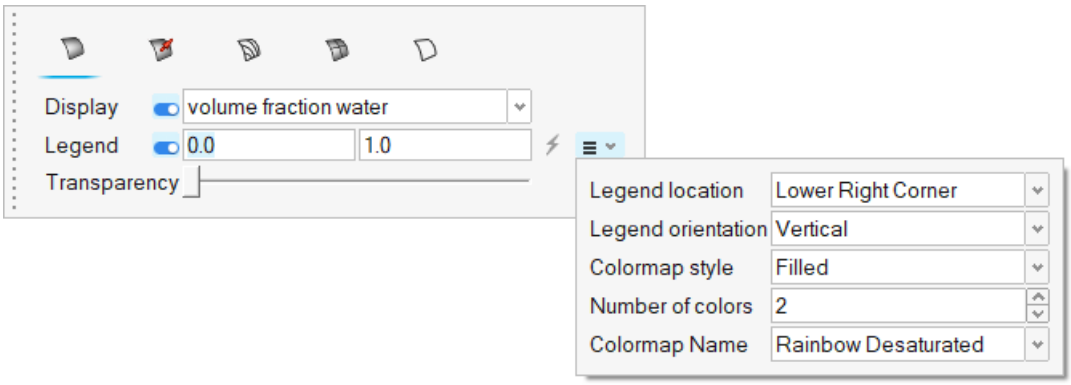


Figure 708:

9. Click ✓ on the guide bar.

10. Click ▶ at the bottom of the modeling window to view a live animation of the flow.

Figure 709:

11. Save the animation.

a) Go to **File** > **Screen Capture** > **Advanced Capture**.

b) Click ≡ on the toolbar.

c) Uncheck **Include mouse cursor**.

d) Set the frame rate to `30`.

e) Click ✛ on the toolbar then drag over the area you want to record.

f) Click ◉ to start recording and the same button to stop recording.

g) Name the file and save it.

Summary

In this tutorial, you successfully learned how to set up and solve a multiphase flow tank filling problem using HyperMesh CFD and AcuSolve. You started by importing the geometry and then completed the flow set up. Once the volume meshing was done, you specified the field initial conditions for the water column using the plane initialization tool. Once the solution was computed, you post-processed the results using the Post ribbon where you generated an animation of the water flow.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/xhwwnxj34x

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4002 Sloshing of Water in a Tank

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides instructions for running a transient simulation of a two-phase flow in a rectangular tank using the level set model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4002_TankSloshing.hm`

`bodyForce.c`

Since the HyperMesh CFD database (.hm file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved is shown schematically in the figure below. It consists of a partially filled water tank and from time t=0, water inside the tank is subjected to a sinusoidal varying body force along x-direction and constant gravity along y-direction.

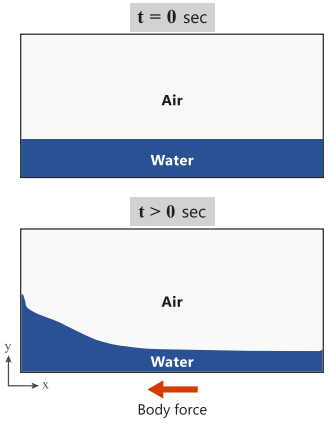


Figure 710:

The body force in the x-direction is given by the expression:

$$A\omega^2 \sin(\omega t + \varphi)$$

Where

- A = Amplitude of oscillation = -0.06 m
- ω = Frequency of oscillation = $\frac{2\pi}{T}$ = 3.6 rad/sec
- T = Time period of oscillation = 1.74 sec
- φ = Phase difference = 0

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 711:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4002_TankSloshing.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Tank_Sloshing` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `tank_sloshing` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 712:

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

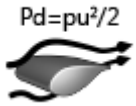


Figure 713:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Immiscible** and the Immiscible material to **Air-Water**
   c) Set the Time step size to `0.006` and the Final time to `4.0`
   d) Select **Laminar** as the Turbulence model.
   e) Set the gravity to `-9.81` m/sec^2 in the y direction.
   f) Select **Absolute** for the Pressure scale.

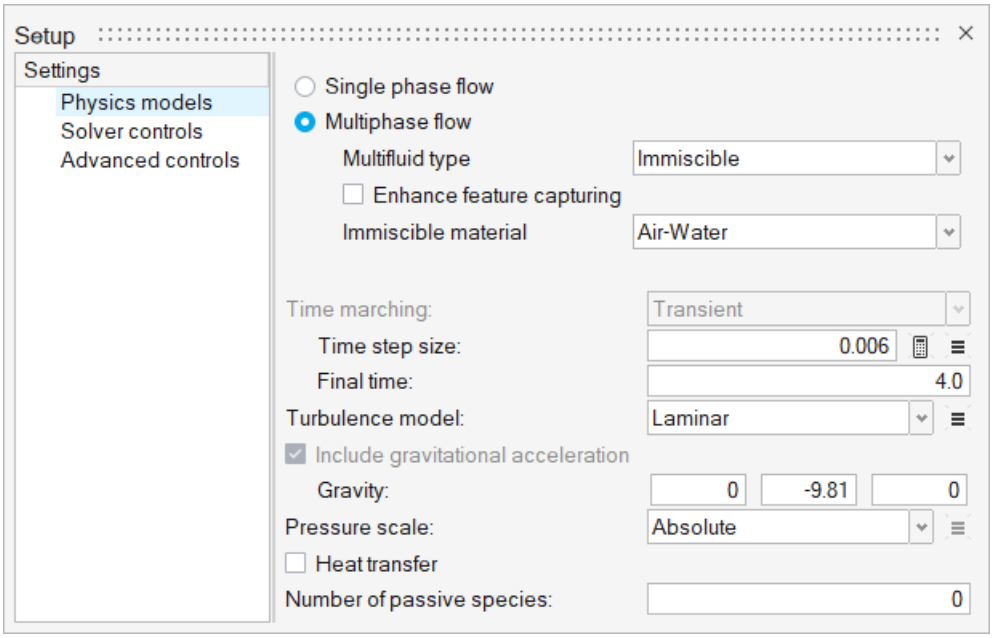


Figure 714:

3. Click the **Solver Controls** setting.

a) Set the Minimum stagger iterations to `2`.

b) Set the Maximum stagger iterations to `6`.

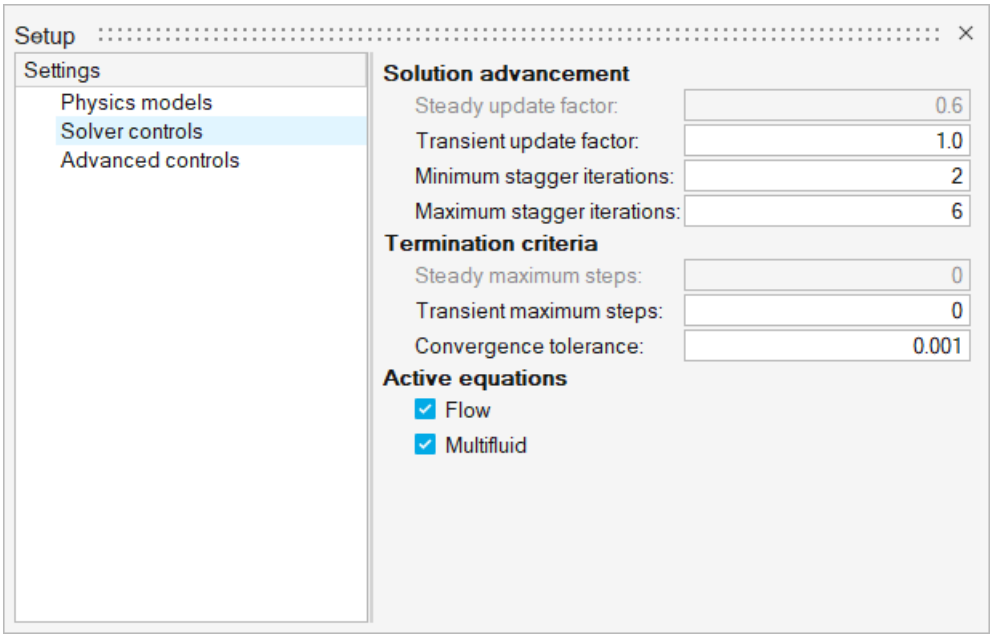


Figure 715:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 716:

2. In the Materials legend, verify that **Air-Water** is assigned.
3. Click ✓ on the guide bar.

Set the Body Force

1. From the Flow ribbon, click the **Sources** > **Momentum** tool.

Figure 717:

2. Select the solid body.
3. In the microdialog, set the Type to **User Function**.
4. Click and enter the following values.

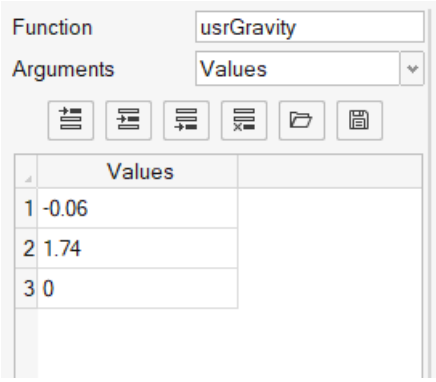


Figure 718:

These values are for the amplitude of oscillation = -0.06 m; time period of oscillation = 1.74 sec; Phase difference = 0

5. In Momentum Source legend, double-click on **Momentum**, rename it to `bodyForce`, then press **Enter**.

> **Note:** The values provided above are the ones described in the problem description. The user values should be provided in the same order as shown above because these values will be passed on to the UDF script, which refers to these values in specific order.

6. On the guide bar, click ✓ to execute the command and exit the tool.

Compile the Body Force UDF

A UDF in the form of C language (`bodyForce.c`) is provided with the tutorial. This program should be compiled using the following steps.

- For Windows:
  a) Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt** .
  b) Change the directory to the present working directory using the `cd` command.
  c) Enter the following command then press **Enter**.

  ```
  acuMakeDll -src bodyForce.c
  ```
- For Linux:
  a) In the terminal, change the directory to the present working directory using the `cd` command.
  b) Enter the following command then press **Enter**.

  ```
  acuMakeLib -src bodyForce.c
  ```

This command creates a set of files necessary for the use of the UDF.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **No Slip** tool.

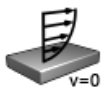


Figure 719:

2. Select all four surfaces shown in the figure below.

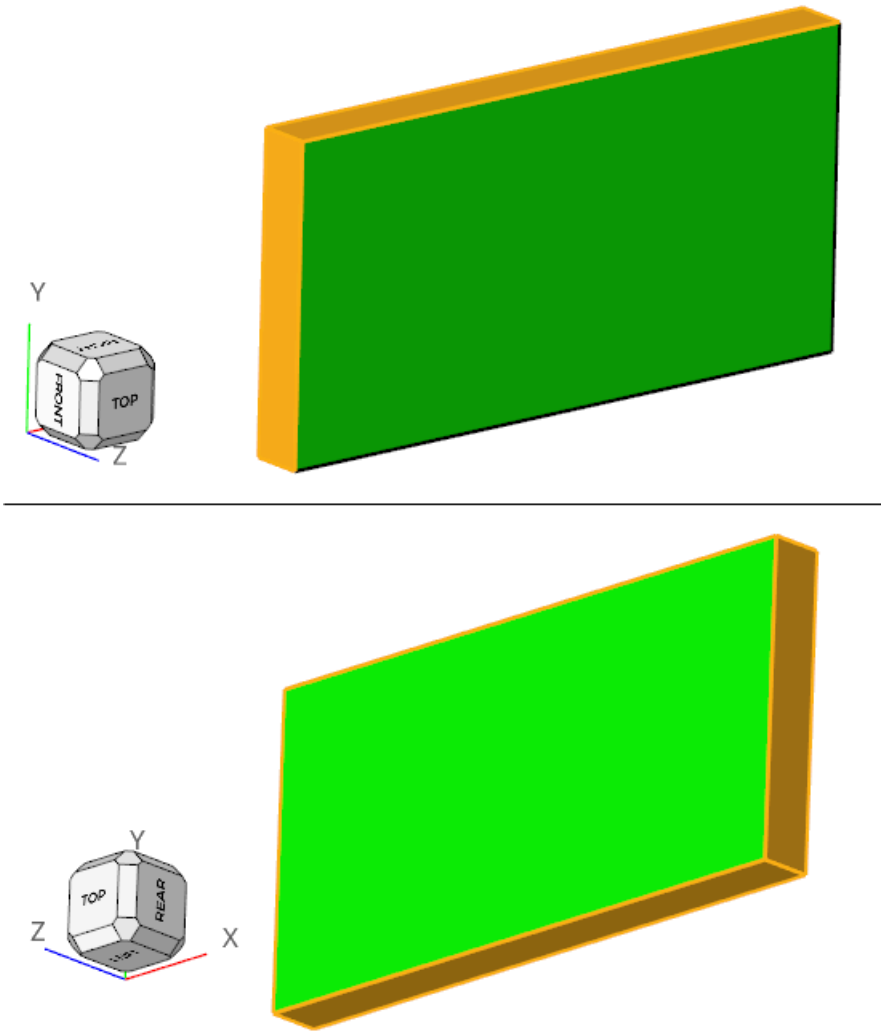


Figure 720:

3. Click ✓ on the guide bar.
4. Click the **Slip** tool.

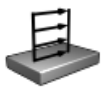


Figure 721:

5. Select the right most face on the positive z-axis, as shown in the figure below.

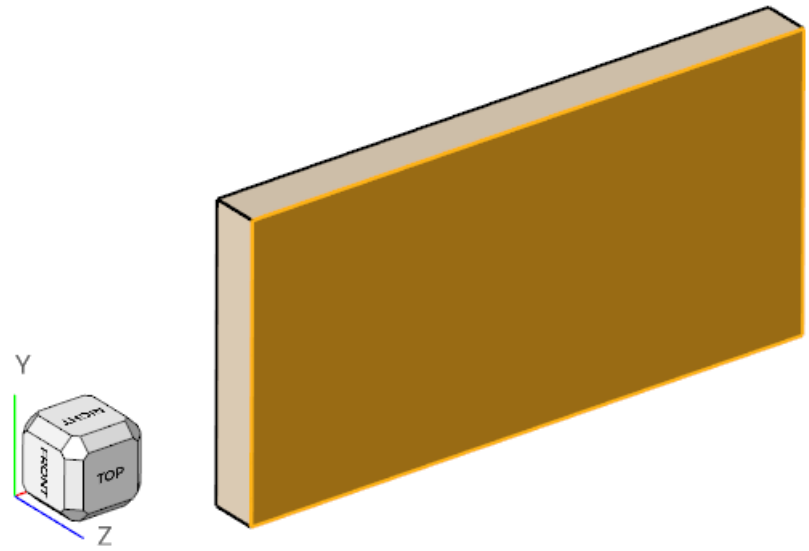


Figure 722:

6. In the Boundaries legend, double-click on **Slip**, rename it to `z_pos`, then press **Enter**.
7. On the guide bar, click ▸ to execute the command and remain in the tool.
8. Rotate the model and select the opposite face.
9. In the Boundaries legend, rename Slip to `z_neg`.
10. Click ✓ on the guide bar.
11. Save the database.

Define Nodal Initial Conditions

1. From the Solution ribbon, click the **Plane** tool.

Figure 723:

2. Select the solid body.
3. Click **Plane** on the guide bar.
4. Click any point on the model to initially position the plane.
5. Click in the microdialog and manually set the plane axis to Y.

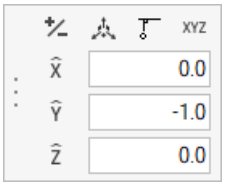


Figure 724:

6. Click XYZ and set the coordinates to `(0.60, 0.12, 0.10)`.

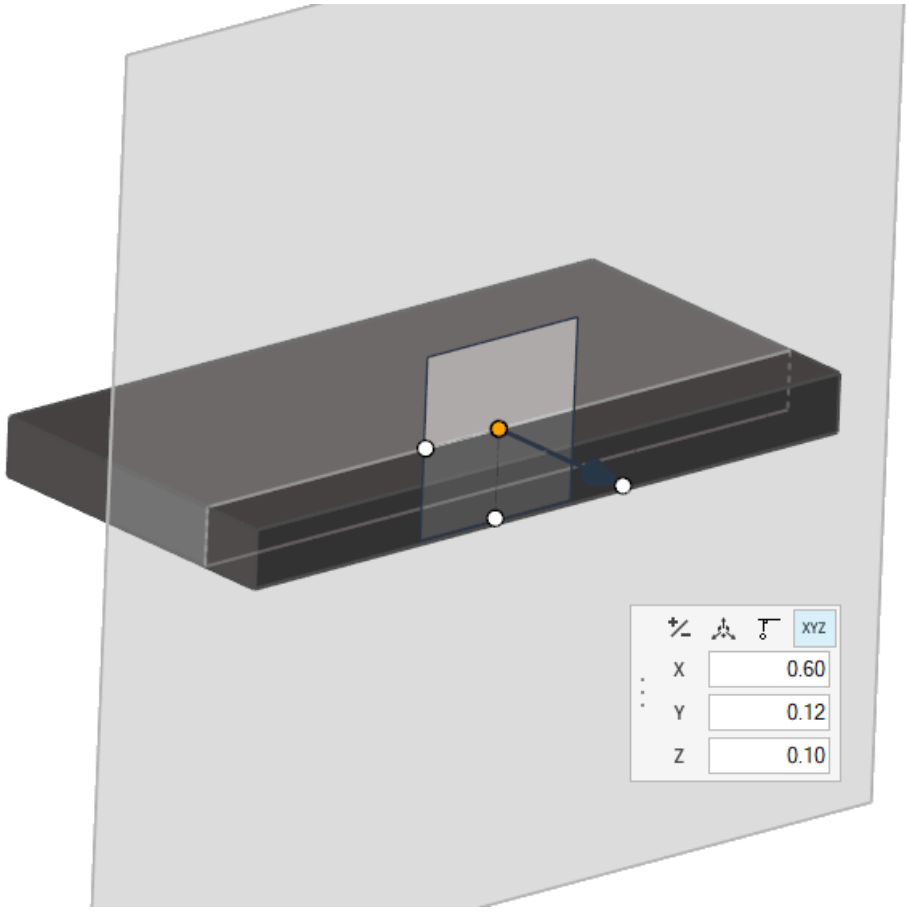


Figure 725:

7. Press **Esc** two times.
8. Click **+** to add a variable then select **Fluid** from the list.
9. Make sure that the Value is set to **Water** then press **Esc** again.

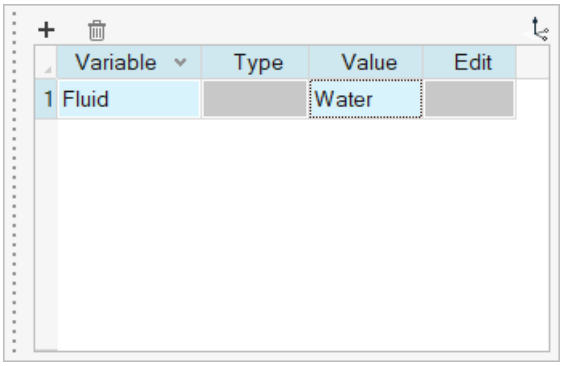


Figure 726:

10. Click ✓ on the guide bar.
11. Save the database.

Define Nodal Outputs

1. From the Solution ribbon, click the **Field** tool.

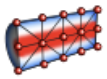


Figure 727:

The **Field Output** dialog opens.

2. Set the parameters for Solution variables as shown in the figure below.

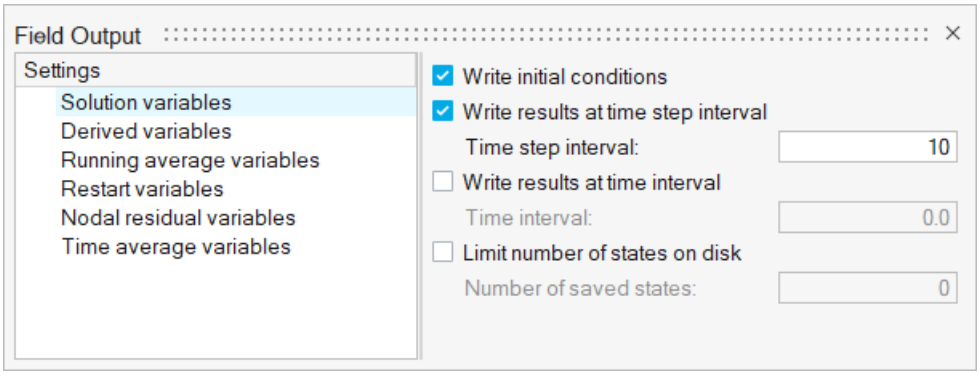


Figure 728:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 729:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand the **Default initial conditions** tab and deactivate the **Pre-compute flow** and **Pre-compute turbulence** options.
5. Set the x-velocity to `0`.
6. Leave the remaining options as default and click **Run** to launch AcuSolve.

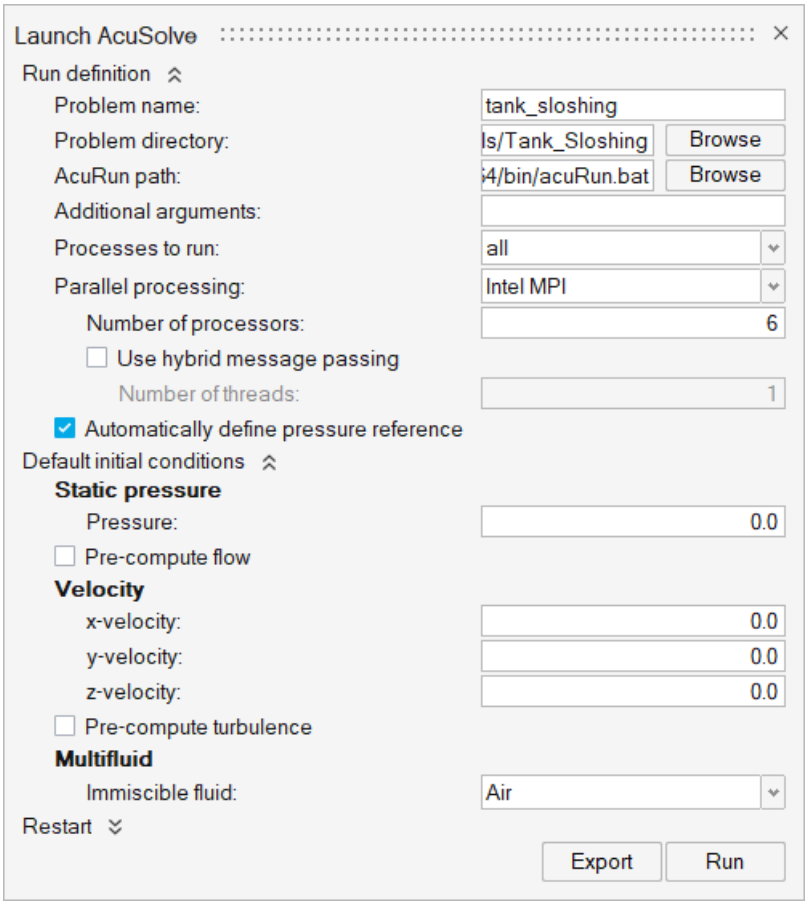


Figure 730:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Top** face on the View Cube to align the model.

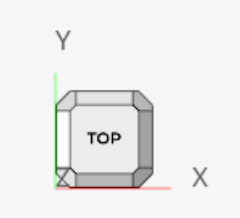


Figure 731:

5. Right-click on the **z_pos** boundary in the Post Browser and select **Edit**.
6. In the display properties microdialog, set the display to **volume fraction water**.
7. Activate the **Legend** toggle and click ⚡ to refresh the range.
8. Click ≡ ˅, and set the Colormap Name to **Rainbow Uniform**.

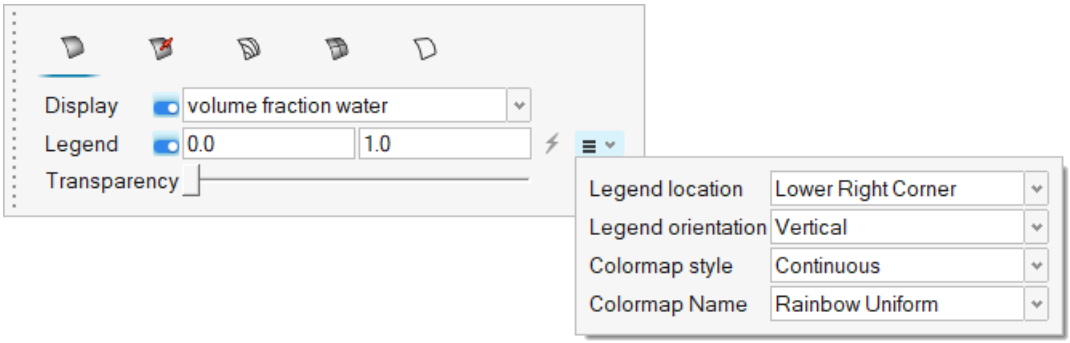


Figure 732:

9. Click ✓ on the guide bar.
10. Click ▶ at the bottom of the modeling window to view a live animation of the flow.

Figure 733:

11. Save the animation.

a) Go to **File** > **Screen Capture** > **Advanced Capture**.
b) Click ≡ on the toolbar.
c) Uncheck **Include mouse cursor**.
d) Set the frame rate to `24`.
e) Click ⌖ on the toolbar then drag over the area you want to record.
f) Click ◉ to start recording and the same button to stop recording.
g) Name the file and save it.

Summary

In this tutorial, you successfully learned how to set up and solve a transient multiphase flow problem involving water sloshing in a tank using HyperMesh CFD and AcuSolve. You also learned how to create a multiphase model using the Level Set method and specify the body force using a user-defined function and then compile the UDF. Once the solution was computed, you post-processed the results using the Post ribbon where you generated an animation of the water sloshing in the tank.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/s32qtj273i

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4003 Freely Falling Water Droplet

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial introduces you to the workflow for setting up a transient droplet simulation using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4003_FallingDroplet.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. In this problem, a water droplet with 2 mm diameter falls freely under gravity. Given the small drop radius, surface tension and contact angle as the imposed wall treatment play a significant role to recover the drop shape after the impact. You will use AcuSolve to simulate the different developing drop shapes after impacting the wall.

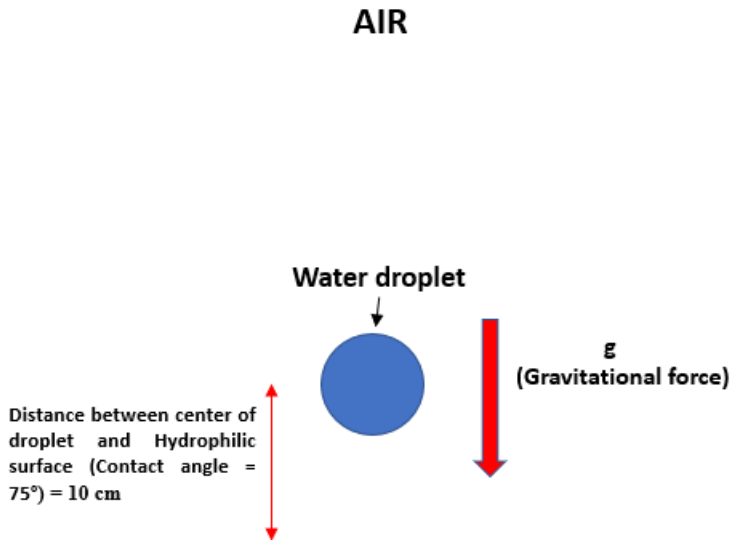


Figure 734:

Note: The schematic is not of exact scale.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 735:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4003_FallingDroplet.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `ContactAngle` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `FallingDroplet` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 736:

Set Up Flow

Create Materials

1. From the Flow ribbon, click the **Material Library** tool.

   Figure 737:

   The **Material Library** dialog opens.
2. Under Settings, click **Immiscible Multiphase** then click the **My Materials** tab.
3. Click **+** to create a new material.
4. In the new dialog, rename the material to `air_water_st` by clicking the name in the top-left corner, then enter the properties shown below.

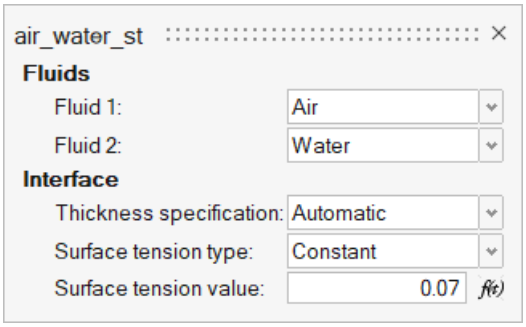


Figure 738:

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

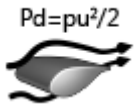


Figure 739:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Immiscible** and the Immiscible material to **air_water_st**
   c) Set the Time step size to `0.0005` and the Final time to `0.15`
   d) Select **Laminar** as the Turbulence model.
   e) Set the gravity to `-9.81` m/sec^2 in the y direction.

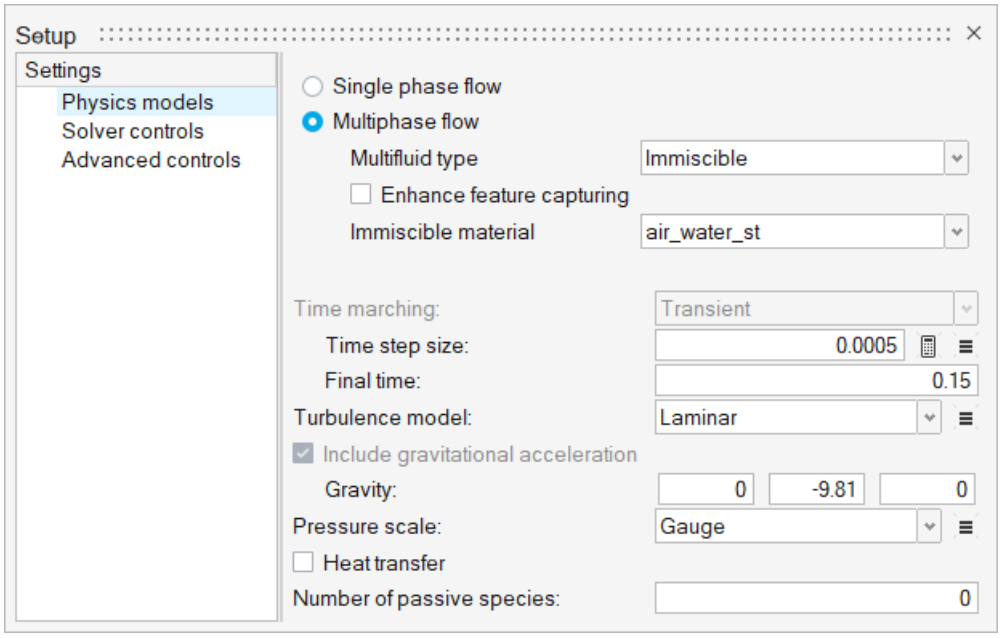


Figure 740:

3. Click **Solver controls** and verify the following:
   - Minimum stagger iterations: `0`
   - Maximum stagger iterations: `6`
   - The **Flow** and **Multifluid** checkboxes are activated.

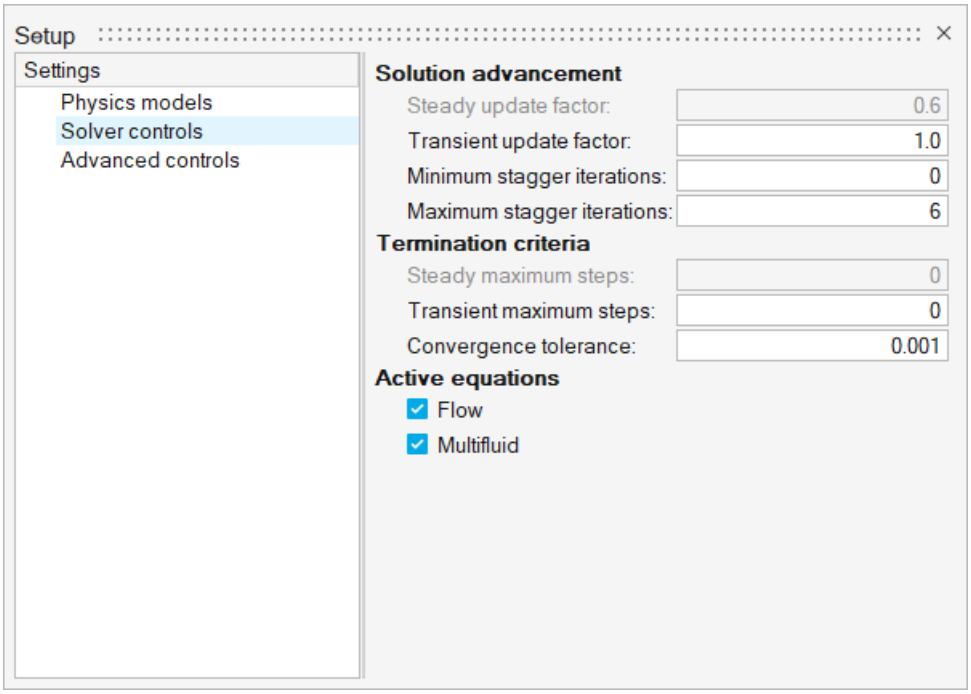


Figure 741:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 742:

2. Verify that **air_water_st** has been assigned to the volume.
3. Click ✓ on the guide bar.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Symmetry** tool.

Figure 743:

2. Click on the View Controls toolbar and set the model orientation to **Isometric**.
3. Select the Z-Positive face, as highlighted below.

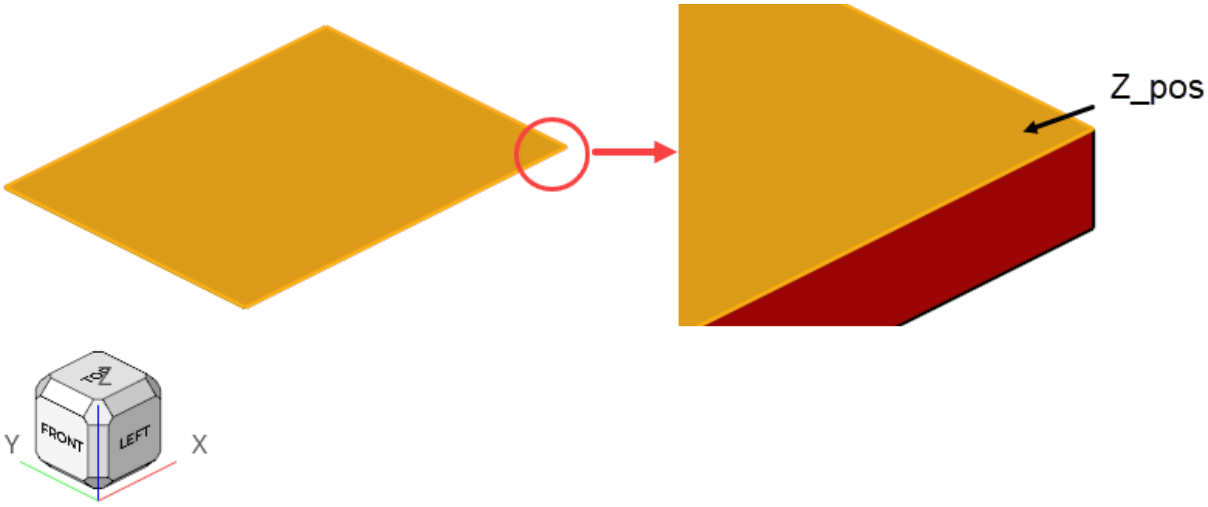


Figure 744:

4. In the Boundaries legend, double-click on **Symmetry** and rename it to `Z_pos`
5. On the guide bar, click to execute the command and remain in the tool.
6. Rotate the model and select the opposite face.
7. In the Boundaries legend, double-click on **Symmetry** and rename it to `Z_neg`
8. On the guide bar, click to execute the command and exit the tool.
9. Click the **Outlet** tool.

Figure 745:

10. In the Boundaries legend, right-click on **Default Wall** and select **Isolate**.
11. Click on the View Controls toolbar and set the model orientation to **Isometric**.
12. Select the top and side faces, as highlighted below.

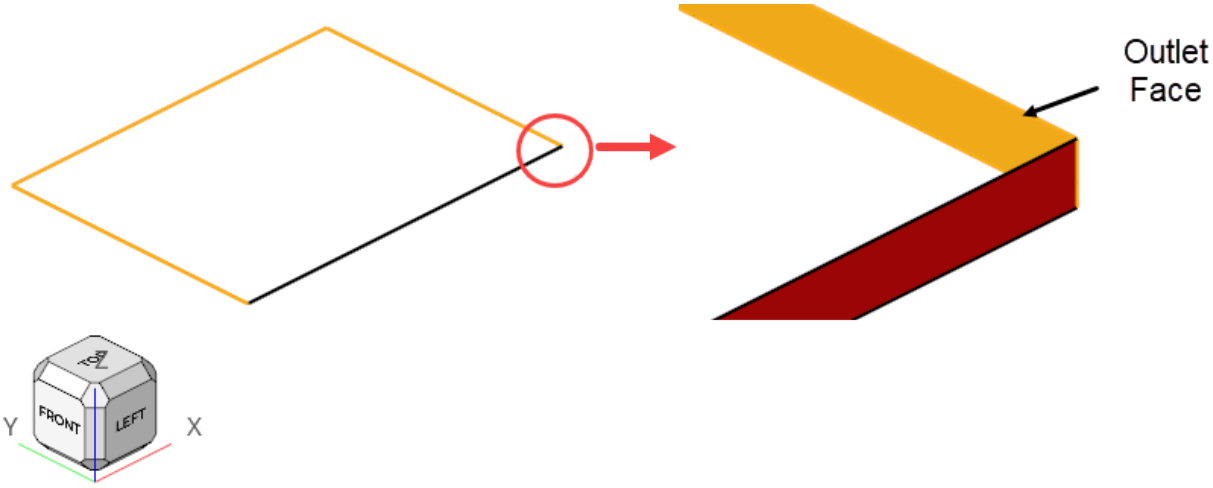


Figure 746:

13. In the microdialog, set the following properties then click ✓ on the guide bar.

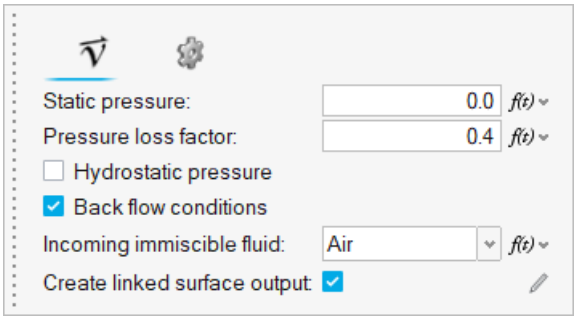


Figure 747:

14. Click the **No Slip** tool.

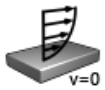


Figure 748:

15. In the Boundaries legend, **Isolate** the Default Wall then **Show** the Outlet faces.

16. Select the face highlighted below.

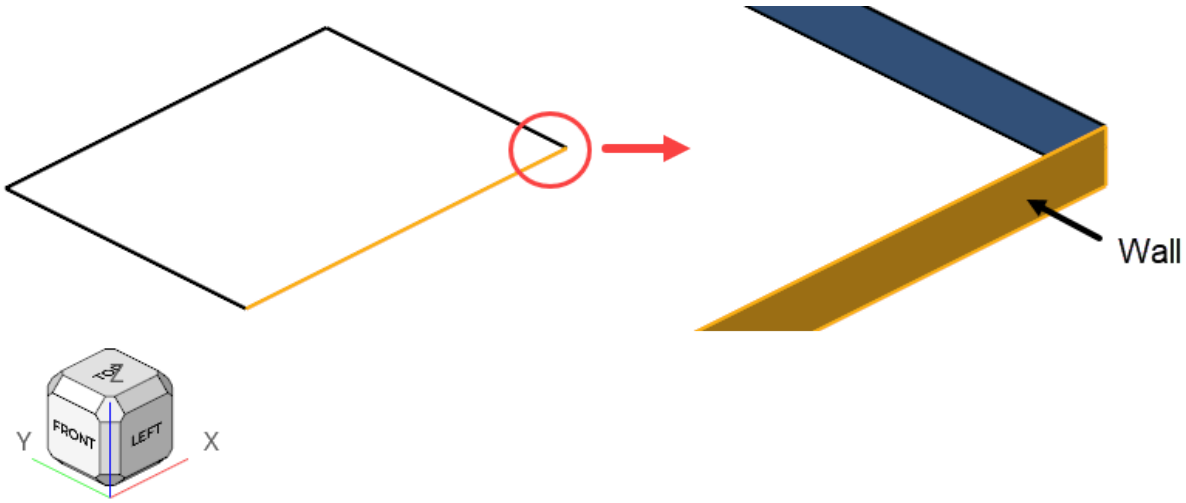


Figure 749:

17. In the microdialog, activate the **Specify contact angle** checkbox, set the Contact angle value to `75`, then click ✓ on the guide bar.

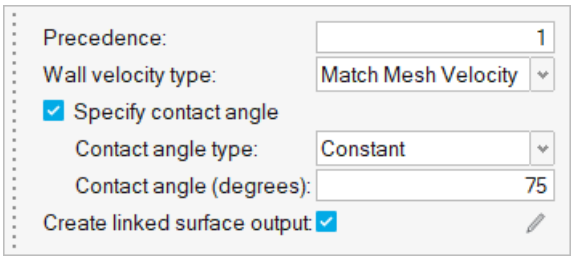


Figure 750:

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

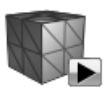


Figure 751:

The **Meshing Operations** dialog opens.

2. Check that the Average element size is 0.006.

3. Accept all other default parameters.

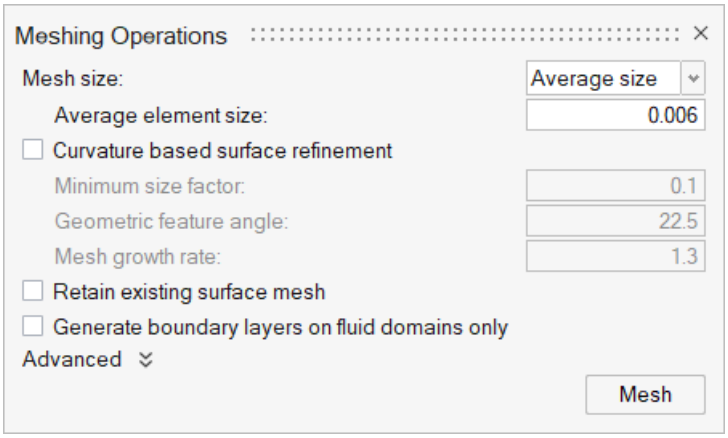


Figure 752:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define Nodal Outputs and Nodal Initial Conditions

In this step, you will define the nodal output frequency and then specify the nodal initial conditions for the water droplet.

Define Nodal Output Frequency

1. From the Solution ribbon, click the **Field** tool.

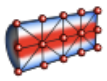


Figure 753:

   The **Field Output** dialog opens.
2. Click the **Solution variables** setting.
3. Activate the **Write initial conditions** checkbox.
4. Verify that the **Write results at time step interval** checkbox is active.

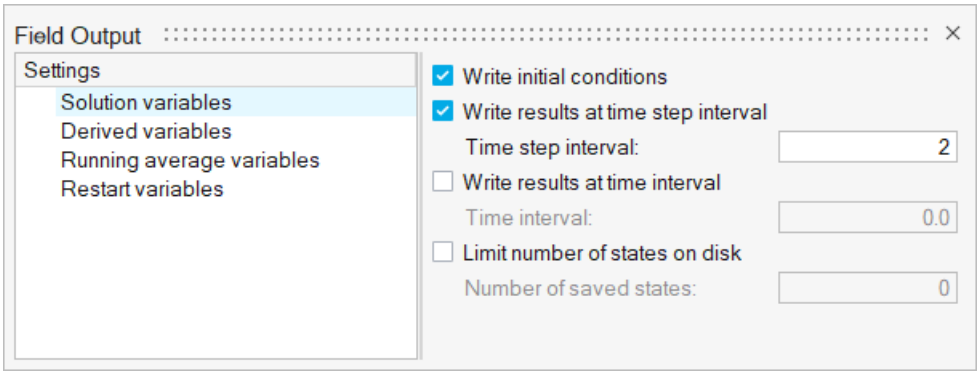


Figure 754:

Define Nodal Initial Conditions

1. From the Solution ribbon, Zones tool group, click the **Cylinder** tool.

Figure 755:

2. Select the **Top** face on the View Cube to orient the model.
3. Double-click on the geometry to create the cylinder-shaped zone then press **Esc**.

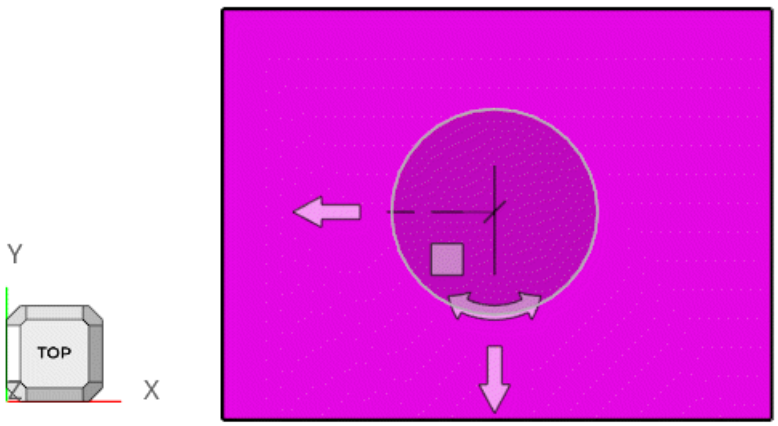


Figure 756:

4. To provide the dimensions of the cylinder, right click on **Nodal initial condition cylinder** in the legend and select **Edit**.

5. In the microdialog, click the icon on the right to expand the menu, then set the radius to `0.001` and the height to `0.00339589`

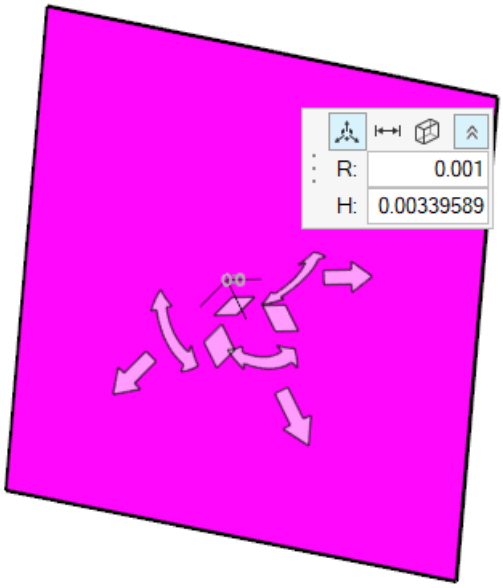


Figure 757:

6. Click on the View Controls toolbar and select **Isometric**.

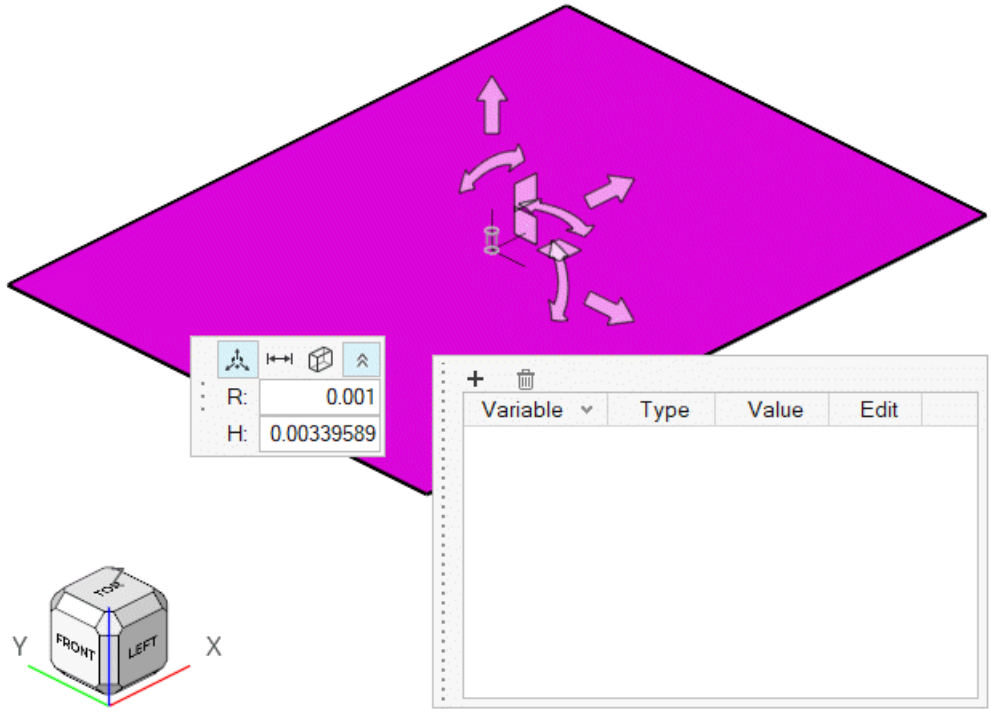


Figure 758:

7. Zoom in on the zone and select the base point of the cylinder, as shown in the figure below. Set the coordinates to `(0.06, 0.02, -0.001)`.

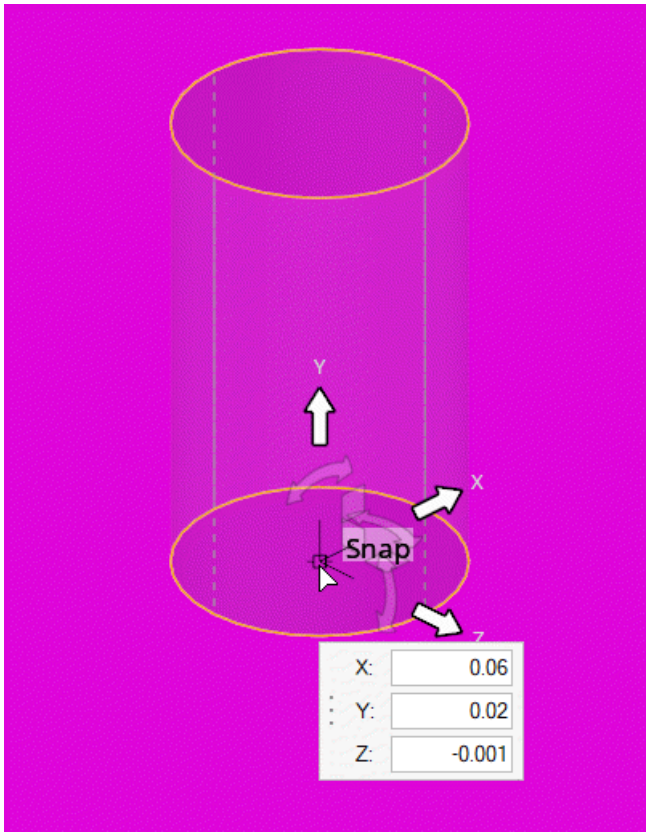
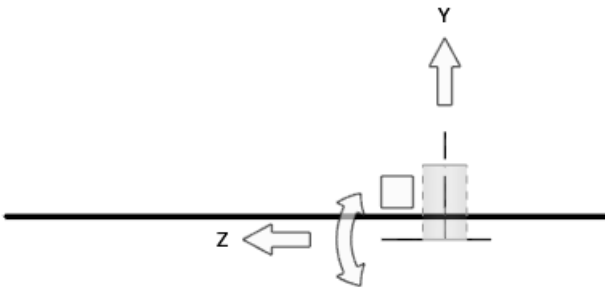


Figure 759:

8. Select the **Rear** face on the View Cube and verify that cylinder zone is passing over the domain completely as shown in below figure.



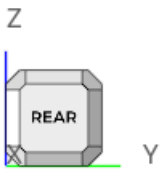


Figure 760:

9. In the initial condition dialog, click +, select **Fluid**, then left-click in empty space.
10. Change the fluid to **Water**.

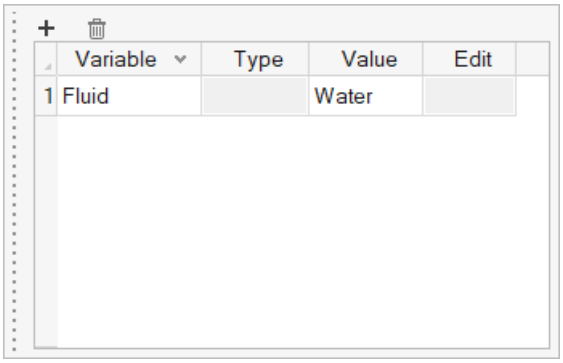


Figure 761:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 762:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Leave the remaining options as default and click **Run** to launch AcuSolve.

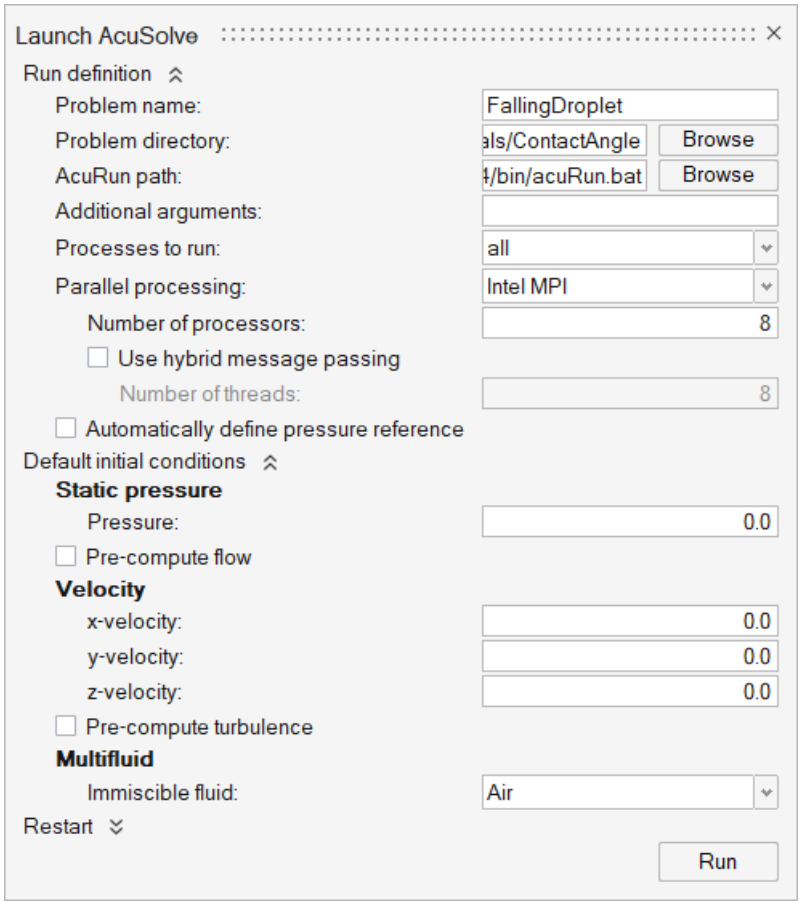


Figure 763:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

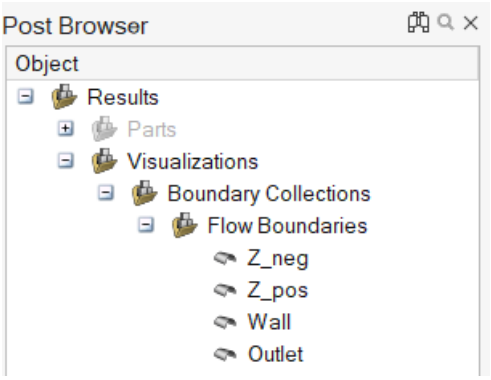


Figure 764:

4. To check the **contours of volume fraction of water** on the solid, right-click on **Z_pos** in the Post Browser and select **Isolate**.
5. Right-click on **Z_pos** again and select **Edit**.
   A microdialog for surface coloring opens.
6. Set the display to **volume fraction water** and toggle on the **Legend** radio button.
7. Click ≡ ˅ in the same dialog and to verify the legend properties shown below.
   You will be able to see the contours of volume fraction of water at time step=1.

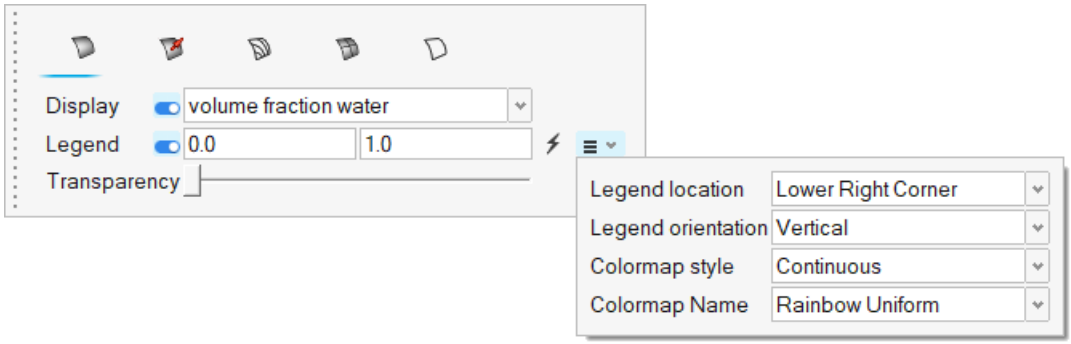


Figure 765:

Figure 766:

8. Make sure the time scale at the bottom of modelling window is at time step =1. Click on the bar to see the falling of droplet and interaction of it with surface at time step = 65, 75, 85 and final time step (where 1 time step = 2 secs).

Figure 767:

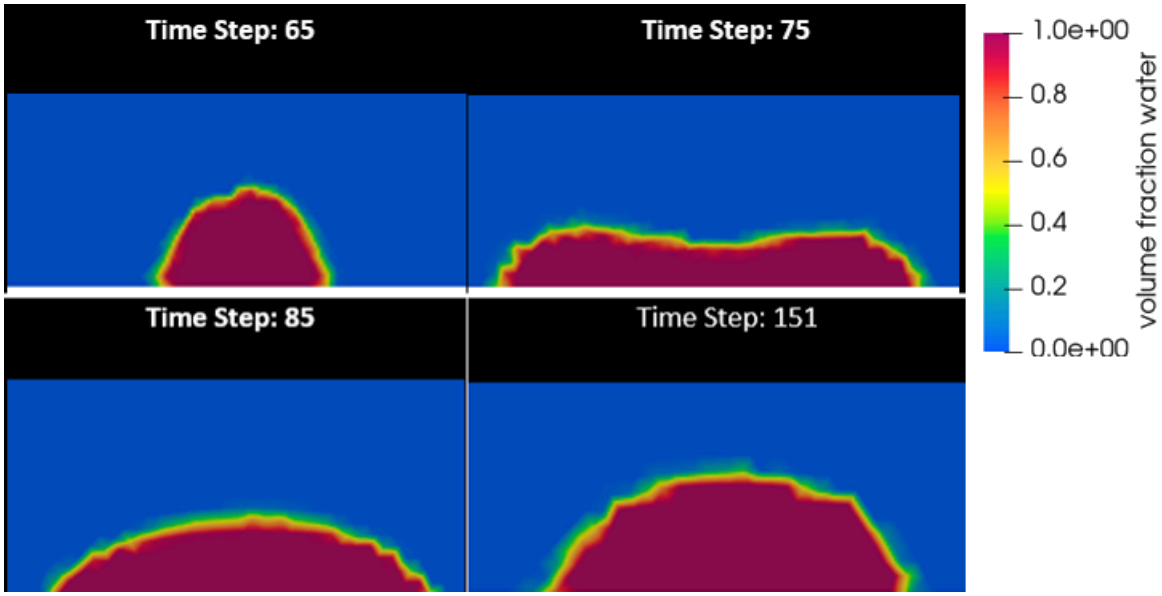


Figure 768:

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving contact angle and surface tension using HyperMesh CFD. You imported the geometry and then defined the simulation parameters and flow boundary conditions. Once the solution was computed, you used HyperMesh CFD Post to create the contours of volume fraction of water

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/f2tre6mry3

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4100 Multiphase Flow using Algebraic Eulerian Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial provides instructions for running a transient simulation of a two-phase flow in a pipe using the Algebraic Eulerian model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4100_LPipe.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 769. The Algebraic Eulerian (AE) model is used to simulate the momentum exchange between a carrier field and a dispersed field. When simulating multiphase flows using the AE model, the carrier field has to be a heavier fluid (or higher density fluid).

In this problem, Water is considered a Carrier field material and Air is considered as Dispersed field material. Fluid enters the Inlet at an Average Velocity of 1 m/sec and the Water and Air volume fractions at the inlet are 96% and 4% respectively.

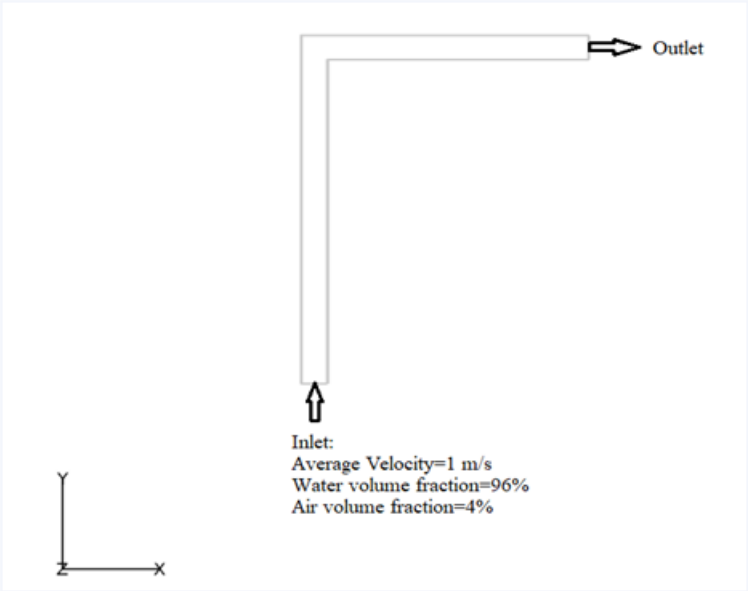


Figure 769:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

   Figure 770:

   The **Open File** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4100_LPipe.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Algebraic_Eulerian` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `LPipe_Disperse` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 771:

Set Up Flow

Create Materials

1. From the Flow ribbon, click the **Material Library** tool.

   Figure 772:

   The **Material Library** dialog opens.
2. Under Settings, click **Disperse Multiphase**, then click the **My Materials** tab.
3. Click **+** to create a new material.
4. Name the material `Air in Water` and assign the properties shown below.

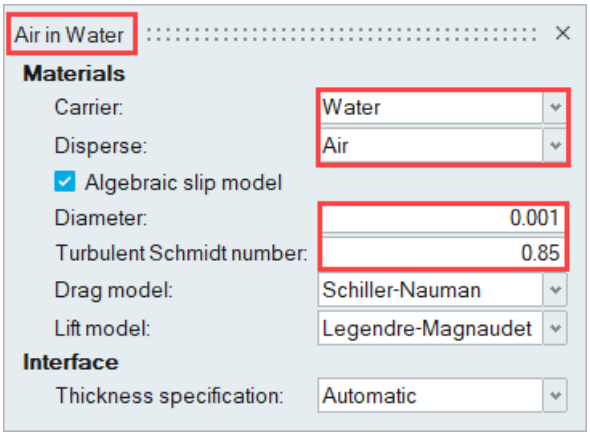


Figure 773:

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

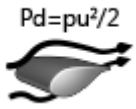


Figure 774:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Activate the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Disperse** and the Disperse material to **Air in Water**
   c) Set the Time step size to `0.01`.
   d) Select **Spalart-Allmaras** as the Turbulence model.
   e) Set the gravity to `-9.81` m/sec^2 in the y direction.

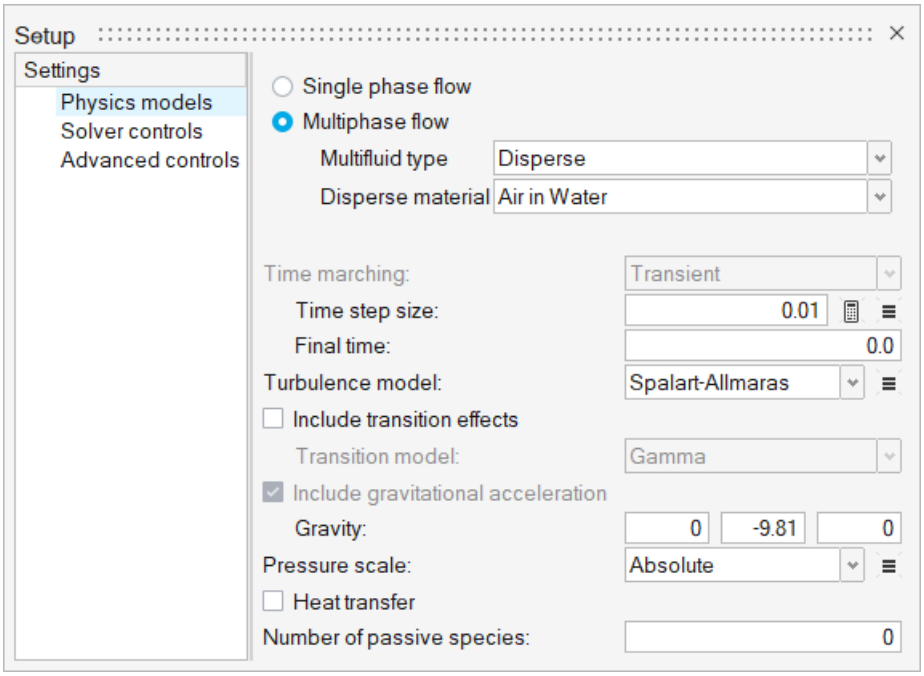


Figure 775:

3. Click the **Solver controls** setting and verify that:

a) Minimum stagger iteration is set to `2`.

b) Maximum stagger iteration is set to `4`.

c) Transient maximum step is set to `150`.

d) The **Flow**, **Multifluid**, and **Turbulence** checkboxes are activated.

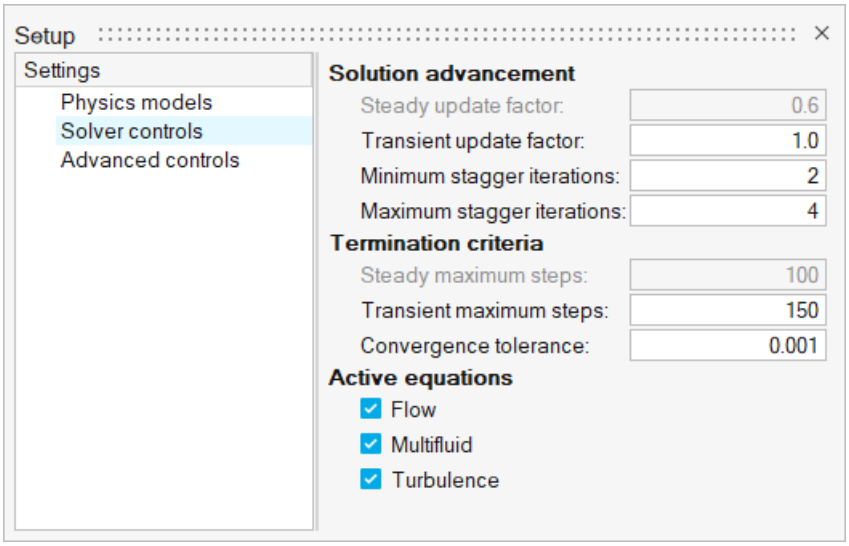


Figure 776:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 777:

2. Select the model solid.
3. Select **Air in Water** from the Material drop-down menu.

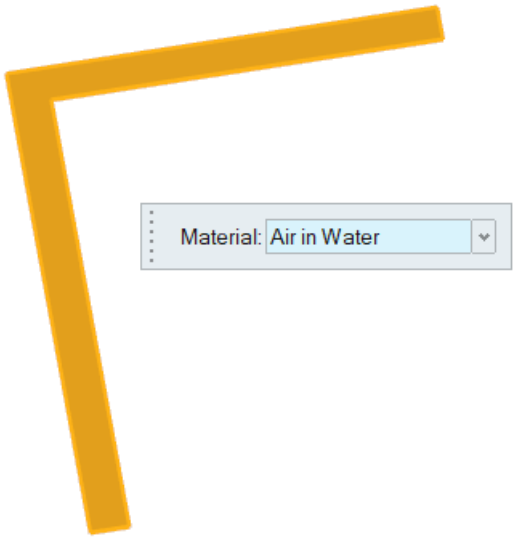


Figure 778:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 779:

2. Click on the inlet face, highlighted in the figure below.

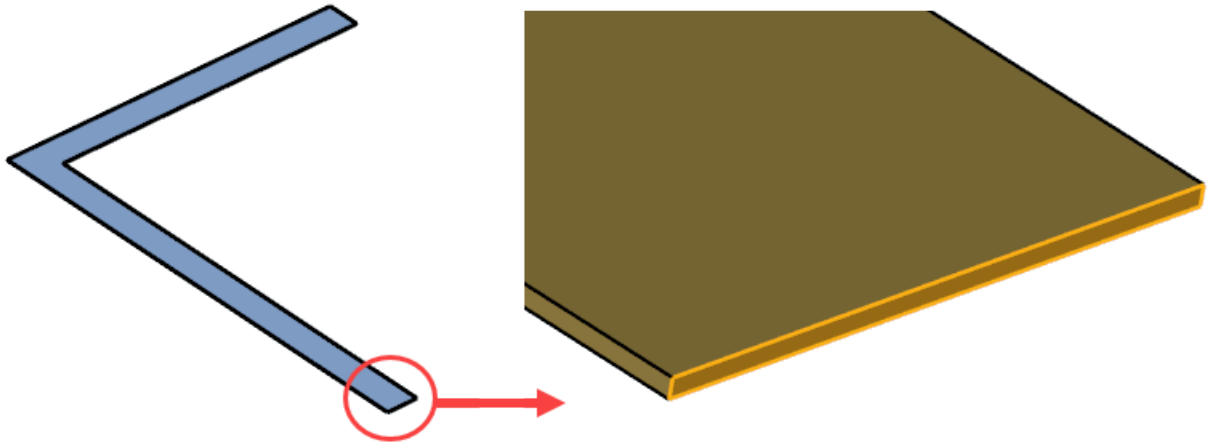


Figure 780:

3. In the microdialog, enter `1.0` for the Average velocity and `0.96` for the Carrier fluid volume fraction.

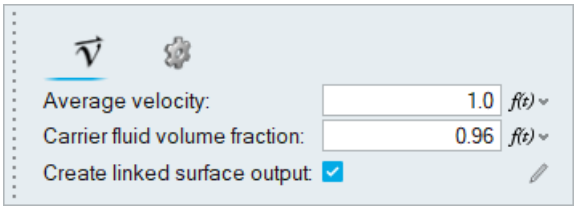


Figure 781:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 782:

6. Select the face highlighted in the figure below then click ✓ on the guide bar.

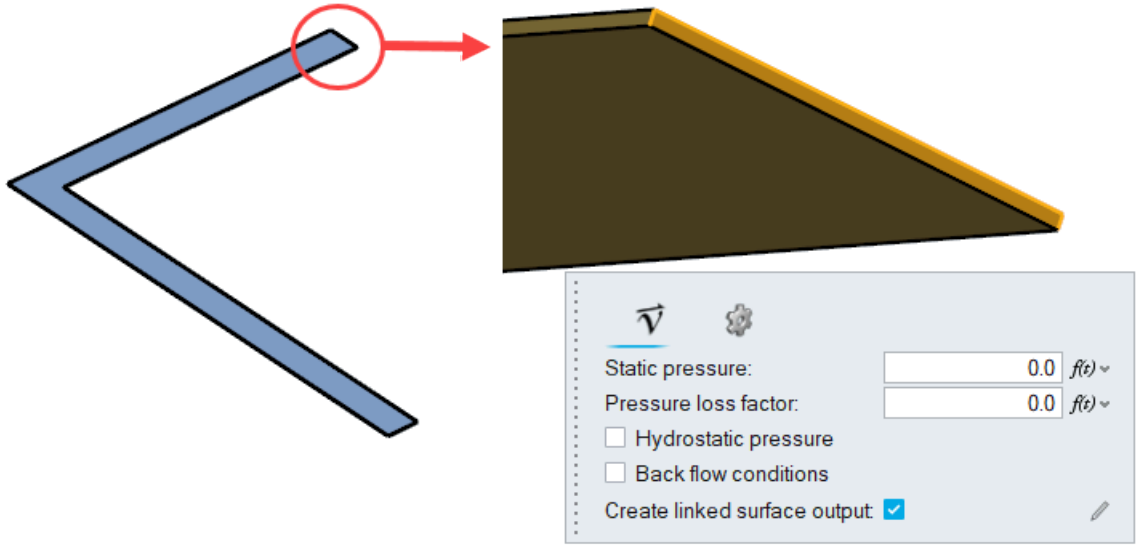


Figure 783:

7. Click the **Slip** tool.

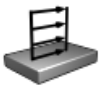


Figure 784:

8. Select the top and bottom faces highlighted below then click ✓ on the guide bar.

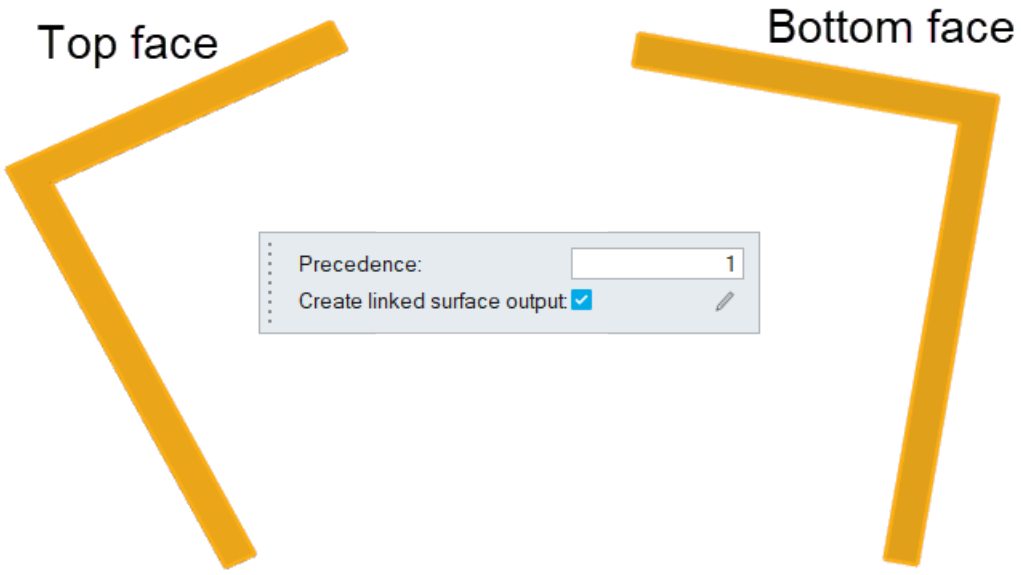


Figure 785:

9. Save the model.

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

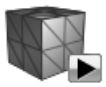


Figure 786:

2. In the **Meshing Operations** dialog, set the Average Element size to `0.00104` (if not set already).

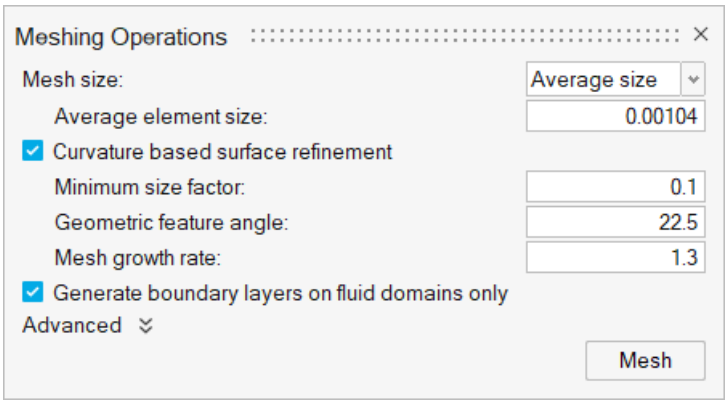


Figure 787:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   > **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 788:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Check the box for **Use hybrid message passing** and set the number of threads to the same as the number of processors.
5. Expand **Default initial conditions** and verify that the Carrier field volume fraction is `1.0`.

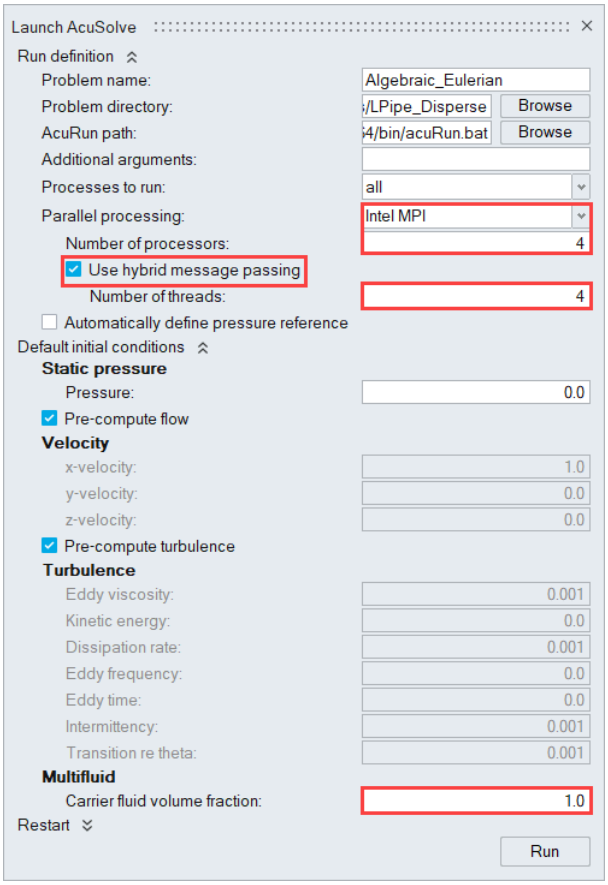


Figure 789:

6. Click **Run** to launch AcuSolve.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

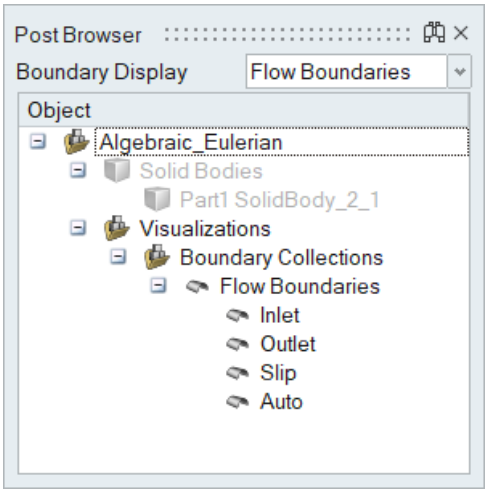


Figure 790:

4. In the Post Browser, right-click on the **Slip** surface and select **Isolate**.
5. Right-click on **Slip** again and select **Edit**.
6. In the microdialog, set the Display option to **Volume fraction air**.
7. Activate the **Legend** radio button then click ≡ and set the Colormap name to **Rainbow Uniform**.

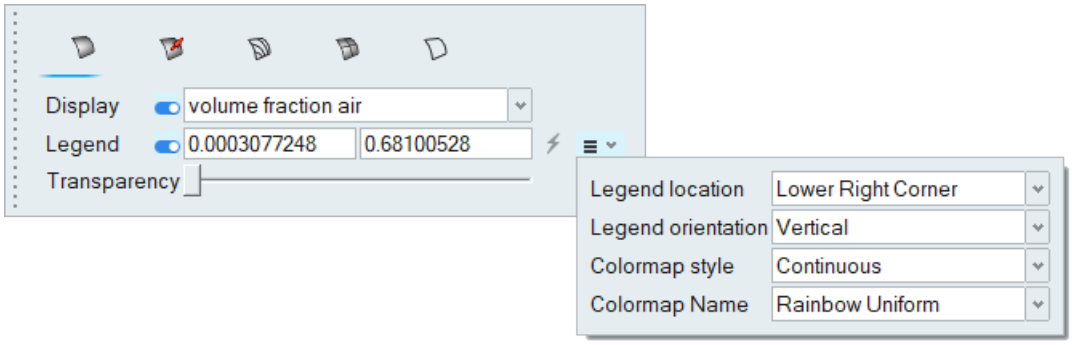


Figure 791:

The volume fraction of air contours are displayed on the model.

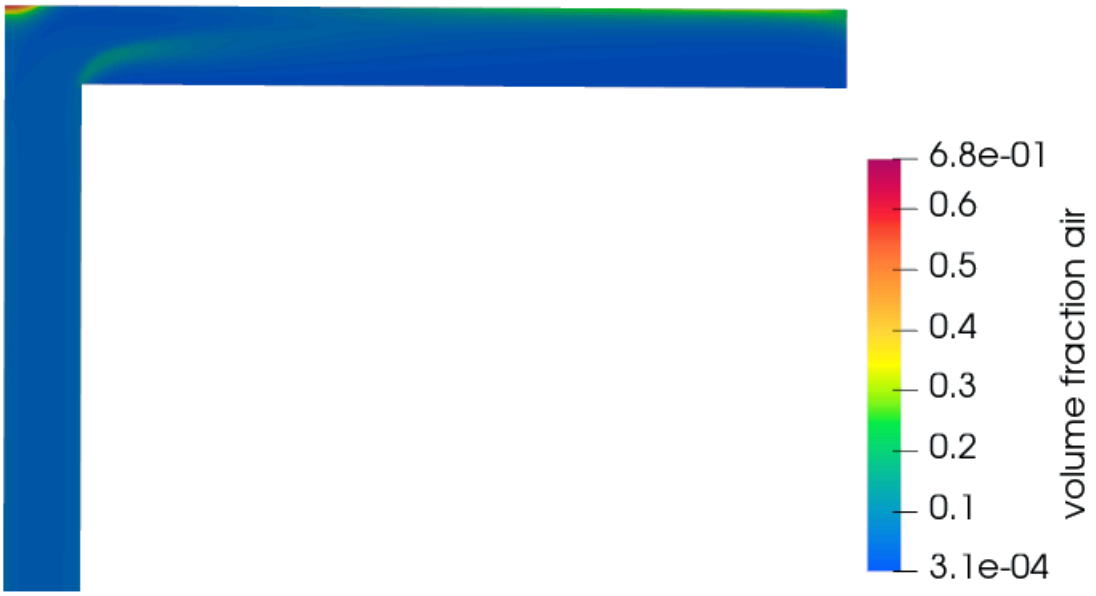


Figure 792:

Summary

In this tutorial, you worked through a basic workflow to set up and solve a transient multiphase flow problem using the Algebraic Eulerian multiphase model in HyperMesh CFD. You started by opening the HyperMesh input file with the geometry and then defined the simulation parameters and flow boundary conditions. Upon completion of the solution by AcuSolve, you used HyperMesh CFD Post to post-process the results and create a contour plot of volume fraction.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from creating the CAD geometry, generating mesh, and setting up the simulation to post-processing the results.
2. Mesh Generation - CAD to Mesh

You can navigate directly to the Mesh Generation section and work from creating the CAD to generating the mesh. Afterwards, you can continue with the solution setup.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/je3luwhw48

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4101 T-Junction Flow using the Eulerian Multiphase Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial provides the instructions for setting up and running a basic transient multiphase simulation using the Eulerian multiphase model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4101_TJunction.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 793. An Air-Water mixture is entering a t-junction pipe through the inlet with a carrier (Water) volume fraction of 0.98. The diameter of the disperse field is 0.001 m. The velocity of the incoming carrier field (Water) is 1.5 m/s, while the velocity of the disperse field (Air) is set to 1.6 m/s. The simulation will be solved as a 2D-problem, therefore a slip condition is applied on the top and bottom surfaces.

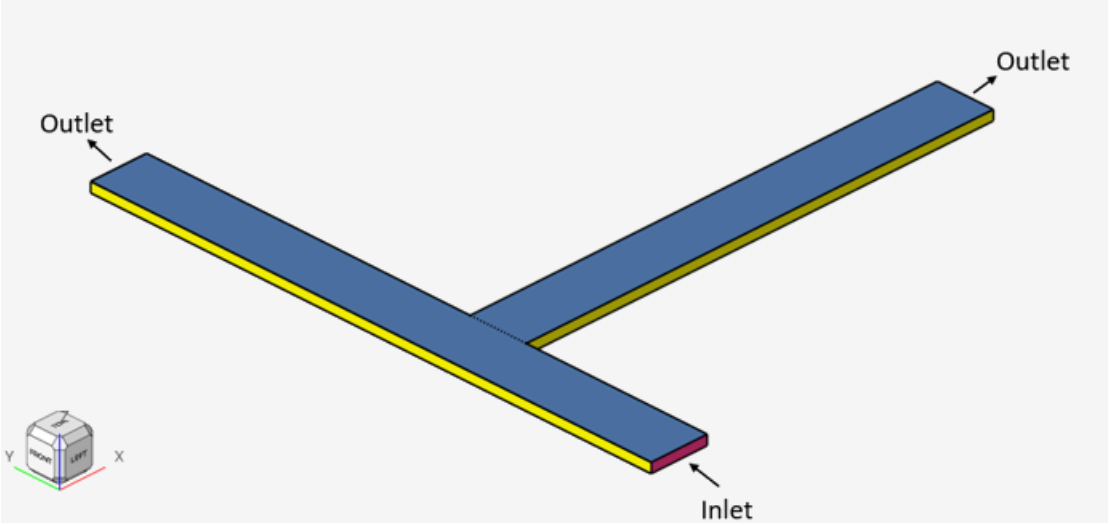


Figure 793:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

   Figure 794:

   The **Open File** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4101_TJunction.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `T_Junction` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `T_Junction` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 795:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

   Figure 796:

   The **Setup** dialog opens.
2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Eulerian**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the **Material Library** dialog, select **Eulerian Multiphase**, switch to the **My Material** tab, then click + to add a new material.
6. In the microdialog, click on the top-left corner and change the name to `WaterAirEul`.
7. Set the Carrier field to **Water** and the Disperse field to **Air**.
8. Set the diameter, drag model, and other parameters as shown in the image below.

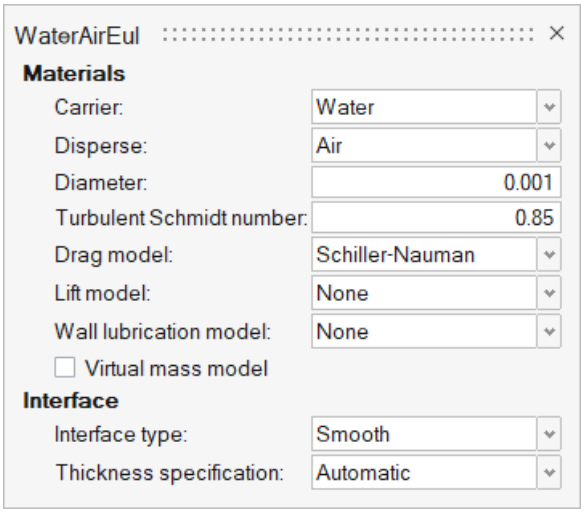


Figure 797:

9. Close the material model microdialog and then close the **Material Library** dialog.
10. In the Setup dialog, set the Eulerian Material to **WaterAirEul**.
11. Set Time step size and Final time to `0.01` and `1`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
12. Set the gravity to `0, -9.81, 0` and the pressure scale to **Absolute**.

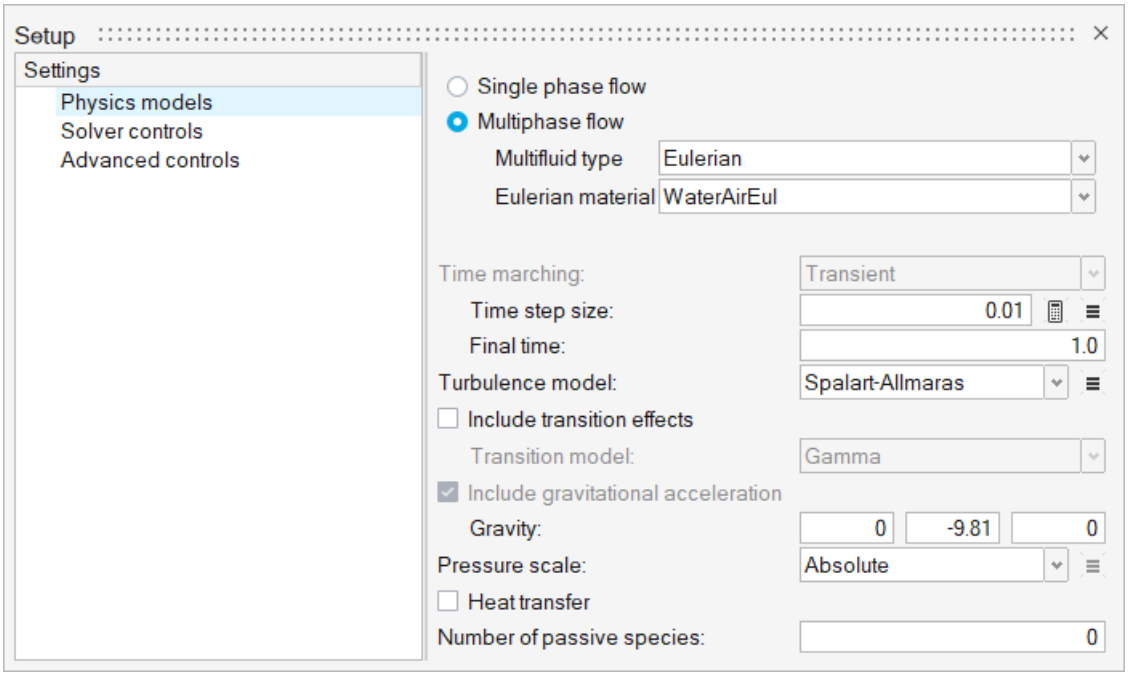


Figure 798:

13. Click the **Solver controls** setting and set the Minimum and Maximum stagger iterations to `2` and `4`, respectively.

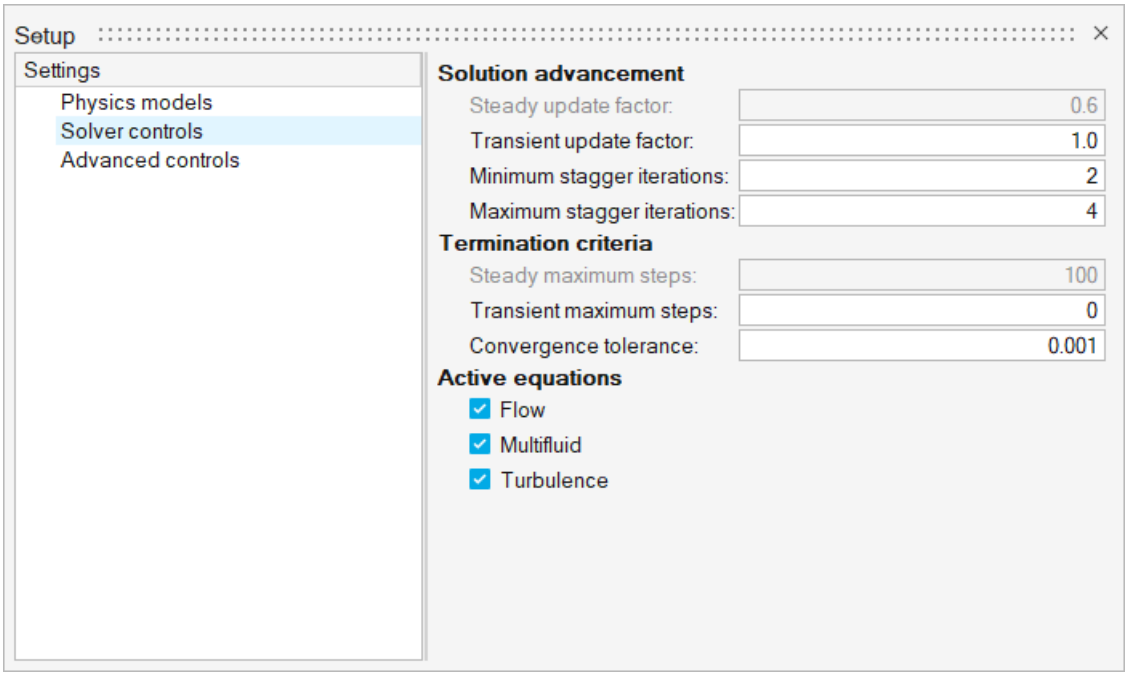


Figure 799:

14. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 800:

2. Verify that WaterAirEul has been assigned as the material.

3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 801:

2. Click the inlet face highlighted in the figure below.

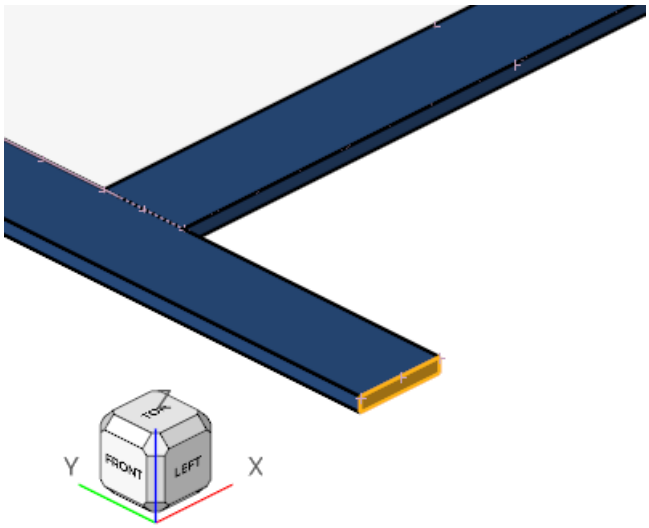


Figure 802:

3. In the microdialog, activate **Phasic inflow** and then change the inflow velocity type to **Normal**.
4. Enter a value of `1.5` and `1.6` m/s for the Normal velocity of the Carrier Field and Disperse Field, respectively.
5. Set the Volume fraction specification type to **Value** and set the Carrier fluid volume fraction to `0.98`.

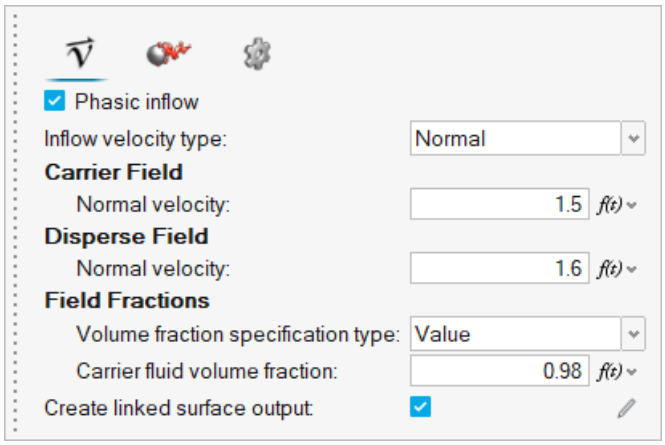


Figure 803:

6. Click the **Turbulence** tab in the microdialog. Set the Turbulence input type to **Direct** and the Eddy viscosity to `0.001`.

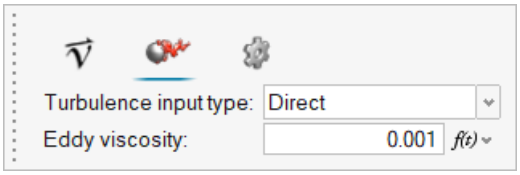


Figure 804:

> **Note:** When phasic inflow is active, Turbulence input type = Direct should be used.

7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Click the **Outlet** tool.

Figure 805:

9. Select the face highlighted below and verify the settings in the microdialog.

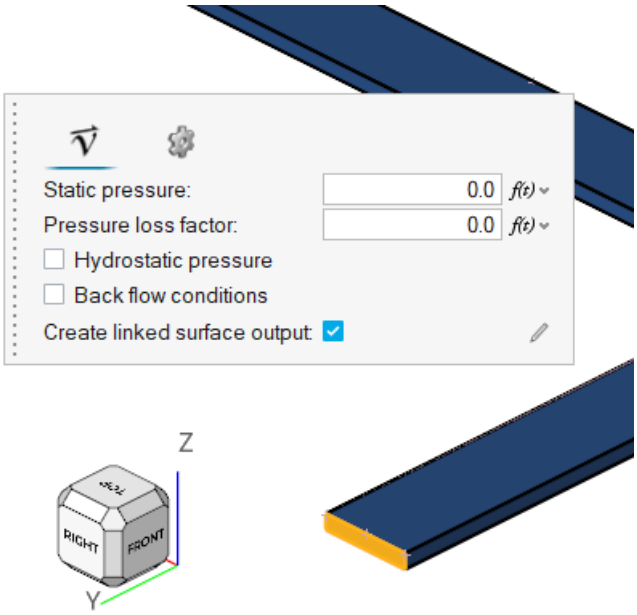


Figure 806:

10. On the guide bar, click ▸ to execute the command and remain in the tool.
11. With the Outlet tool still active, select the face highlighted below after rotating the model. Then click ✓ on the guide bar.

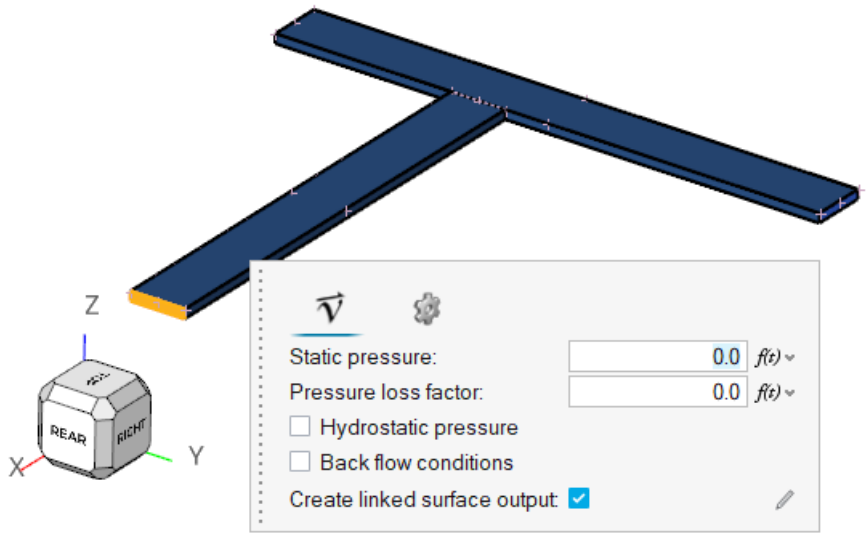


Figure 807:

12. Click the **Slip** tool.

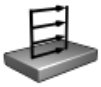


Figure 808:

13. Select the top and bottom faces highlighted below then click ✓ on the guide bar.

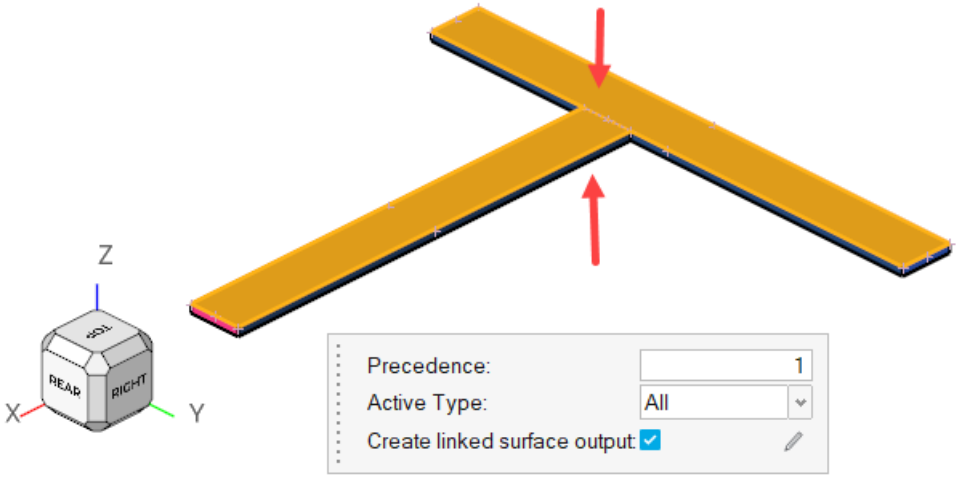


Figure 809:

14. Save the model.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

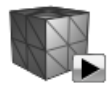


Figure 810:

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. In the **Meshing Operations** dialog, check that the Average Element size is set to `0.0025`.

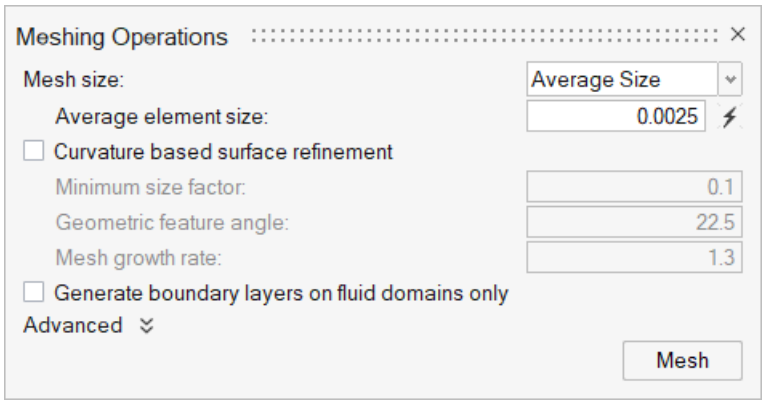


Figure 811:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

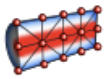


Figure 812:

The **Field Output** dialog opens.

2. Check the box for **Write Initial Conditions**.
3. Set the time interval to `1`.

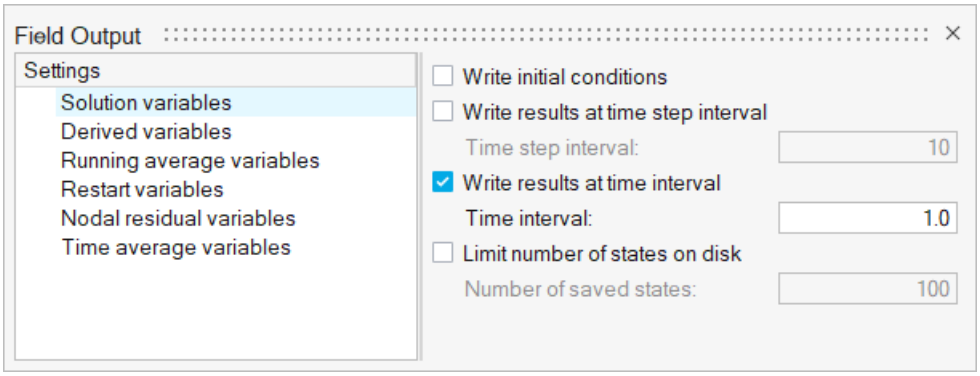


Figure 813:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 814:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial conditions**, uncheck **Pre-compute flow**, and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.

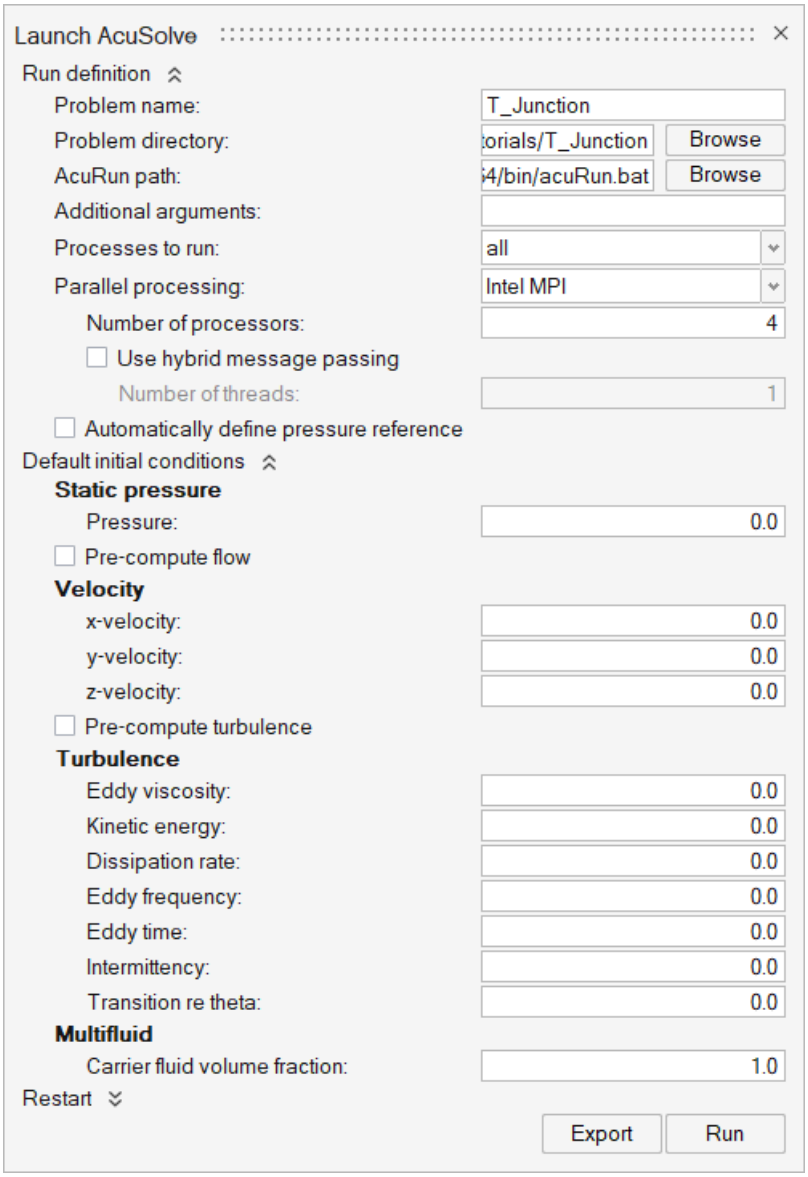


Figure 815:

5. Click **Run** to launch AcuSolve.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

In this step, you will create contour plots for velocity and field volume fraction.

1. Once the solution is complete, right-click the AcuSolve run in the **Run Status** dialog and select **Visualize results**.
2. Once the results are loaded in the Post ribbon, click the **Top** face of the view cube to orient to the xy-plane.
3. Click the **Boundary Groups** tool.

Figure 816:

4. In the modeling window, select the top slip surface.
5. In the microdialog, set the display variable to **velocity**.
6. Activate the **Legend** toggle and set the legend limits to `0` and `2.5`, respectively.
7. Click ≡ and set the colormap properties as shown below.

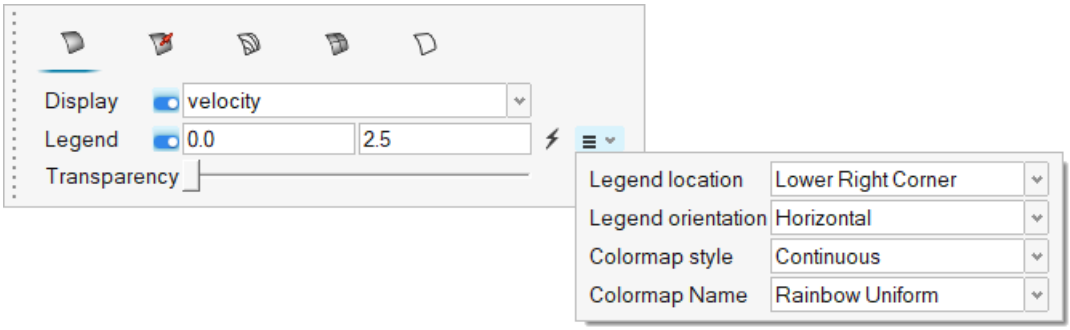


Figure 817:

8. Click ✓ on the guide bar to create the velocity contour plot.

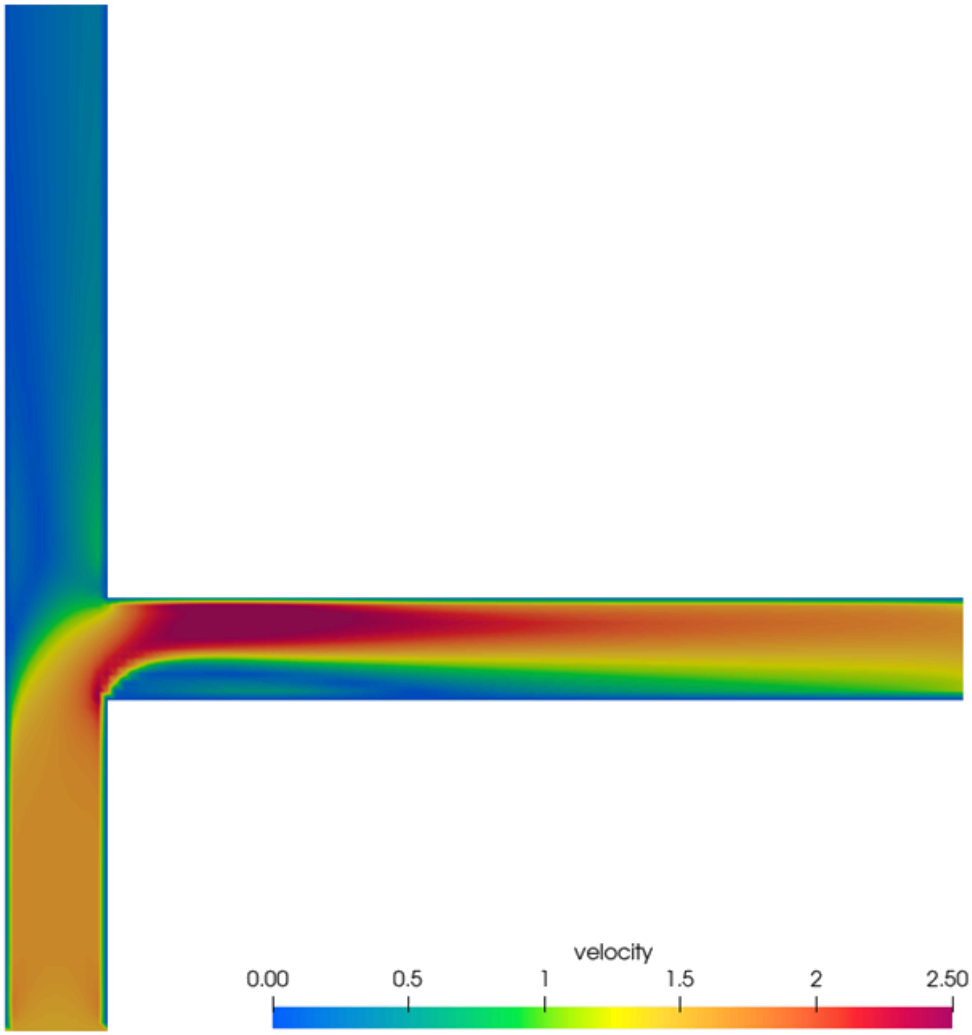


Figure 818:

9. In the Post Browser, right-click **Boundary Group 7** and select **Edit**.
10. In the microdialog, change the display variable to **volume fraction: air** and set the legend limits to `0` and `0.1`.
11. Click ✓ on the guide bar to create the volume fraction contour plot.

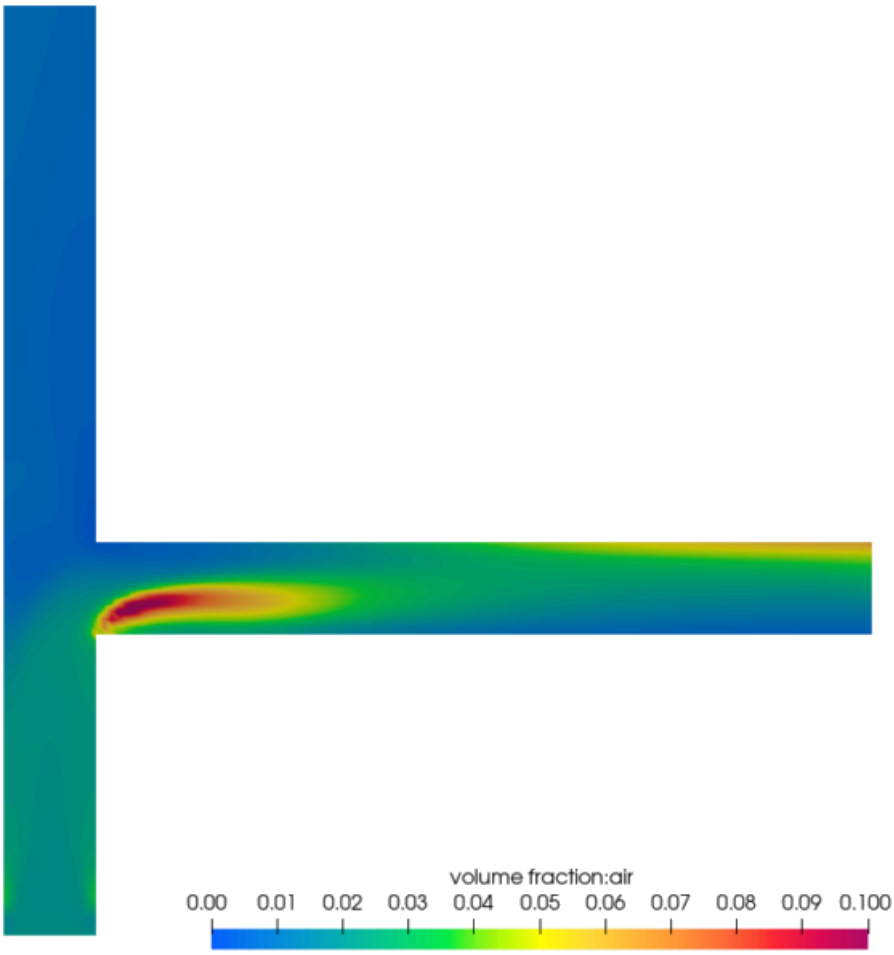


Figure 819:

Summary

In this tutorial, you learned how to set up and solve a multiphase flow simulation using the Eulerian multiphase model available in AcuSolve using HyperMesh CFD. You started by importing the HyperMesh CFD input database and then defined the flow setup. Once the solution was computed, you created a contour plot of velocity and field volume fraction using HyperMesh CFD Post.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/stctobyvem

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4102 Fluidized Bed using the Granular Multiphase Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up and running a gas-solid fluidized bed simulation using the granular multiphase model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4102_FluidizedBed.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in . Some of the most common applications of the granular multiphase model are fluidized beds, mixing tanks, sedimentation of particles in liquid columns, and so on. In this tutorial, you will set up and run a gas-particle fluidized bed. The inlet velocity used for the simulation is 0.3 m/s and the initial bed height is 0.2 m. The initial solid packing fraction is set to 0.63.

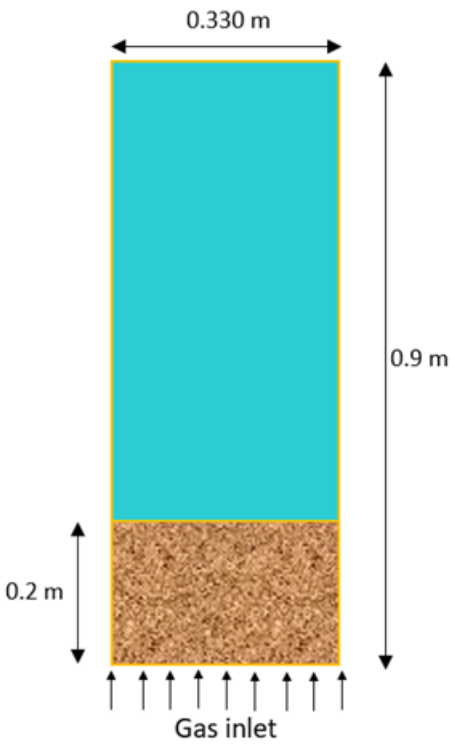


Figure 820:

For simplicity, the gas and solid phase material models have already been defined in the input file that is provided. The steps for creating the granular multiphase material model are included in the tutorial. For information about creating fluid and solid material models, refer to the introductory tutorials. The summary of the material models have been listed below.

Phase	**Density (kg/m^3)**	**Viscosity (Pa s)**
Gas	21.56	1.781e-05
Particle	910	-

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 821:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4102_FluidizedBed.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Fluidized_Bed` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Fluidized_Bed` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 822:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

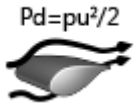


Figure 823:

The **Setup** dialog opens.

2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Granular**.
4. Click the Granular material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the **Material Library** dialog, select **Granular Multiphase**, switch to the **My Material** tab, then click + to add a new material.
6. In the microdialog, click on the top-left corner and change the name to `gas-particle`.
7. Set the Carrier field to **gas** and the Disperse field to **particle**.

   Note: The input file provided with the tutorial has a predefined gas and particle material models in it.

8. Set the diameter, drag model, and other granular parameters as shown in the image below.

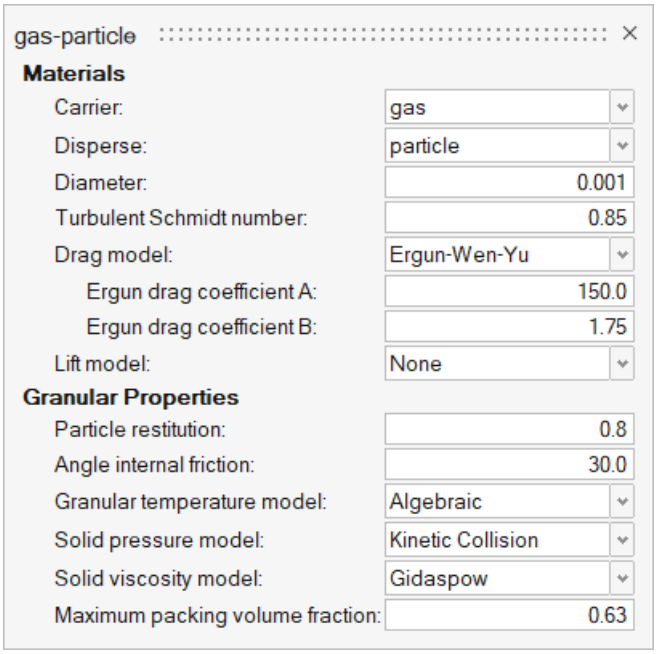


Figure 824:

9. Close the material model microdialog and then close the **Material Library** dialog.
10. In the Setup dialog, set the Granular Material to **gas-particle**.
11. Set Time step size and Final time to `0.005` and `7`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
12. Set the gravity to `0, -9.81, 0` and the pressure scale to **Absolute**.

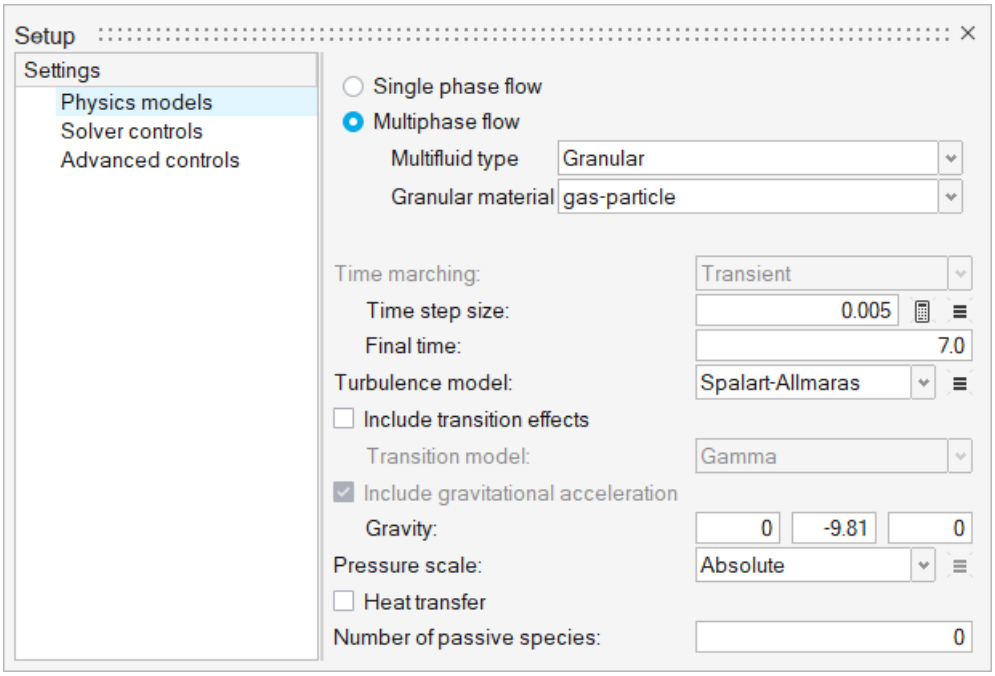


Figure 825:

13. Click the **Solver controls** setting and set the Minimum and Maximum stagger iterations to `2` and `4`, respectively.

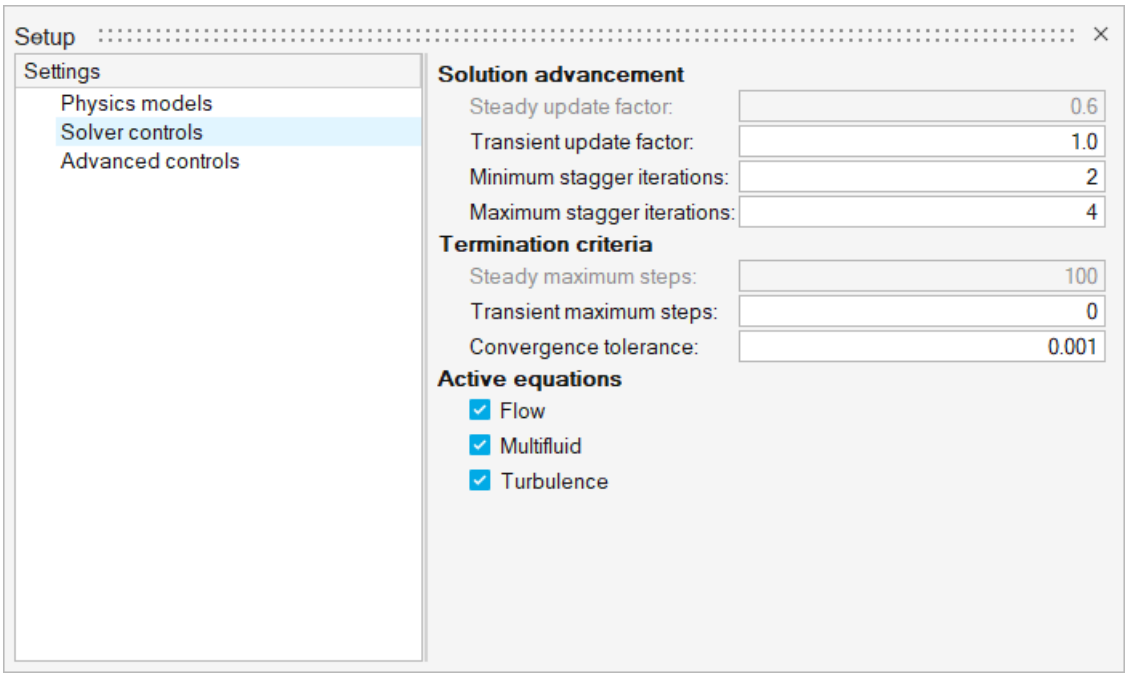


Figure 826:

14. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 827:

2. Verify that gas-particle has been assigned as the material.
3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 828:

2. Click the inlet face highlighted in the figure below.

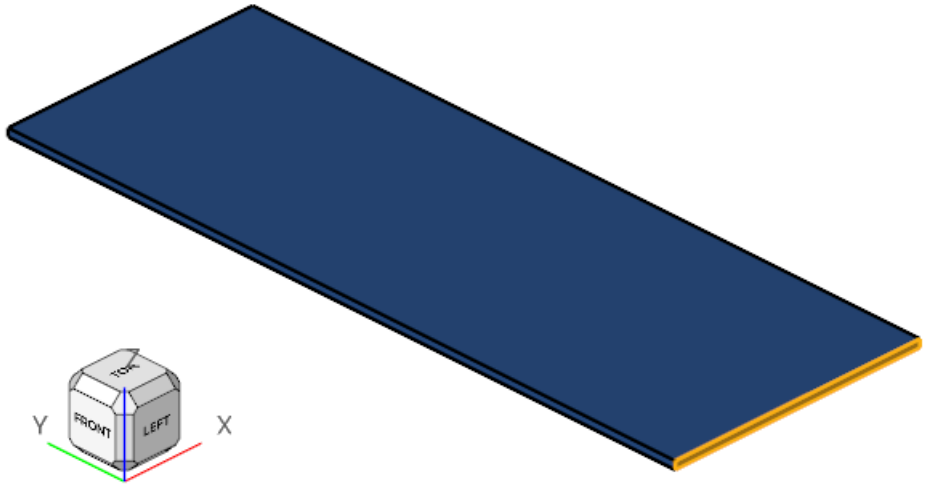


Figure 829:

3. In the microdialog, enter the values shown in the figure below.

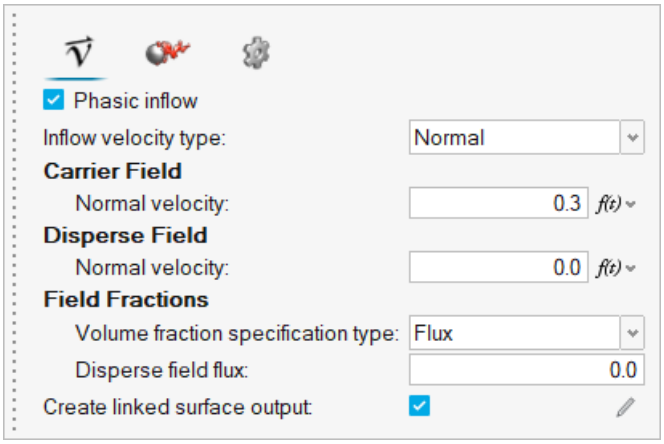


Figure 830:

4. In the turbulence tab, set the turbulence input type to **Direct** and set the eddy viscosity value to `0.0001`.

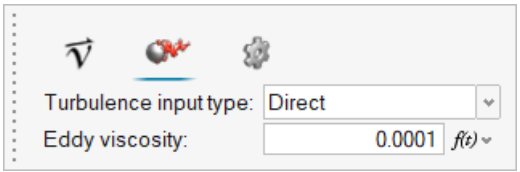


Figure 831:

5. On the guide bar, click ✓ to execute the command and exit the tool.
6. Click the **Outlet** tool.

Figure 832:

7. Select the face highlighted below and verify the settings in the microdialog.

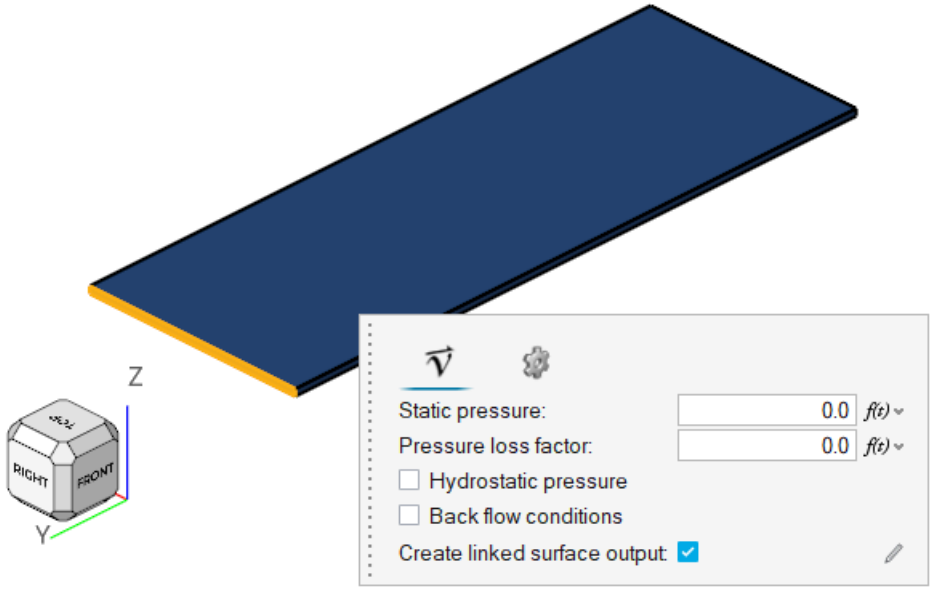


Figure 833:

8. Click ✓ on the guide bar.
9. Click the **Slip** tool.

Figure 834:

10. Select the top and bottom faces highlighted below then click ✓ on the guide bar.

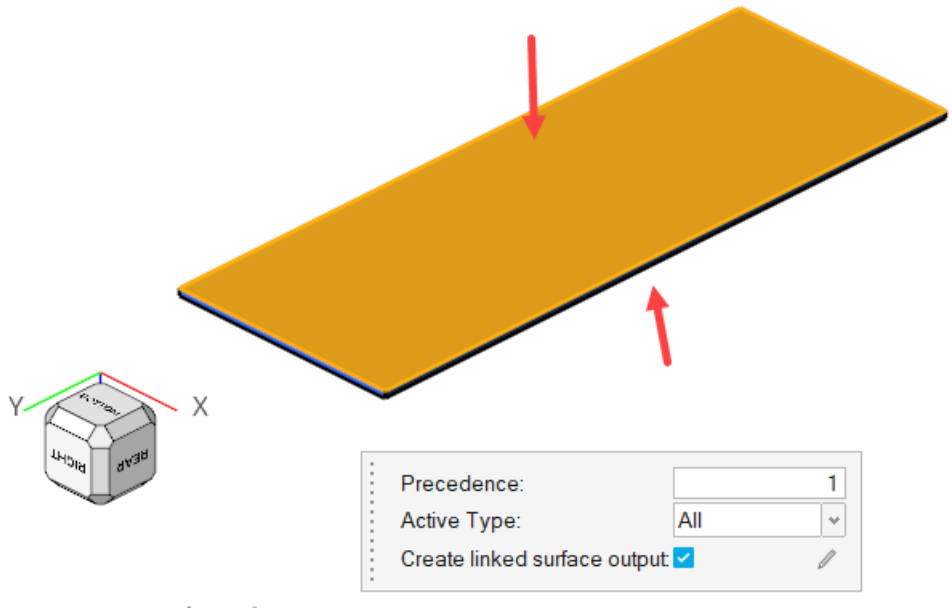


Figure 835:

11. Save the model.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

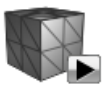


Figure 836:

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. In the **Meshing Operations** dialog, check that the Average Element size is set to `0.01`.

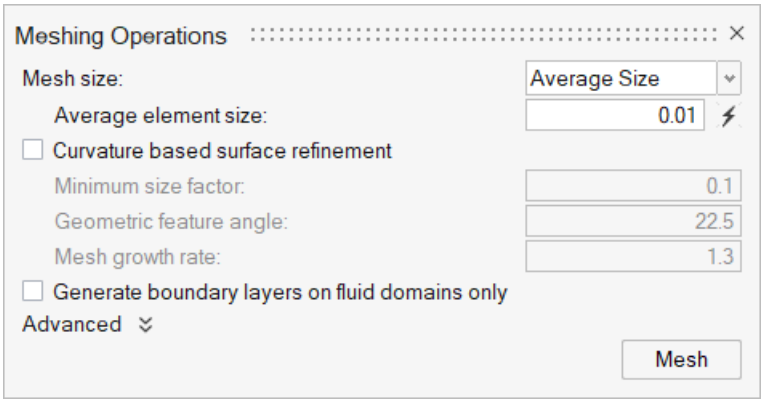


Figure 837:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

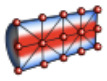


Figure 838:

   The **Field Output** dialog opens.
2. Check the box for **Write Initial Conditions**.
3. Set the time interval to `10`.

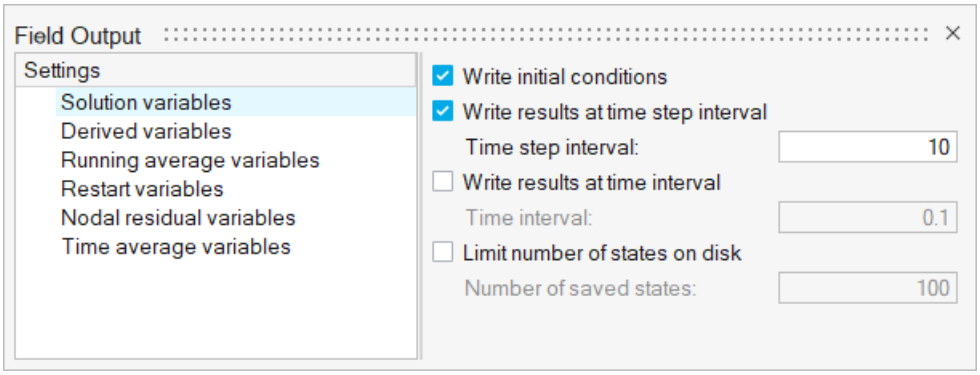


Figure 839:

Define the Nodal Initial Conditions

In this step, you'll define the nodal initial condition for the volume fraction of the gas and solid phases.

1. From the Solution ribbon, click the **Plane** tool.

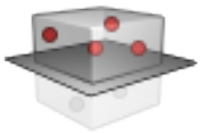


Figure 840:

2. Select the solid in the modeling window.
3. On the guide bar, change the active selection to **Plane**.
4. Click anywhere on the solid body to define the plane location. By default, the plane normal is aligned with the y-axis.
5. In the variable dialog, click + in the top-left corner and select **Carrier Volume Fraction**.

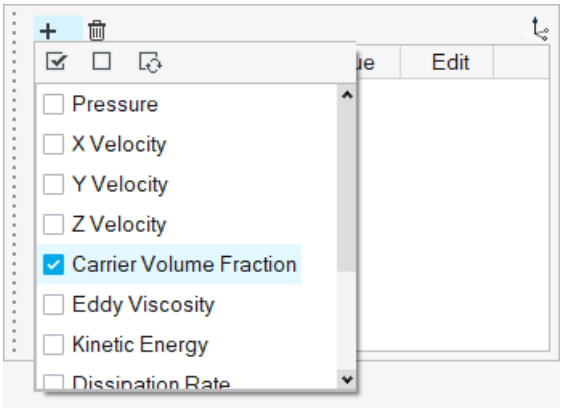


Figure 841:

6. Click on the white space in the dialog and set the value of the carrier volume fraction to `0.37`.
7. Click in the top-right corner of the dialog.
   The Vector tool appears, which can be used change the location and orientation of the plane defining the initial condition.
8. In the Vector tool, verify that the orientation of the tool is along the negative y-axis then click **XYZ**.

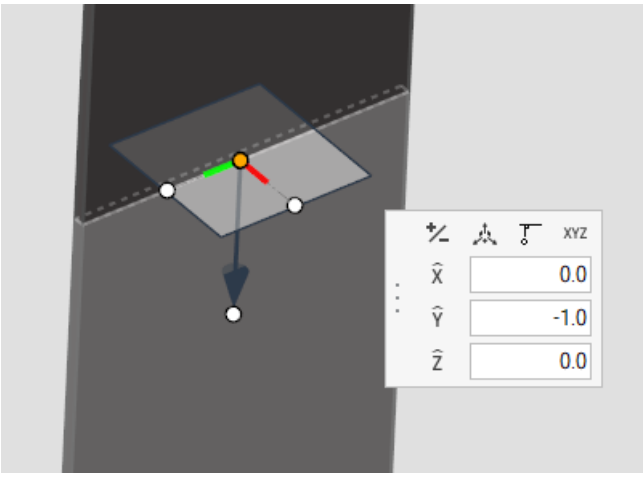


Figure 842:

9. Enter the coordinates of the center of plane as shown in the figure below.

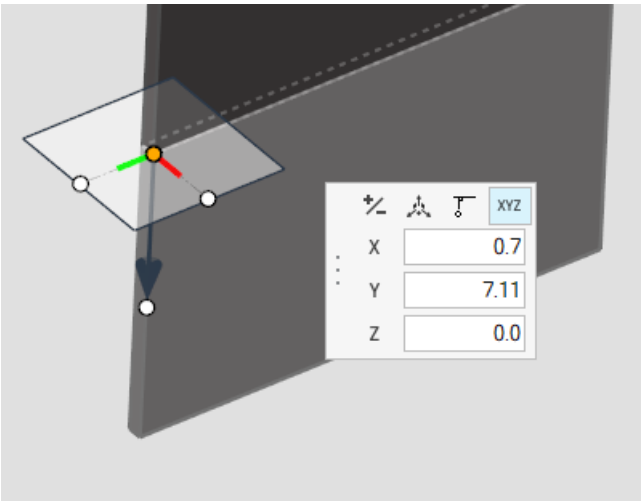


Figure 843:

10. Click ✓ on the guide bar then save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 844:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial conditions**, uncheck **Pre-compute flow**, and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.

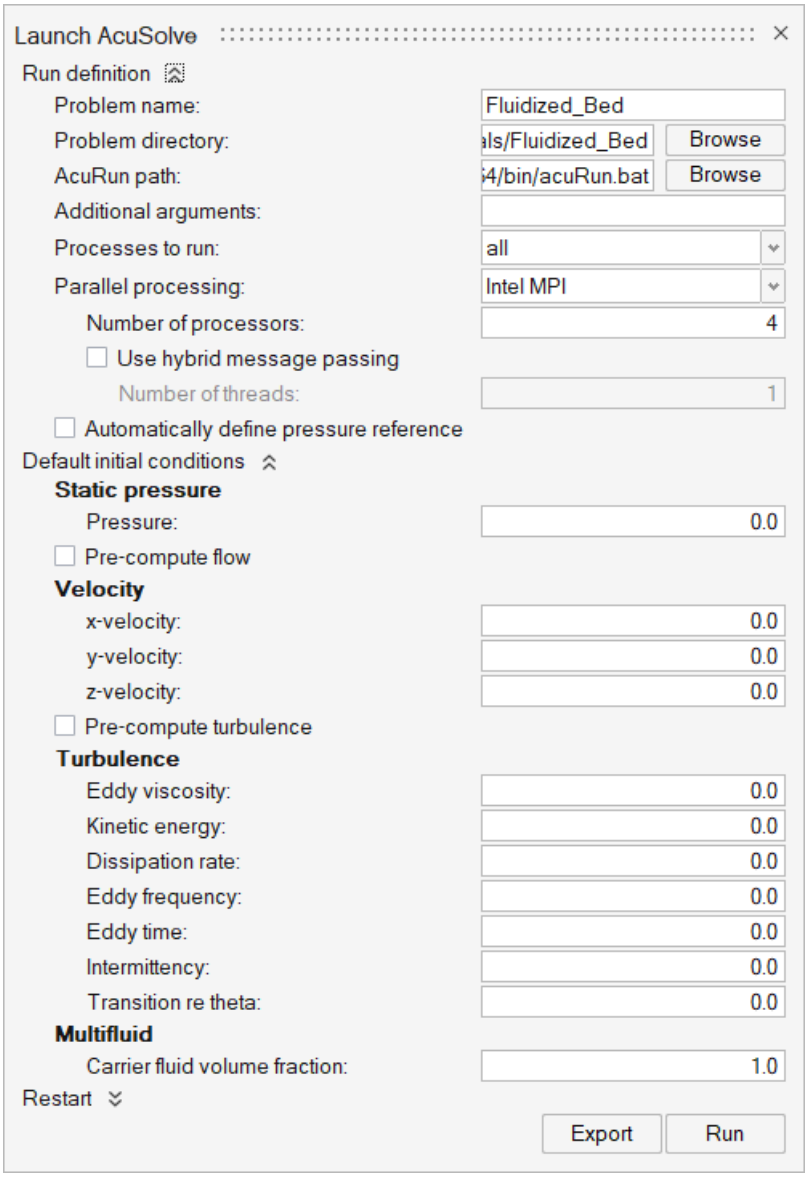


Figure 845:

5. Click **Run** to launch AcuSolve.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

In this step, you will create a contour plot for the particle volume fraction.

1. Once the solution is complete, right-click the AcuSolve run in the **Run Status** dialog and select **Visualize results**.
2. Once the results are loaded in the Post ribbon, click the **Top** face of the view cube to orient to the xy-plane.
3. Click the **Boundary Groups** tool.

Figure 846:

4. In the modeling window, select the top slip surface.
5. In the microdialog, set the display variable to **volume fraction:particle**.
6. Activate the **Legend** toggle and set the legend limits to `0` and `0.63`, respectively (if not already set).
7. Click ≡ and set the colormap properties as shown below.

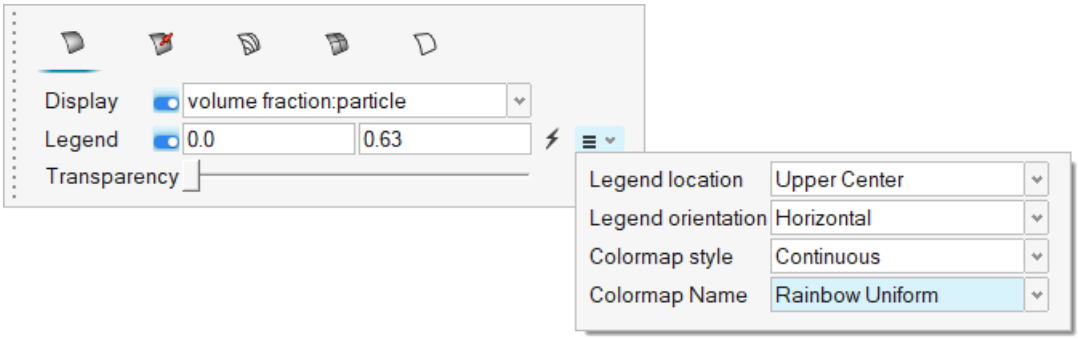


Figure 847:

8. Click ✓ on the guide bar to create the volume fraction contour plot.
9. Click the play icon at the bottom of the modeling window to play the animation.

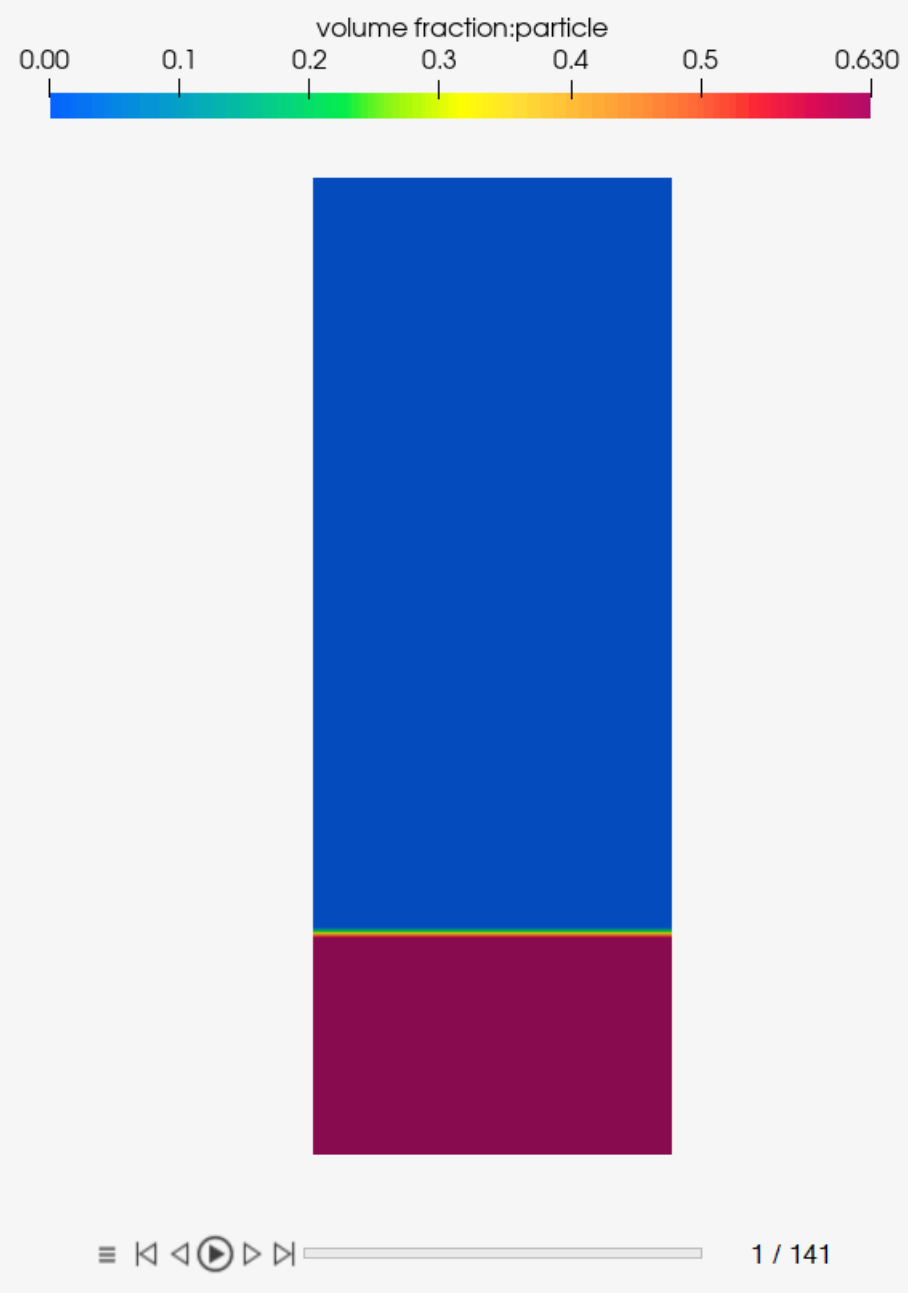


Figure 848:

Summary

In this tutorial, you learned how to set up and solve a fluidized bed simulation using the Granular multiphase model available in AcuSolve using HyperMesh CFD. You started by importing the HyperMesh CFD input database and then defined the flow setup. Once the solution was computed, you created a contour plot of particle volume fraction using HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/ls5y4eskx6

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4200 Humidity – Pipe Junction

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up and running a basic transient humidity transport simulation using a pipe junction model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4200_Humidity.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 849. As an example, a pipe junction problem is attached here to show the capability of the Humidity modelling in AcuSolve. In this problem, there are two inlets with different flow, thermal, and humidity conditions. As the flow proceeds downstream of the pipe, two pipes merge into a single pipe to create a single outlet and a distinct profile of temperature and humidity is attained. The geometry is symmetric about the XZ midplane of the pipe, as shown in the figure.

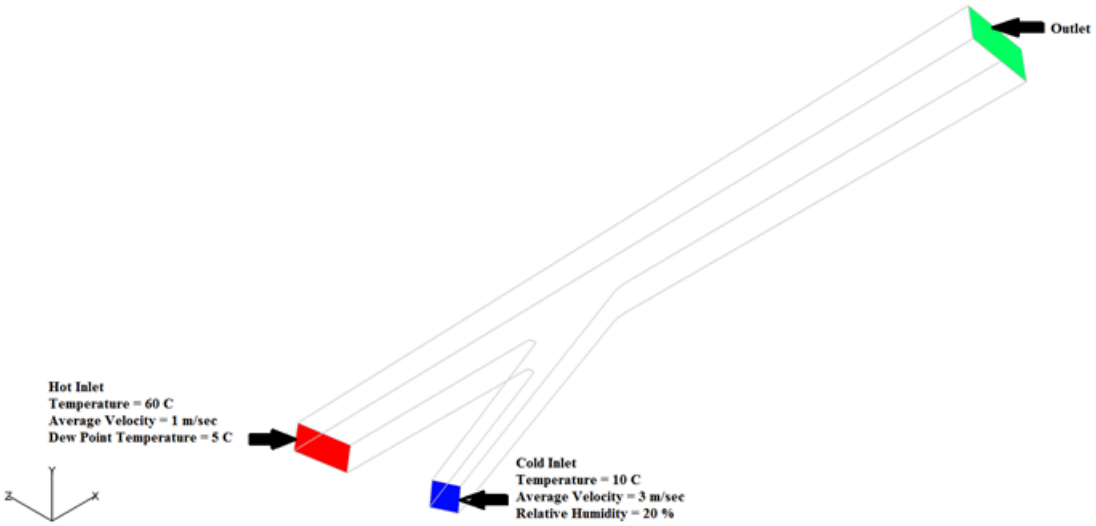


Figure 849:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 850:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4200_Humidity.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Humidity` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Humidity` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 851:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

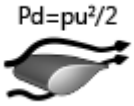
   

   Figure 852:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Select the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Humidity transport**.
   c) Set the Time step size to `1` s and the Final time to `50` s.
   d) Set the Turbulence model to **Spalart-Allmaras**.
   e) Set the Gravity to `(0,-9.81,0)`.
   f) Set the Pressure scale to **Gauge** and click ≡. In the microdialog, set the Absolute pressure offset to `101325` Pa then press **Esc**.

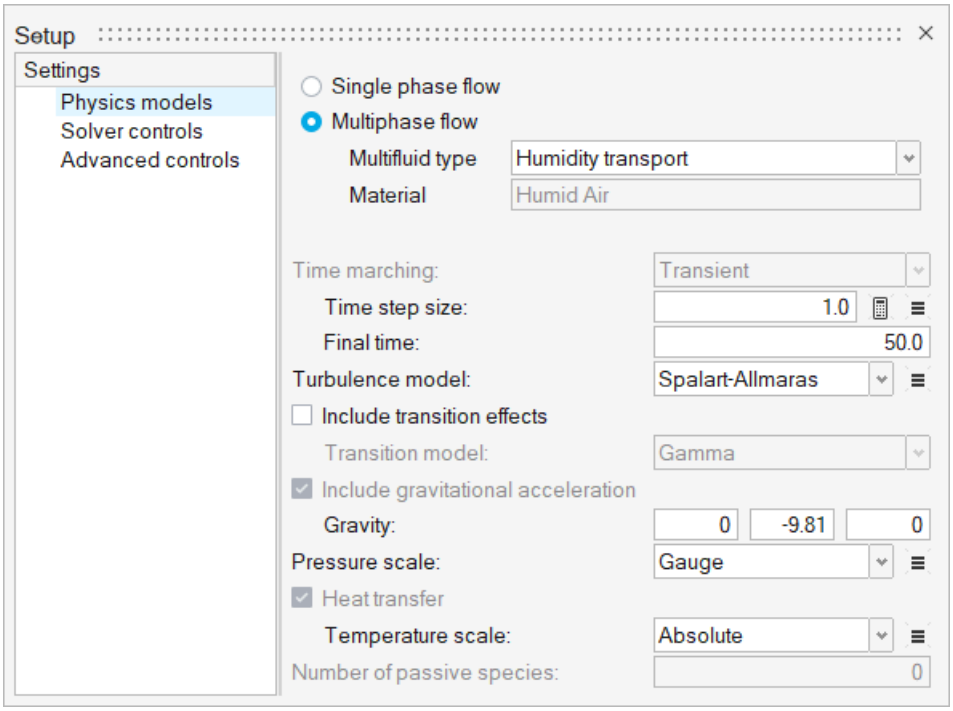


Figure 853:

3. Click the **Solver controls** setting.
4. Set both the Minimum and Maximum stagger iterations to `2`.

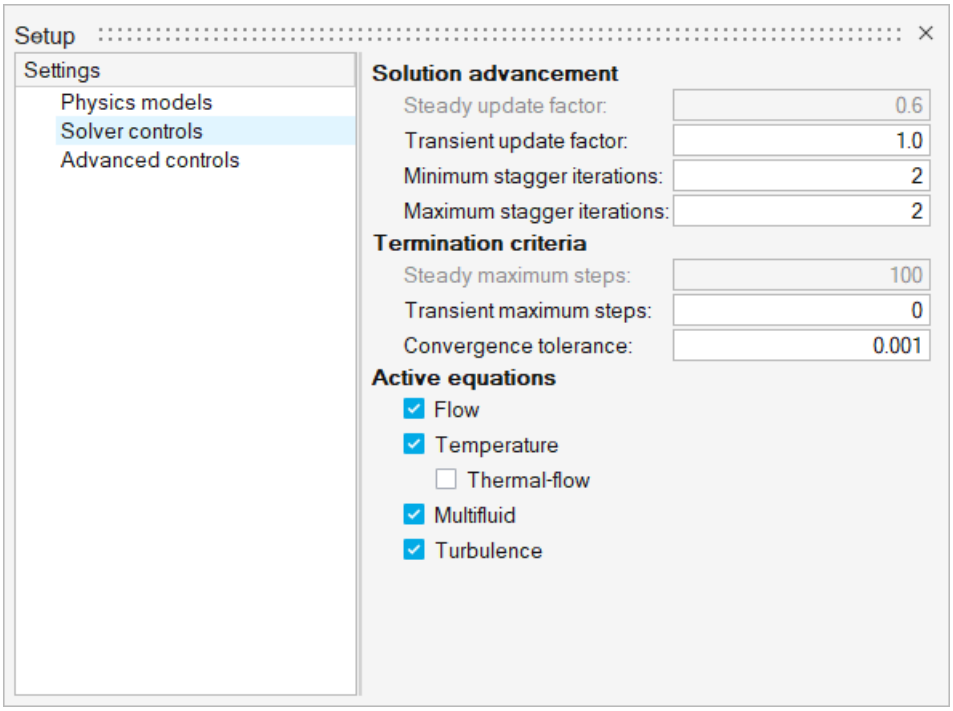


Figure 854:

5. Close the dialog and save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 855:

2. Select the surface highlighted in the figure below.
3. In the microdialog:
   a) Set the Average velocity to `1` m/sec.
   b) Set the Temperature to `333.15` K.
   c) Set the Humidity input type to **Dewpoint Temperature**.
   d) Set the Dewpoint temperature to `278.15` K.

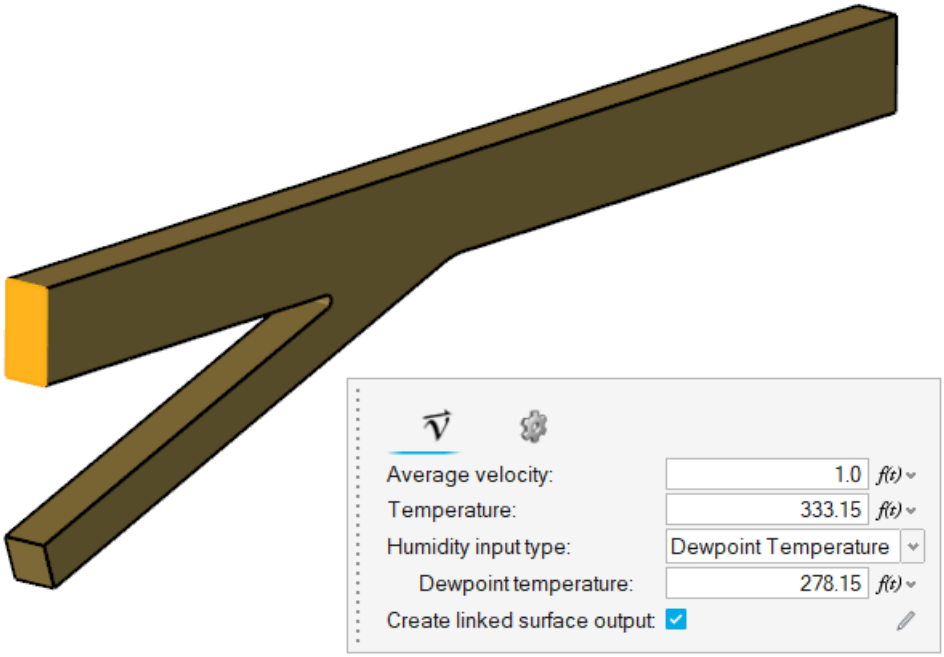


Figure 856:

4. In the Boundaries legend, double-click on **Inlet**, change the name to `Hot_inlet`, then press **Enter**.
5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. Select the surface highlighted in the figure below.

7. In the microdialog:

a) Set the Average velocity to `3` m/sec.

b) Set the Temperature to `283.15` K.

c) Set the Humidity input type to **Relative Humidity**.

d) Set the Relative humidity to `20`.

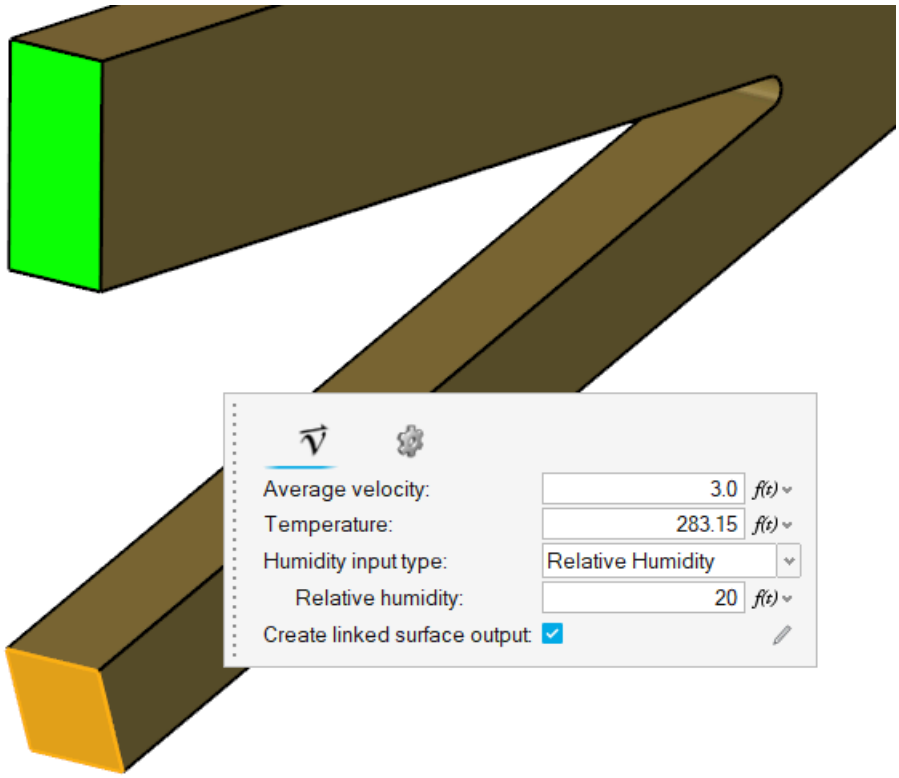


Figure 857:

8. In the Boundaries legend, double-click on **Inlet**, change the name to `Cold_inlet`, then press **Enter**.

9. On the guide bar, click ✓ to execute the command and exit the tool.

10. Click the **Outlet** tool.

Figure 858:

11. Select the surface highlighted in the figure below then click ✓ on the guide bar.

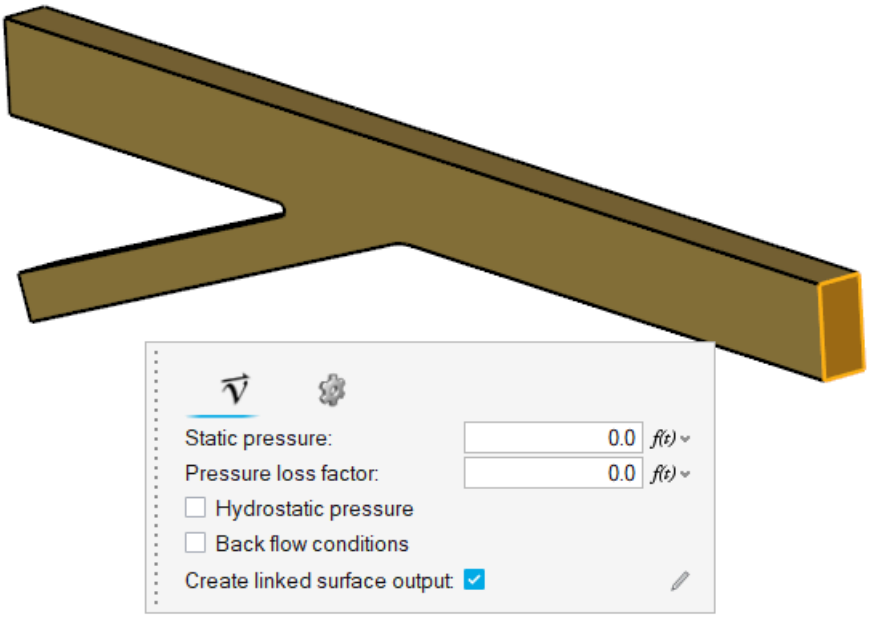


Figure 859:

12. Click the **Slip** tool.

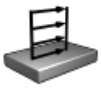


Figure 860:

13. Select the surface highlighted in the figure below (the surface with the minimum y-coordinate).

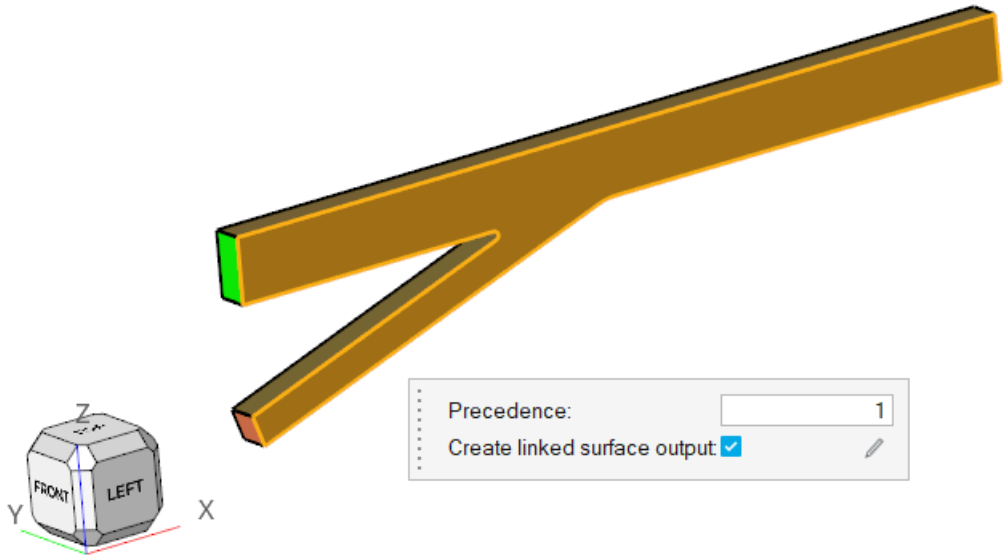


Figure 861:

14. In the Boundaries legend, double-click on **Slip**, change the name to `y_neg`, then press **Enter**.

15. Click ▸ on the guide bar.

16. Select the surface highlighted in the figure below (the surface with the maximum y-coordinate).

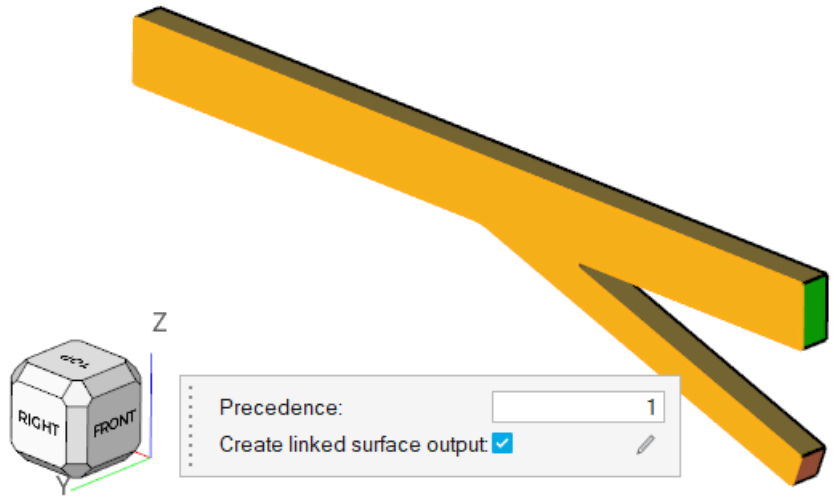


Figure 862:

17. In the Boundaries legend, double-click on **Slip**, change the name to `y_pos`, then press **Enter**.

18. Click ✓ on the guide bar.

19. Save the model.

Compute the Solution

The input HyperMesh database contains the mesh, hence you do not need to generate the mesh again.

Define the Nodal Initial Conditions

1. From the Solution ribbon, click the **Part** tool.

Figure 863:

2. Select the pipe solid.
3. In the dialog, click **+**, select **Relative Humidity** and **Temperature** from the list of variables, then click on the white space in the dialog.
4. Set the initial values of Relative Humidity and Temperature to `20` and `333.15` K, respectively

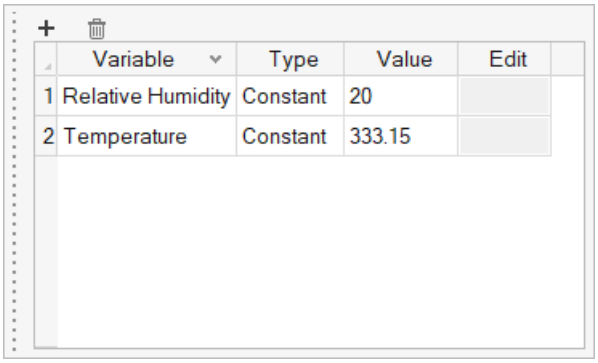


Figure 864:

5. Click ✓ on the guide bar.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 865:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.

4. Expand **Default initial conditions**.
5. Set the Temperature to `333.15` K and the remaining fields as shown in the figure below.
6. Click **Run** to launch AcuSolve.

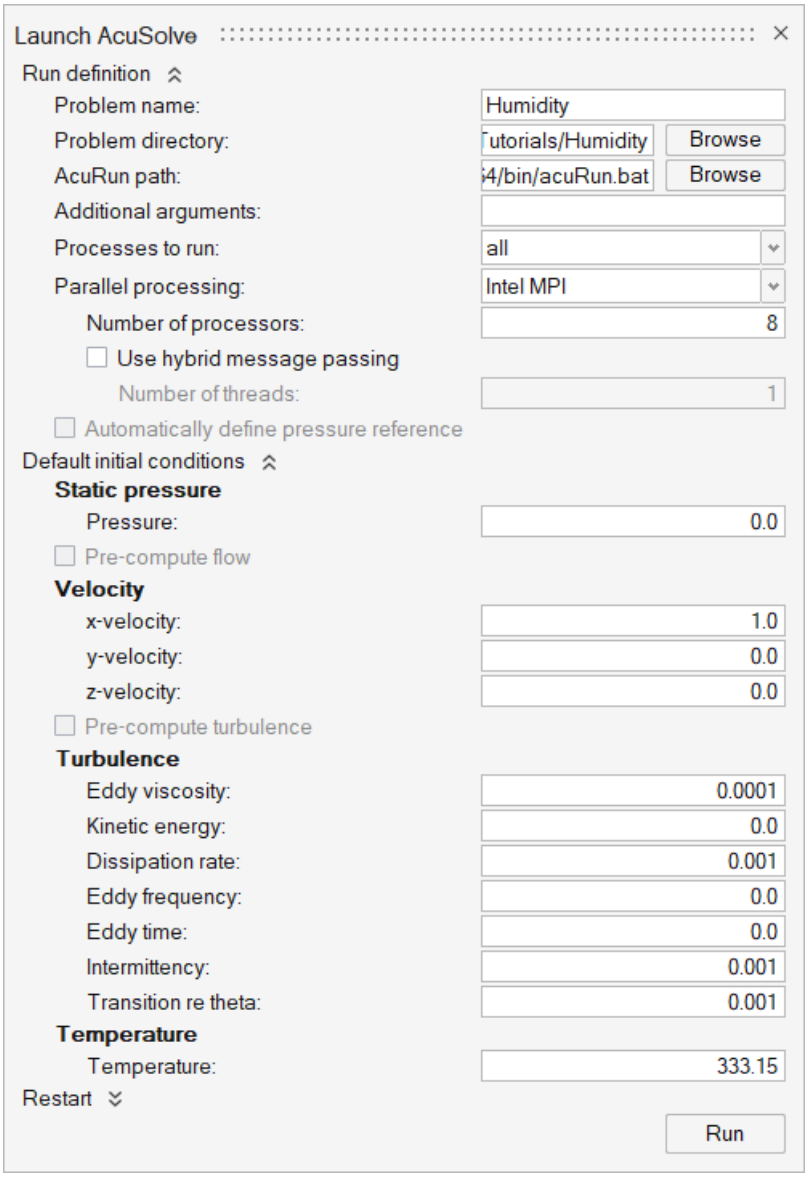


Figure 866:

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

In this step, you will create contour plots for temperature, relative humidity, mass fraction humidity, and velocity.

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. In the Post Browser, click on the icon beside **Flow Boundaries** to turn off the display of all the surfaces.

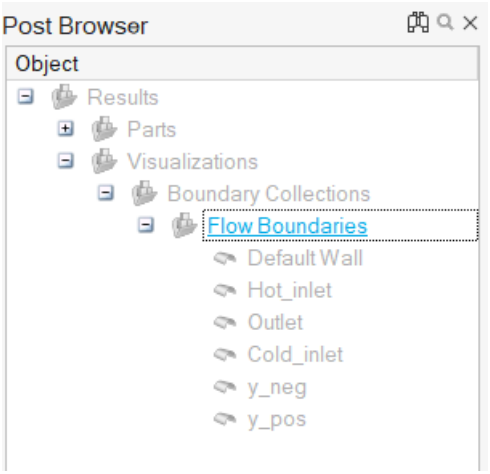


Figure 867:

5. Click the **Slice Planes** tool.

Figure 868:

6. Select the x-z plane in the modeling window.

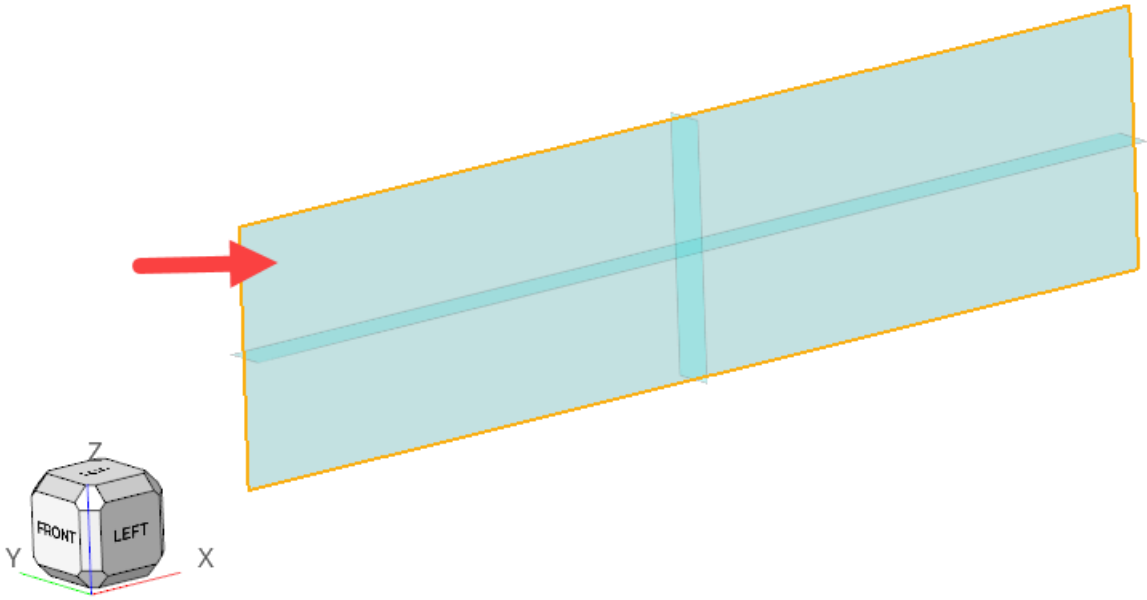


Figure 869:

7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, set the display to **temperature**.
9. Click then activate the **Legend** toggle.
10. Click and set the Colormap Name to **Rainbow Uniform**.

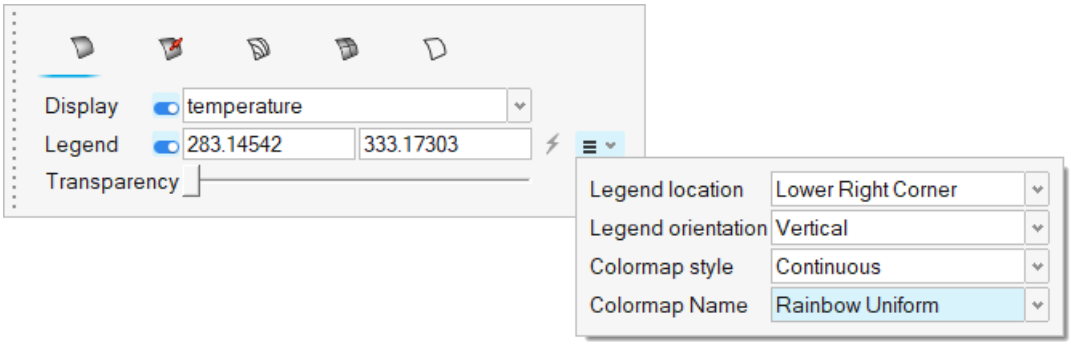


Figure 870:

11. On the guide bar, click to create the temperature contour plot.

Figure 871:

12. Hide the temperature contour and repeat the steps 5-11 to create a similar contour plot for relative humidity.

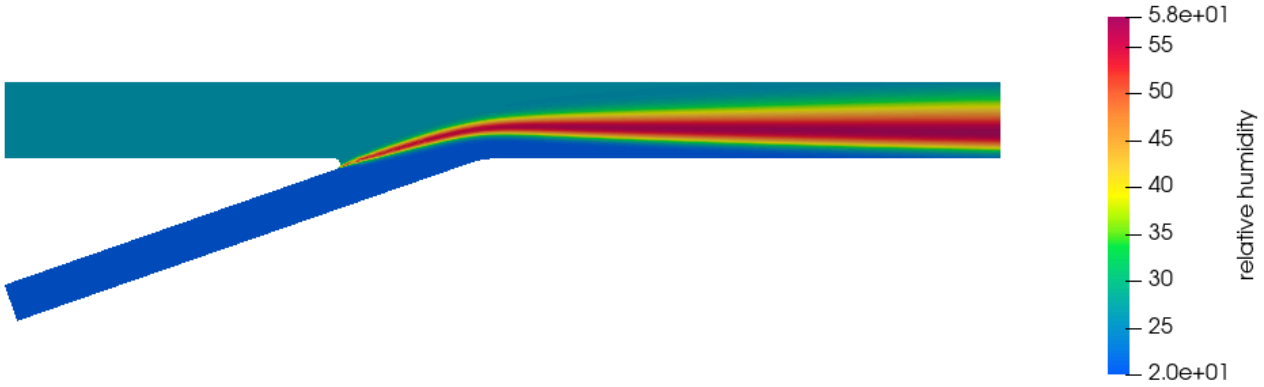


Figure 872:

13. Hide the relative humidity contour and repeat the steps 5-11 to create a similar contour plot for mass fraction humidity.

Figure 873:

14. Hide the mass fraction humidity contour and repeat the steps 5-11 to create a similar contour plot for velocity.

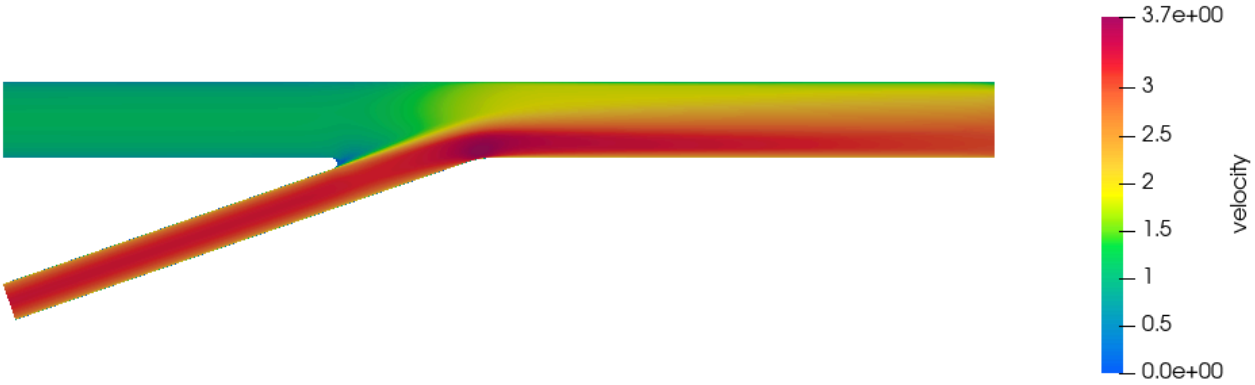


Figure 874:

Summary

In this tutorial, you learned how to set up and solve a humidity transport simulation using HyperMesh CFD and AcuSolve. You started by importing the HyperMesh CFD input database and then defined the flow setup. Once the solution was computed, you created a contour plot of temperature distribution, relative humidity, humidity mass fraction, and velocity using HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/grihwxz1en

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4201 Condensation & Evaporation - Air Box

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides instructions for running a transient simulation of an enclosed air-box using the humidity model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4201_Air_Box.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 875. As an example, an enclosed air-box problem is attached here to show the capability of the humidity modelling (condensation and evaporation) using AcuSolve. The main goal is to demonstrate and quantify accumulation and loss of water vapor on the bottom wall surface due to temperature change in an air domain at 70% relative humidity.

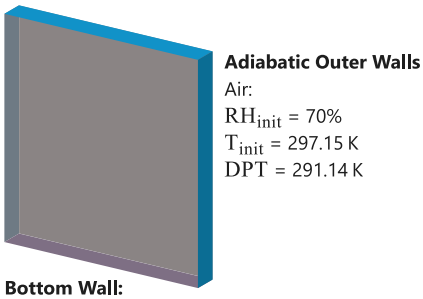


Figure 875:

In this particular domain, we assume that air is at a certain level of humidity. Condensation, evaporation, relative humidity, and temperature are all associated with dew point temperature. When the surface temperature drops below the dew point temperature, that is when condensation should start accumulating, and then accumulate up to a point where it reaches 100%. And again on the surface as the temperature rises and attains dew point again, it starts evaporating. This whole definition of both condensation and evaporation is explained in the below attached figure.

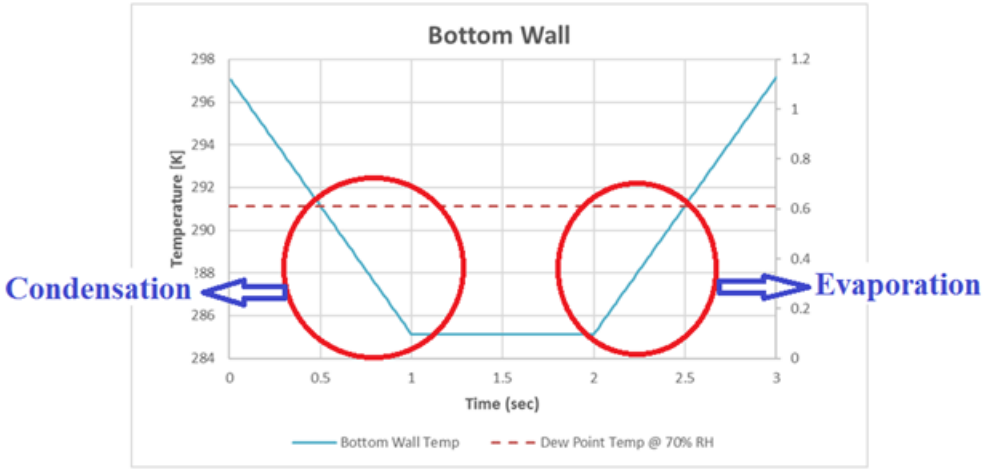


Figure 876:

From the above plot, we can see that the Air volume initial temperature is set to 297.15 K. The Bottom Wall temperature drops to 285.13 K over 1 sec, maintains 285.13 K for 1 sec, and then rises back to 297.15 K over 1 sec. The dew point temperature of the air at 70% RH is 291.14 K and is reached at 0.5 and 2.5 sec. On the whole we can see that both condensation and evaporation occurs when the dew point temperature is reached, as explained above.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 877:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4201_Air_Box.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Air_Box` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Air_Box` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 878:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

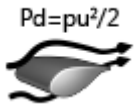
   

   Figure 879:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Select the **Multiphase flow** radio button.
   b) Set the Multifluid type to **Humidity transport**.
   c) Set the Time step size to `0.1` s and the Final time to `10` s.
   d) Set the Turbulence model to **Spalart-Allmaras**.
   e) Set the Gravity to `(0,-9.81,0)`.
   f) Set the Pressure scale to **Gauge** and click ≡. In the microdialog, set the Absolute pressure offset to `101325` Pa then press **Esc**.

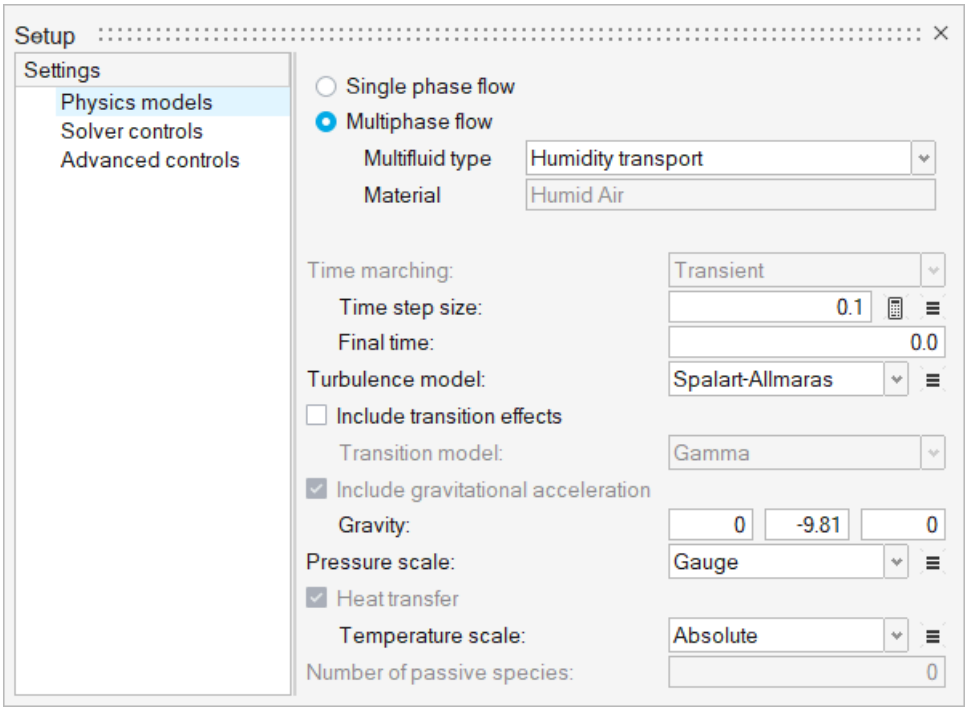


Figure 880:

3. Click the **Solver controls** setting.
4. Set the Minimum stagger iterations to `2` and the Maximum to `4`.

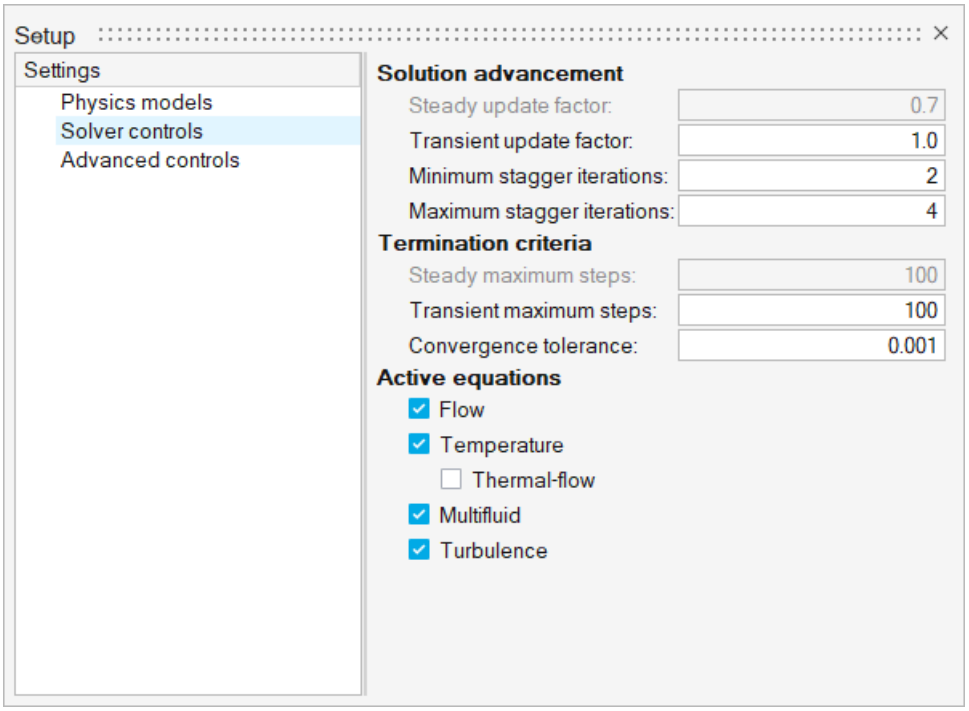


Figure 881:

5. Close the dialog and save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **No Slip** tool.

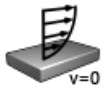


Figure 882:

2. Select the bottom surface (the surface with the minimum y-coordinate).

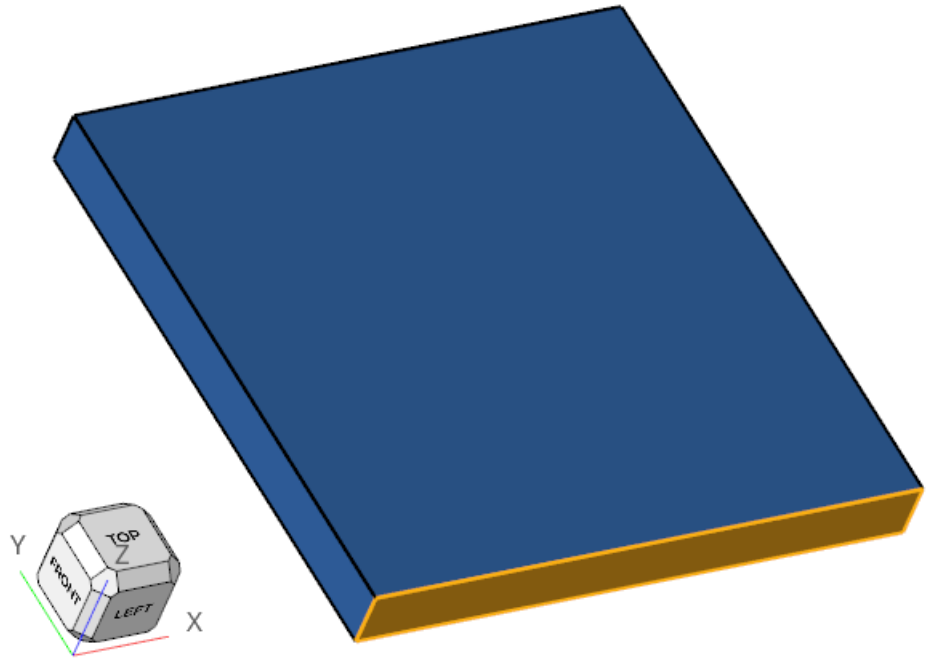


Figure 883:

3. In the microdialog, click on **Temperature** tab and set the Thermal boundary condition to **Temperature**.
4. Set the Temperature to `1`, expand the multiplier function drop-down, and select **Create new**.

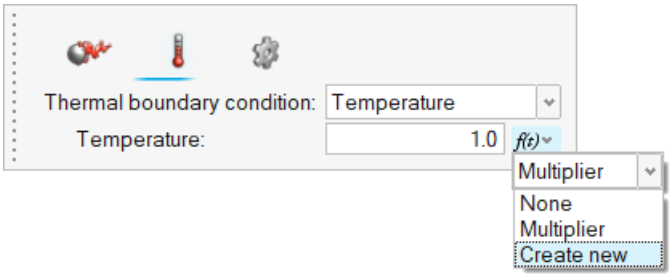


Figure 884:

5. In the multiplier dialog, set the Type to **Piecewise Linear**.
6. Set the Variable to **Time Step**.
7. Click twice to add two rows.
8. Enter the values as shown in the figure below then close the dialog.

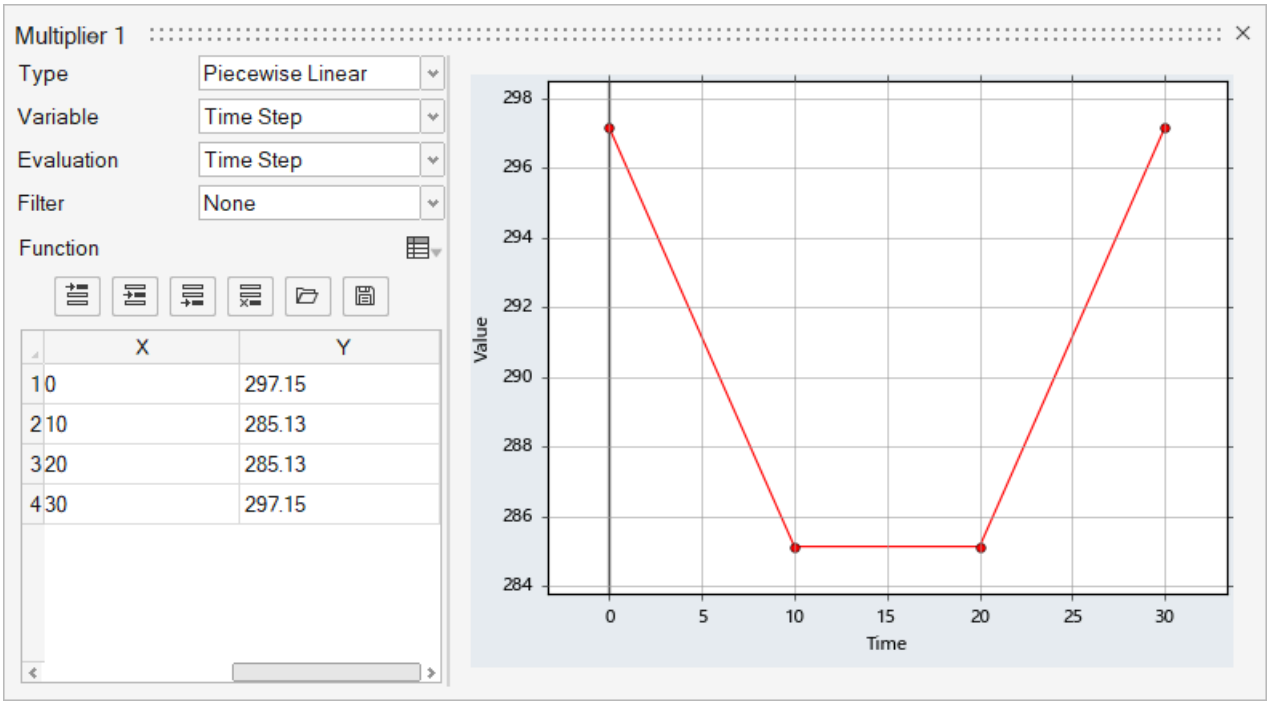


Figure 885:

9. In the Boundaries legend, double-click on **Wall** and rename it to `Bottom`.
10. On the guide bar, click ✓ to execute the command and exit the tool.
11. Click the **Slip** tool.

Figure 886:

12. Select the surface with the maximum z-coordinate.

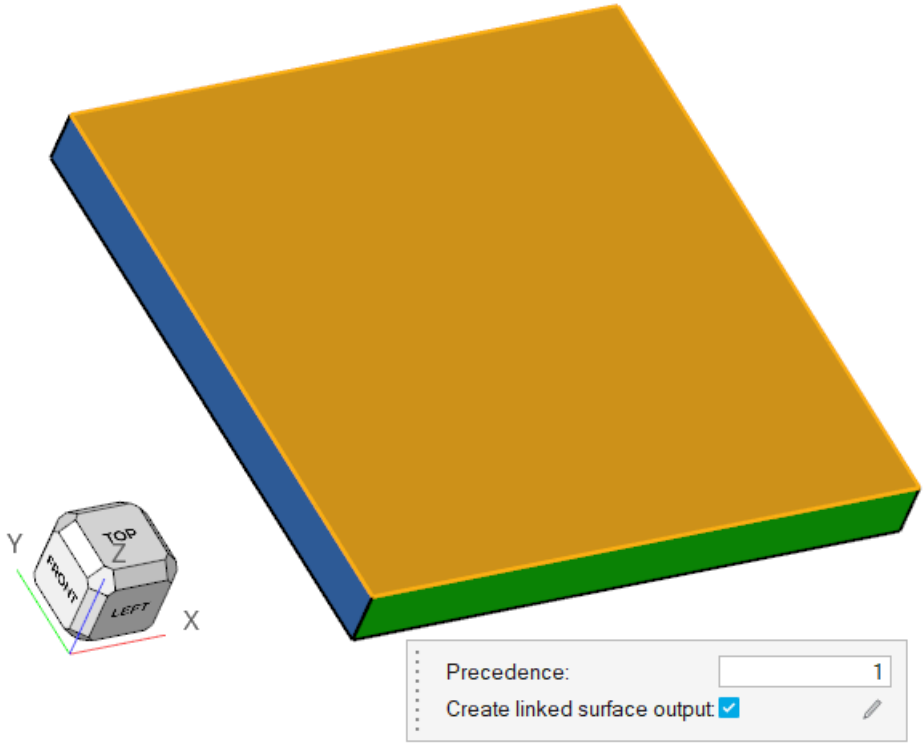


Figure 887:

13. In the Boundaries legend, double-click on **Slip** and rename it to `z_pos`.

14. On the guide bar, click ▸ to execute the command and remain in the tool.

15. Select the surface with the minimum z-coordinate.

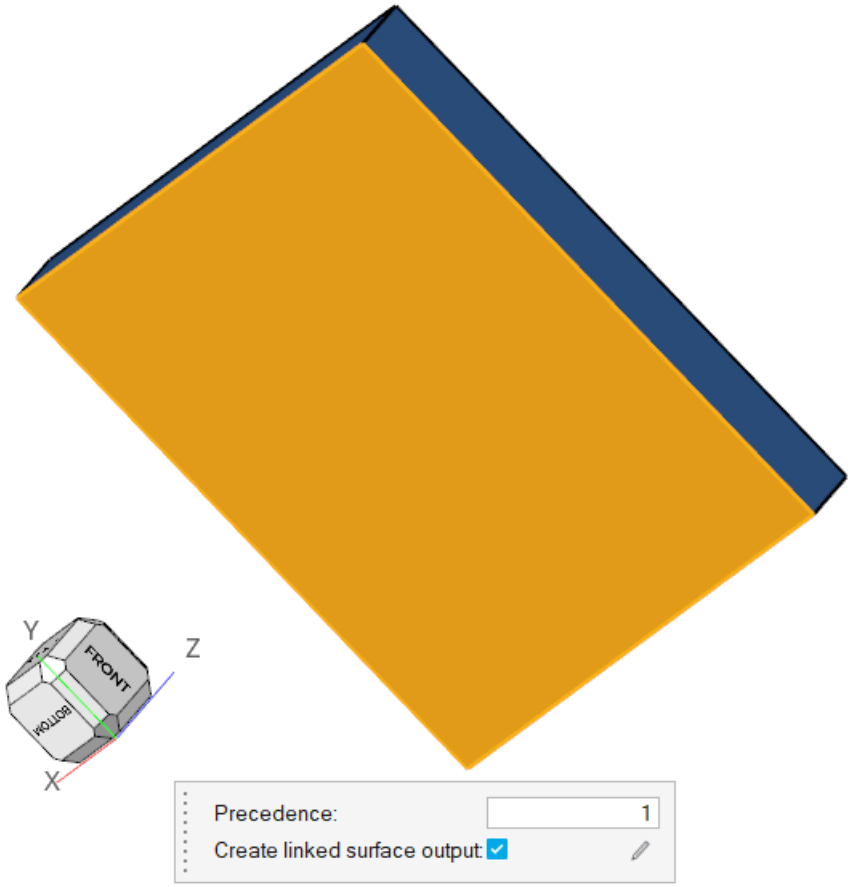


Figure 888:

16. In the Boundaries legend, double-click on **Slip** and rename it to `z_neg`.
17. Click ✓ on the guide bar.
18. Save the model.

Compute the Solution

The input HyperMesh database contains the mesh, hence you do not need to generate the mesh again.

Define the Nodal Initial Conditions

1. From the Solution ribbon, click the **Part** tool.

Figure 889:

2. Select the box solid.
3. In the dialog, click +, select **Relative Humidity** from the list of variables then click on the white space in the dialog.
4. Set the value to `70`.

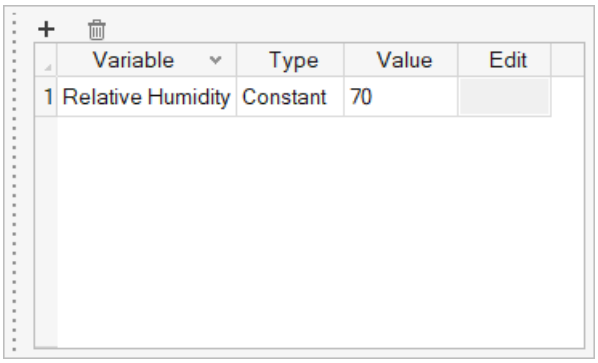


Figure 890:

5. Click ✓ on the guide bar.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 891:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Expand **Default initial conditions** and enter the values as shown below.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

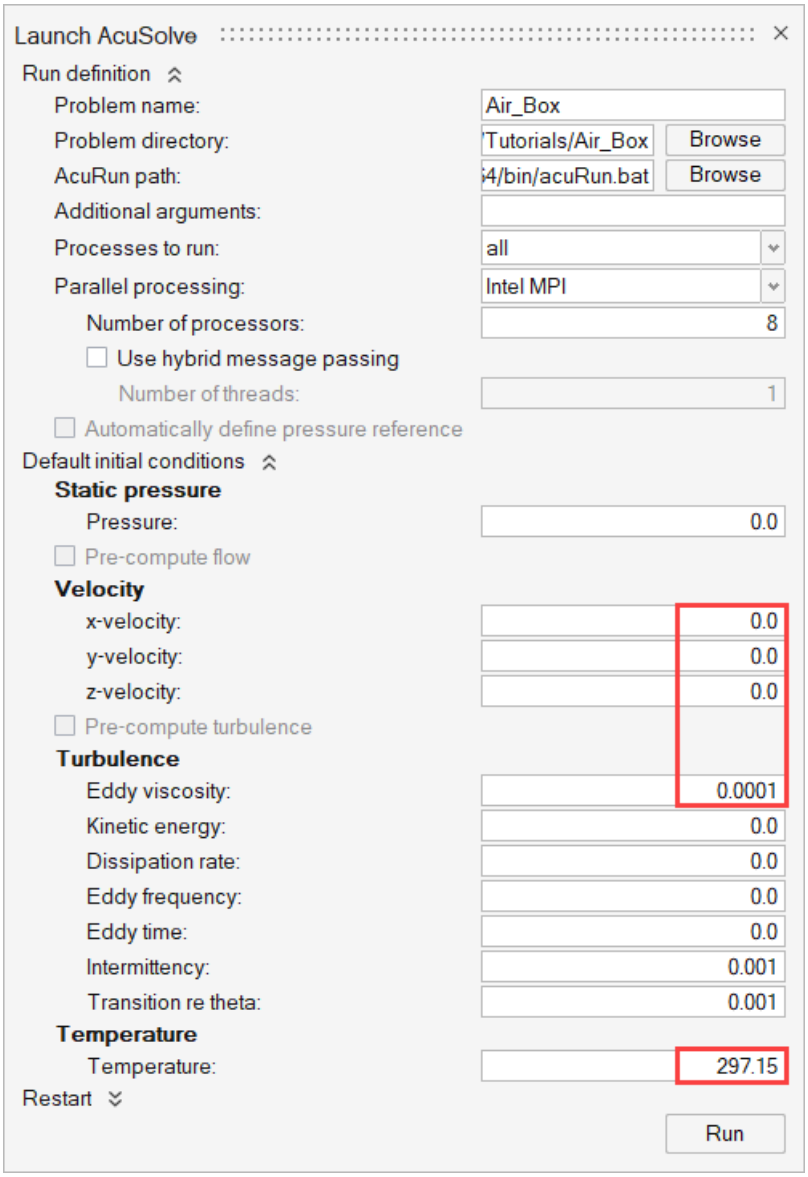


Figure 892:

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

6. Click the **Plot** tool.

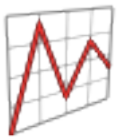


Figure 893:

7. In the **Plot Utility** dialog, double-click on **Residual Ratio** to plot the residuals.

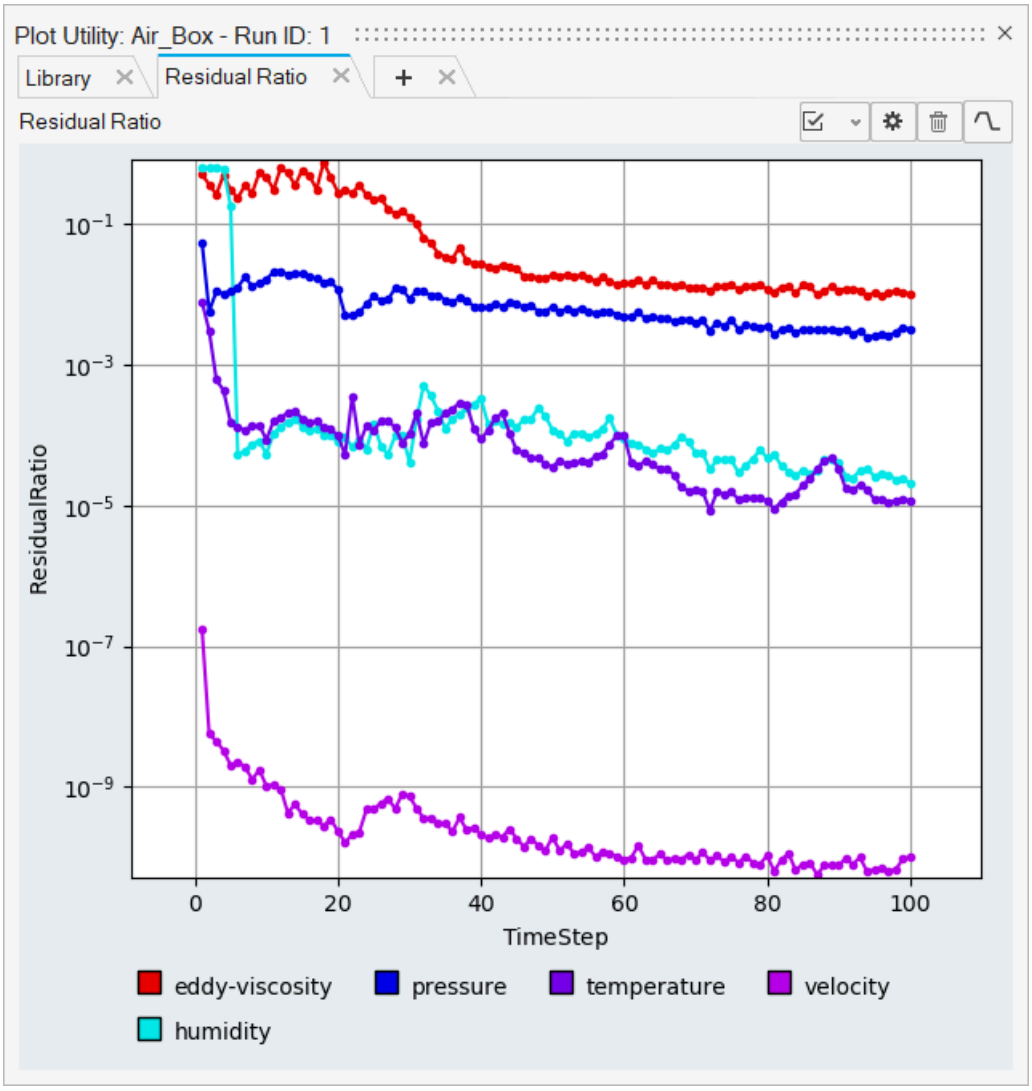


Figure 894:

8. Click + to add a new plot.
9. Set the X-Axis to **Time**.
10. To set the Y-Axis variable, click ☰ and then double click on **temperature** under Surface Output from the list of variables.

11. In the list of surface outputs, select **Bottom - Output**.

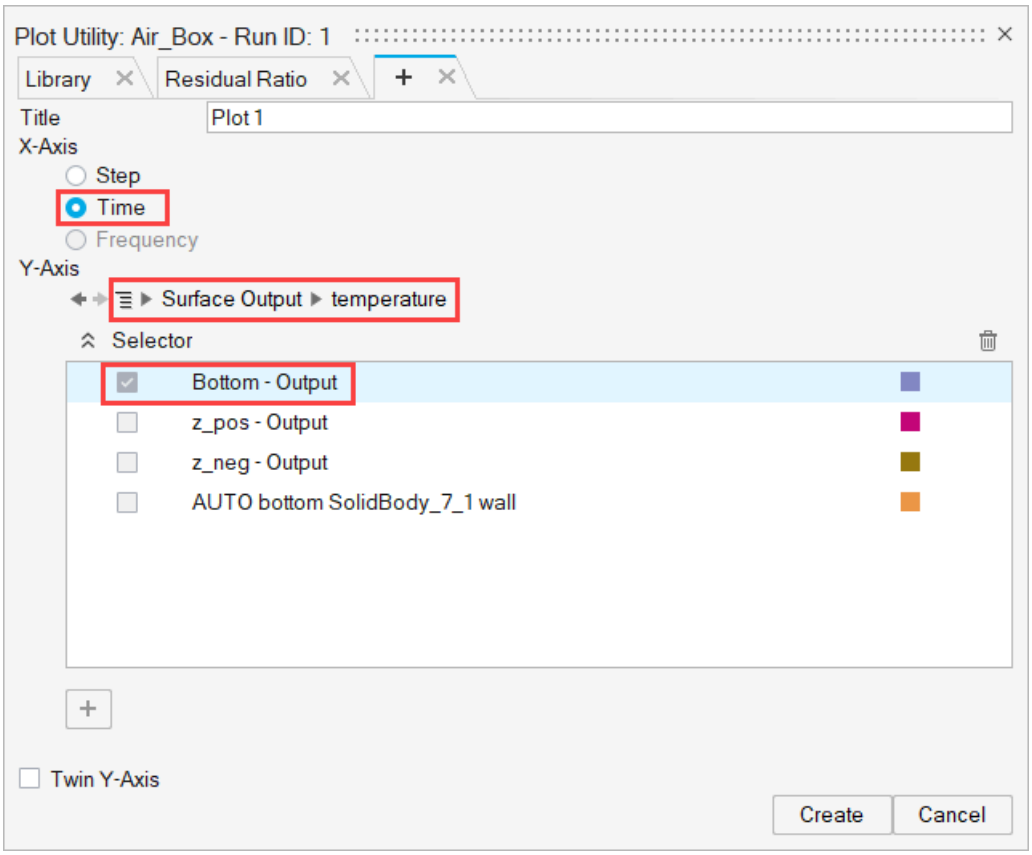


Figure 895:

12. Click **Create**.

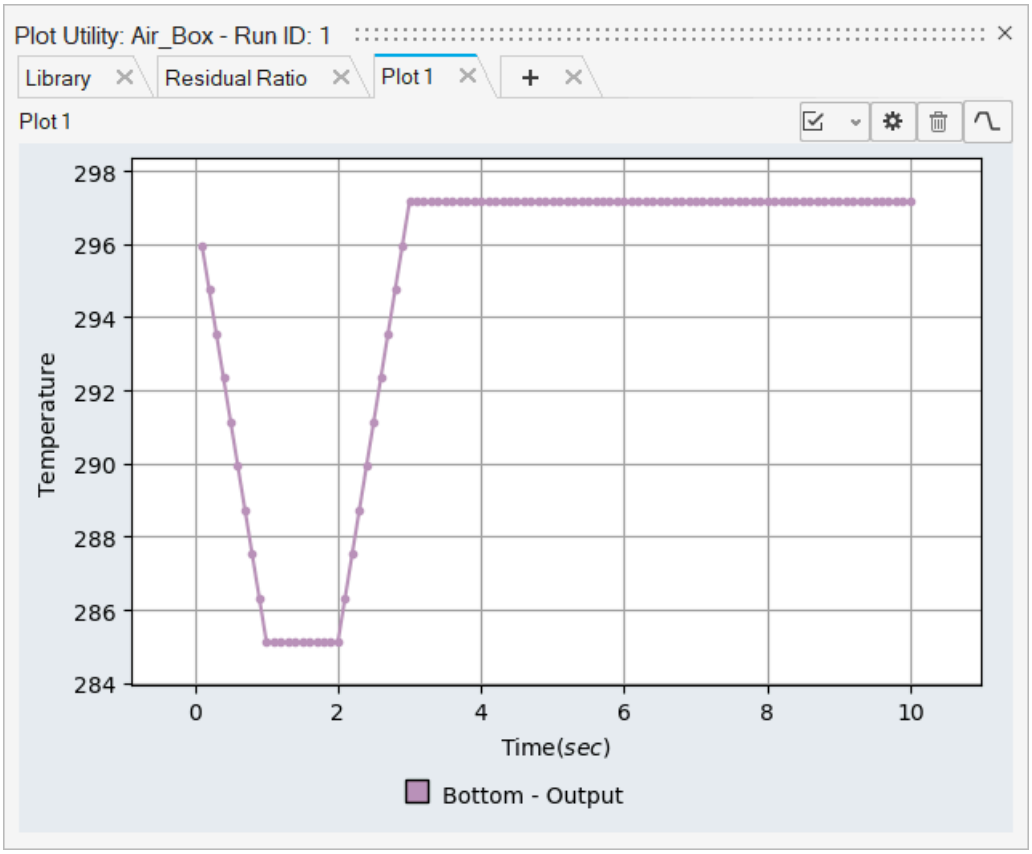


Figure 896:

13. Similarly, view the plots for Relative Humidity and Dewpoint Temperature.

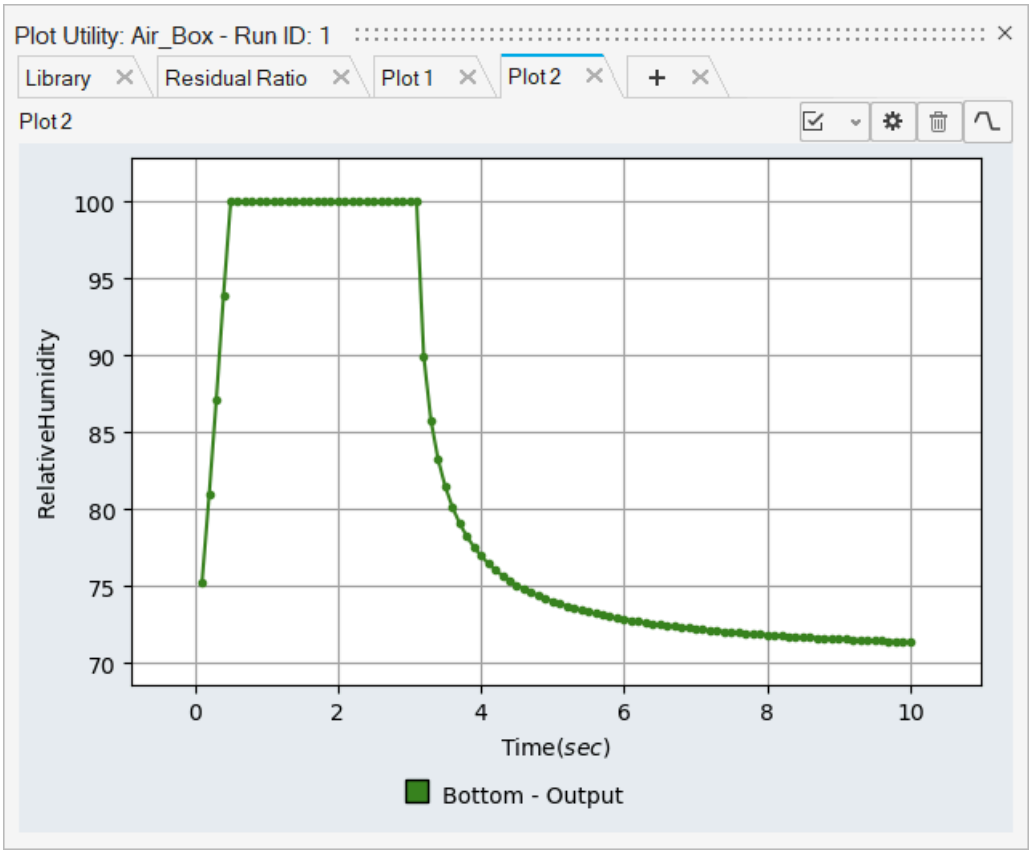

Plot Utility: Air_Box - Run ID: 1
Library
Residual Ratio
Plot 1
Plot 2
Plot 2
RelativeHumidity
100
95
90
85
80
75
70
0
2
4
6
8
10
Time(sec)
Bottom - Output

Figure 897:

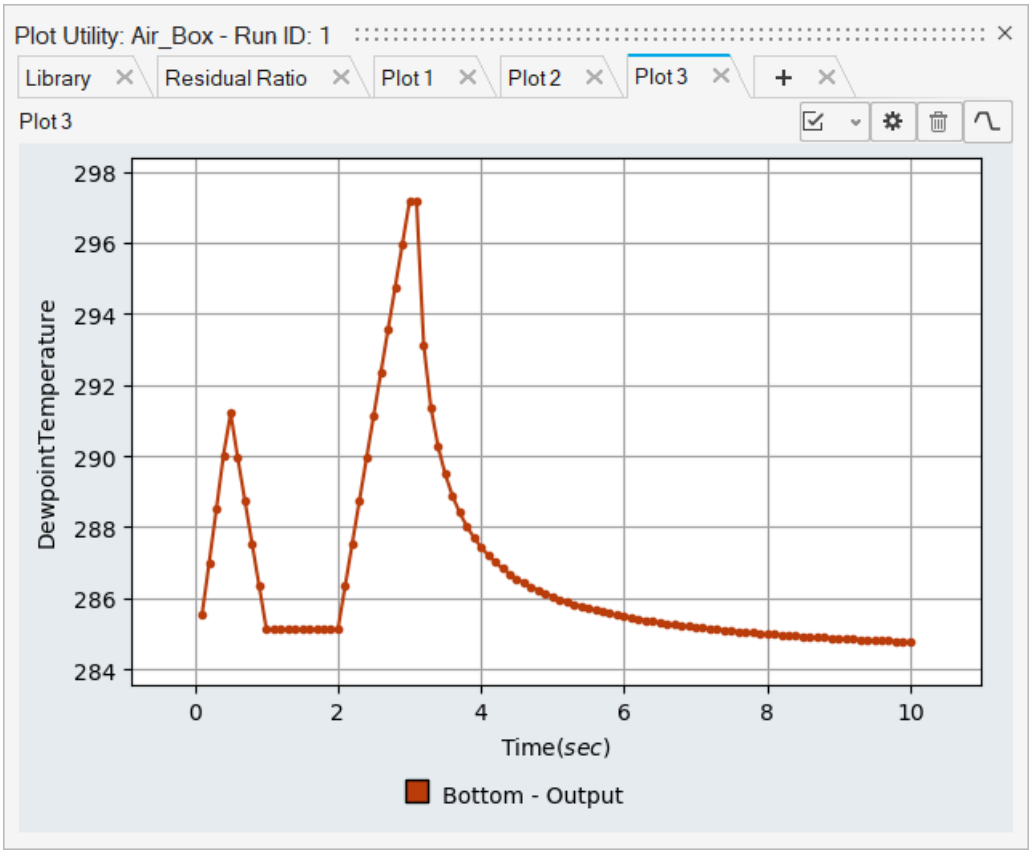


Figure 898:

Post-Process the Results with HM-CFD Post

In this step, you will create contour plots for temperature, relative humidity, and dew point temperature.

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. In the Post Browser, click on the icon beside **Flow Boundaries** to turn off the display of all the surfaces.

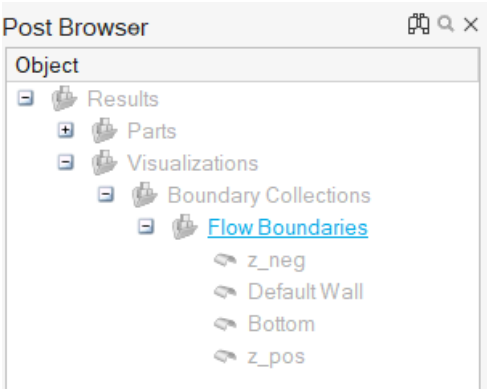


Figure 899:

5. Drag the slider on the Animation toolbar to the 10th frame.

Figure 900:

6. Click the **Slice Planes** tool.

Figure 901:

7. Select the x-y plane in the modeling window.

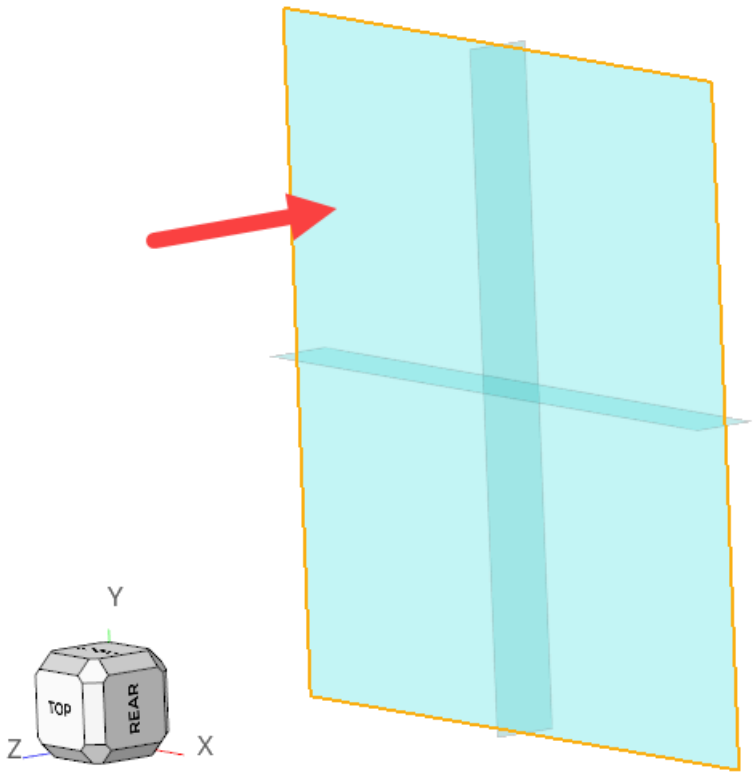


Figure 902:

8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, set the display to **temperature**.
10. Click then activate the **Legend** toggle.
11. Click and set the Colormap Name to **Rainbow Uniform**.

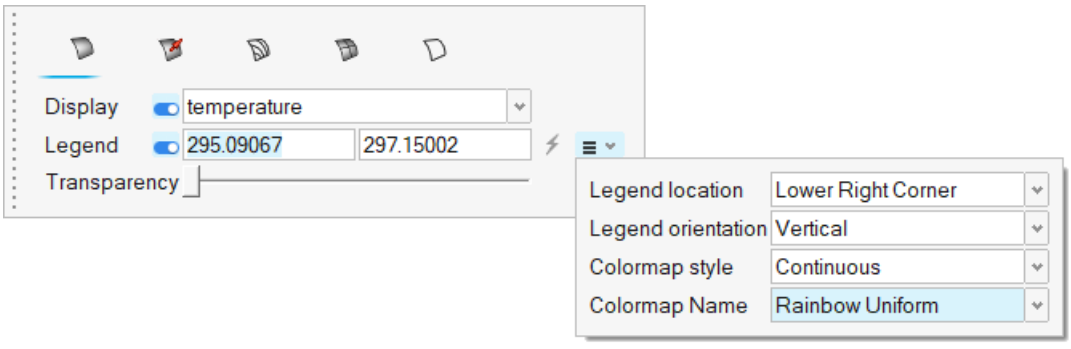


Figure 903:

12. On the guide bar, click to create the temperature contour plot.

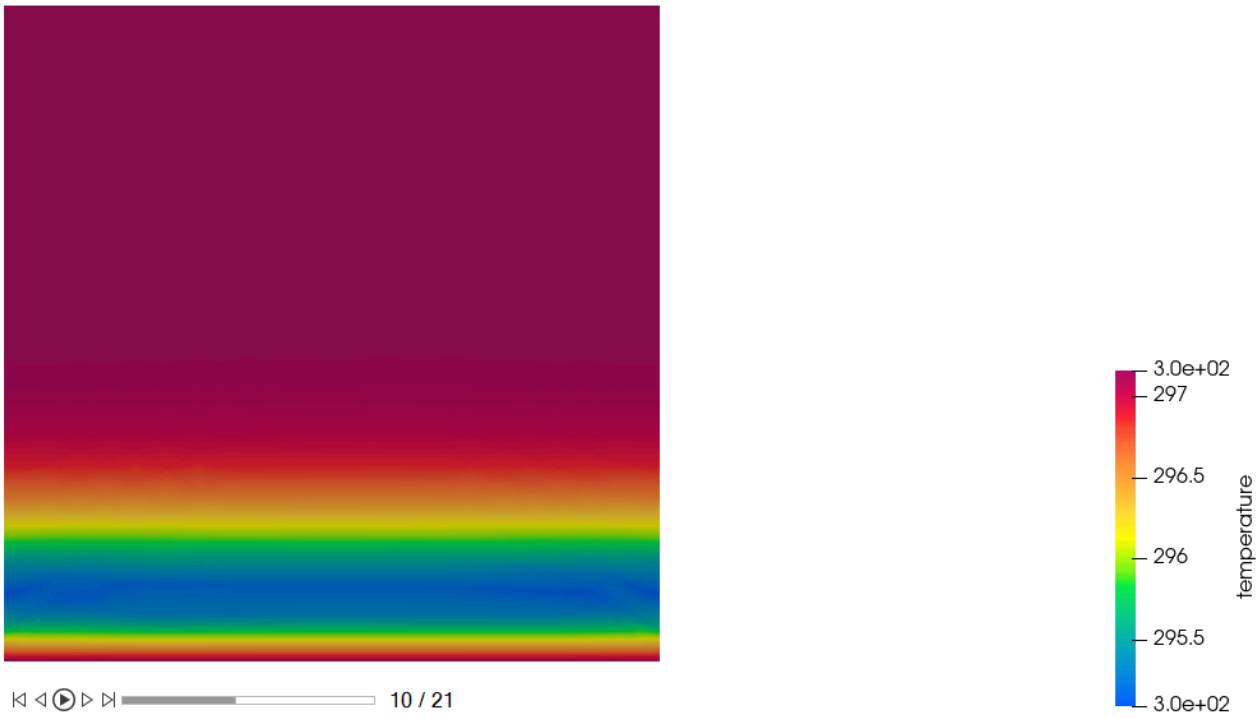


Figure 904:

13. Hide the temperature contour, then repeat the steps 6-12 to create a similar contour plot for relative humidity.

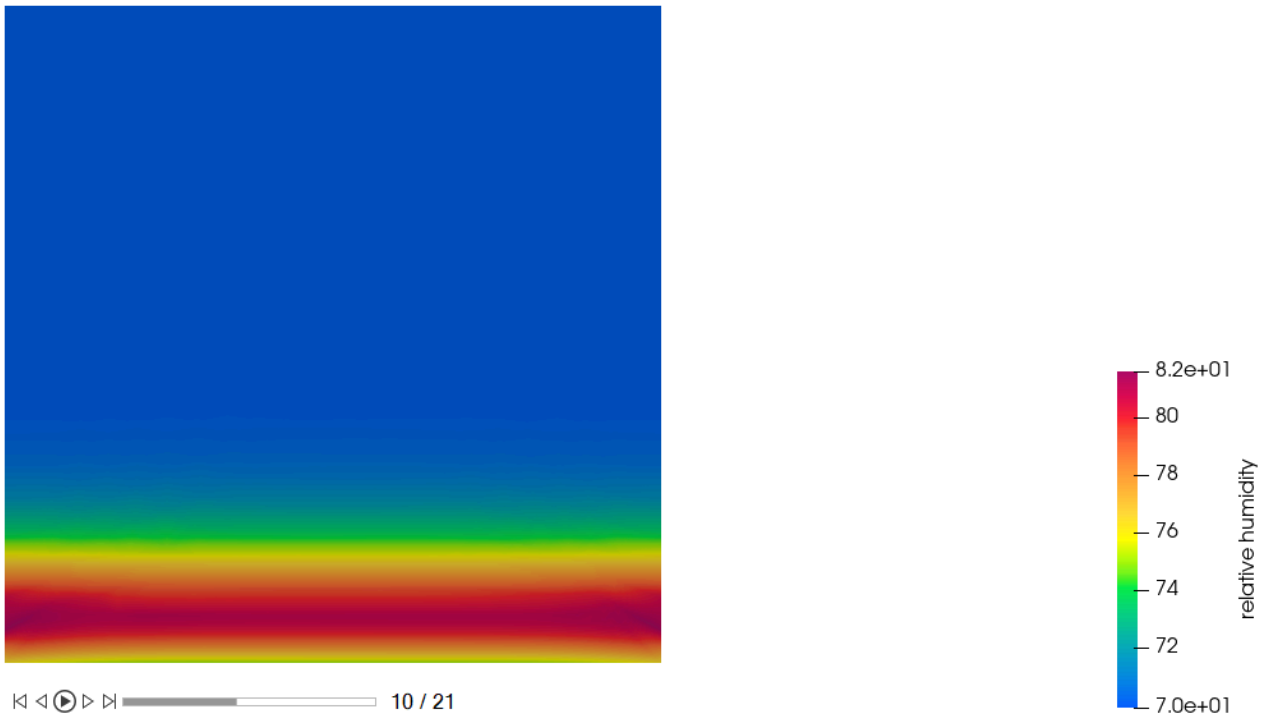


Figure 905:

14. Hide the relative humidity contour, then repeat the steps 6-12 to create a similar contour plot for dew point temperature.

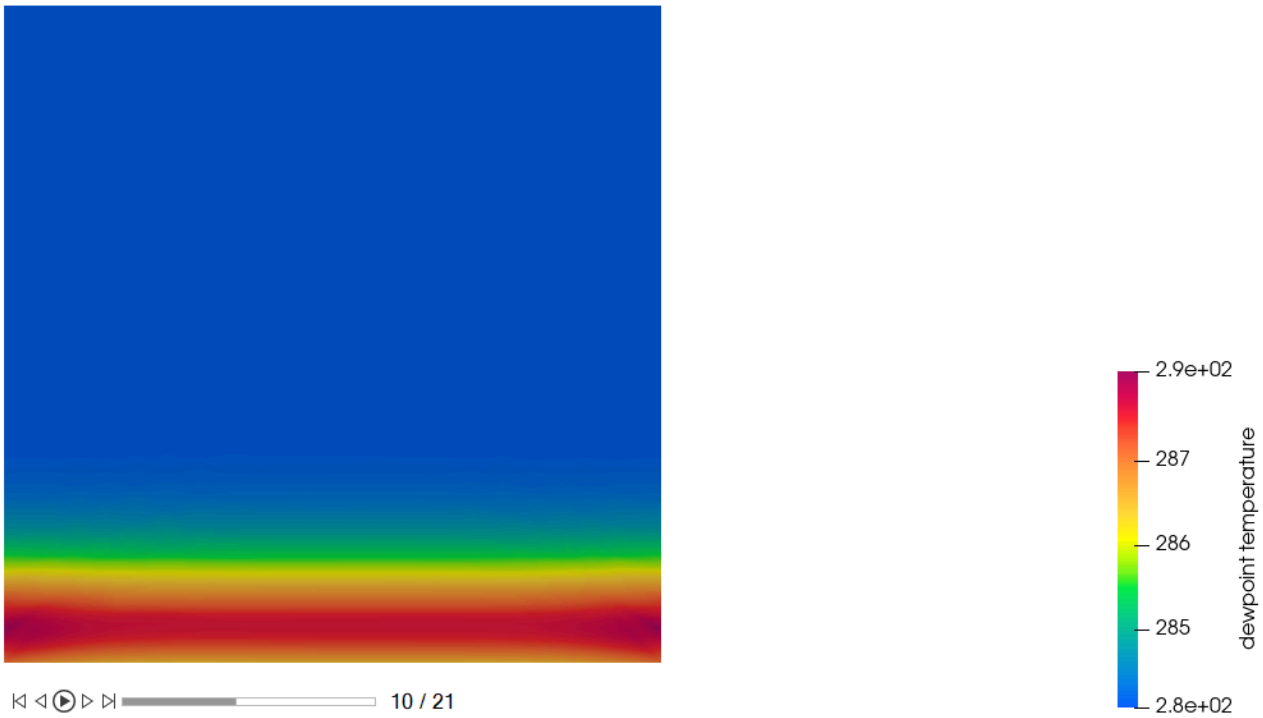


Figure 906:

Summary

In this tutorial, you learned how to set up and solve a multiphase humid air condensation and evaporation simulation using HyperMesh CFD and AcuSolve. You started by importing the HyperMesh CFD input database and then defined the flow setup. Once the solution was computed, you created a plot of residual ratios using the plot utility in HyperMesh CFD. Finally, you created a contour plot of temperature distribution, relative humidity, and dew point temperature using HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/iez1comcr9

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4300 Species Transport Modeling

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

This tutorial provides instructions for modeling species transport using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T4300_HoneyTeaSpecies.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 907. It consists of two sections, Cup_Honey and Cup_Main. Honey enters from Cup_Honey to Cup_Main and mixes with water.

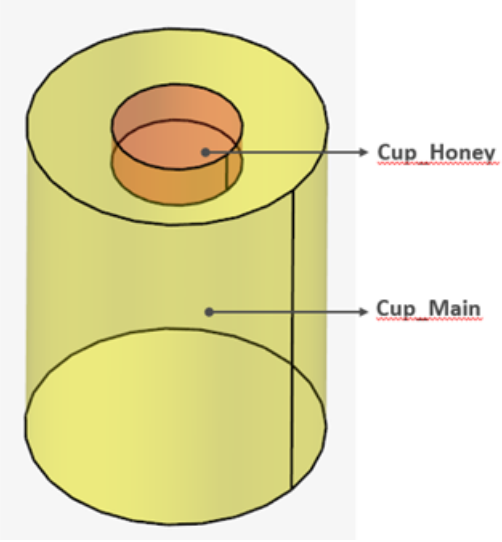


Figure 907:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 908:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4300_HoneyTeaSpecies.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `HoneyTeaPlug` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `HoneyTeaPlug` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 909:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 910:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

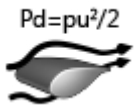


Figure 911:

The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.125` sec.
   c) Select **Laminar** as the Turbulence model.
   d) Check the **Include Gravitational Acceleration** option and set the gravity to `9.81` m/s^2 in the z direction.
   e) Set the Number of passive species to `1`.

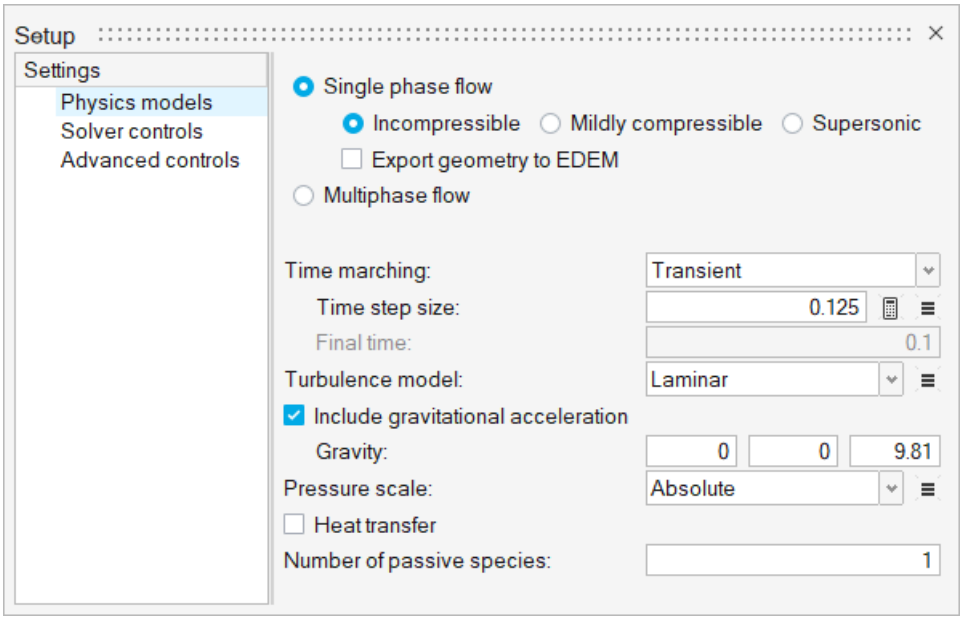


Figure 912:

3. Click the **Solver controls** setting then set the Transient maximum steps to `80`.

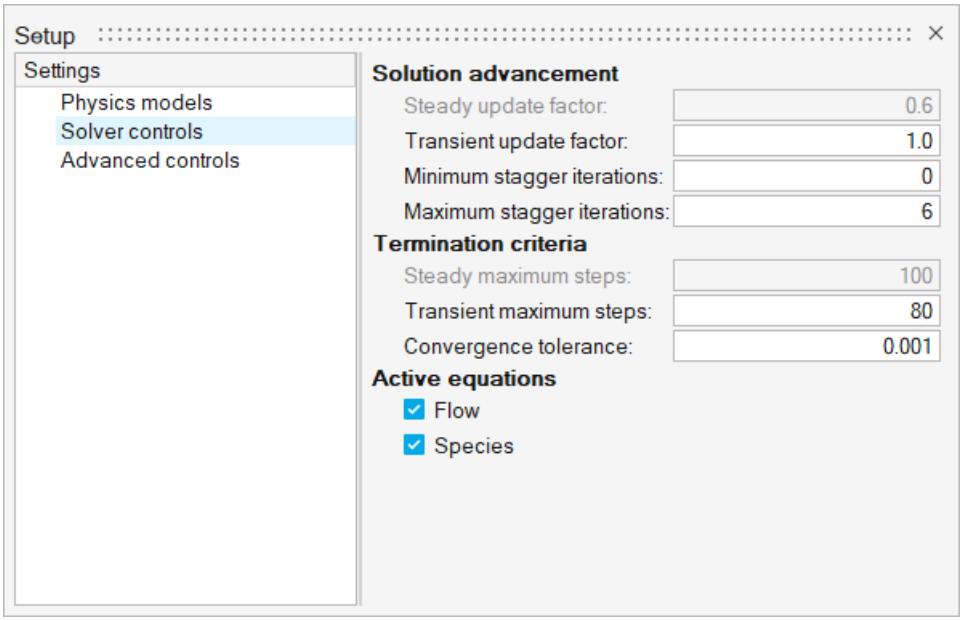


Figure 913:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material Library** tool.

Figure 914:

The **Material Library** dialog opens.

2. Under Settings, click **Fluid**, then click the **My Materials** tab.
3. Click + to create a new material.
4. For density:
   a) Change the type to **Piecewise Linear**.
   b) Define the Curve fit variable as **Species 1**.
   c) Click twice under Function then enter `1000` kg/m^3 and `1500` kg/m^3 in the Y column for the density of water and honey, respectively.

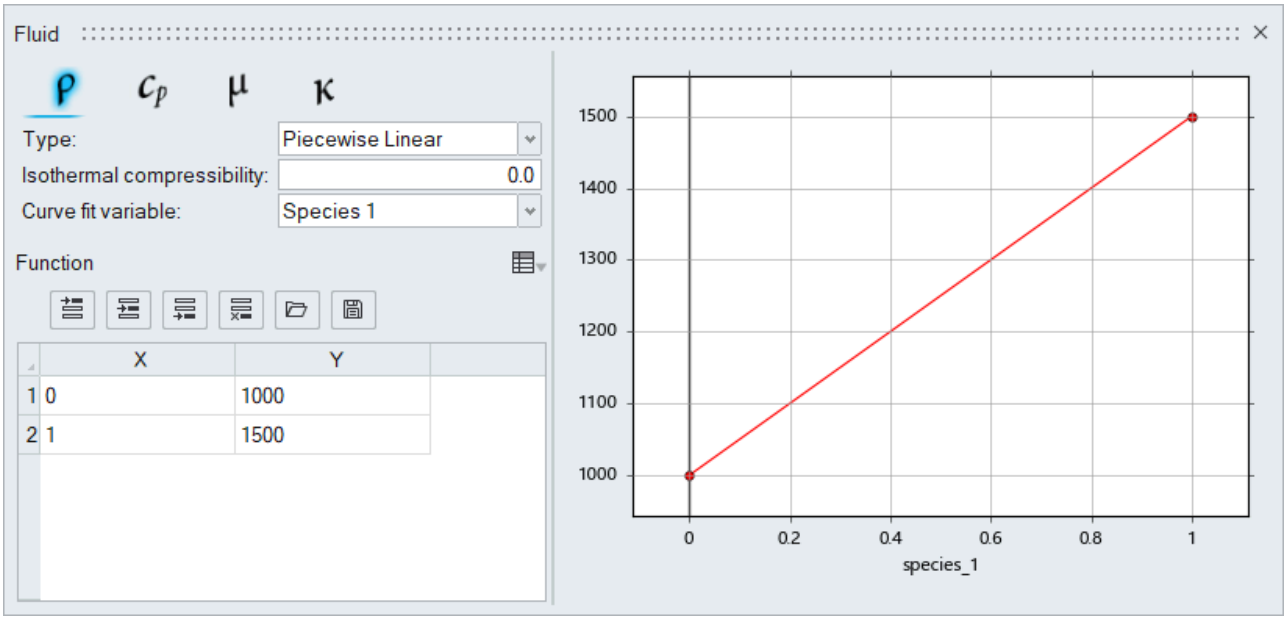


Figure 915:

5. Similarly, define the viscosity (kg/m-sec) values for water and honey according to the figure below.

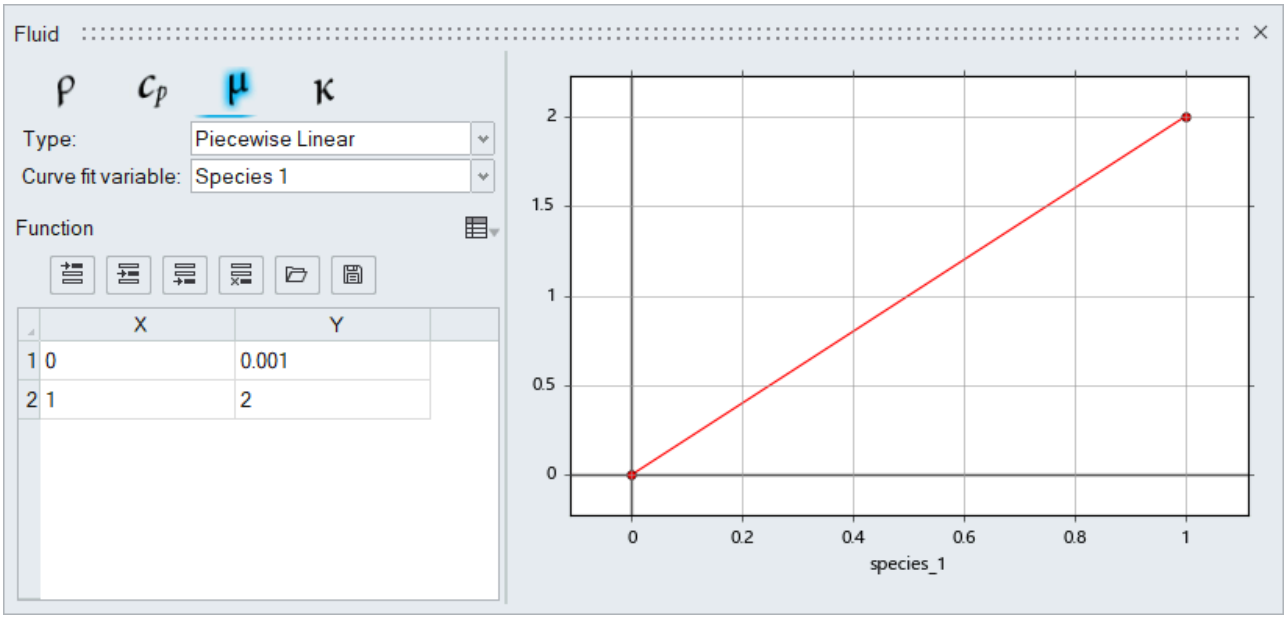


Figure 916:

6. Close the material editing dialog, then rename the Fluid material to `Honey`.

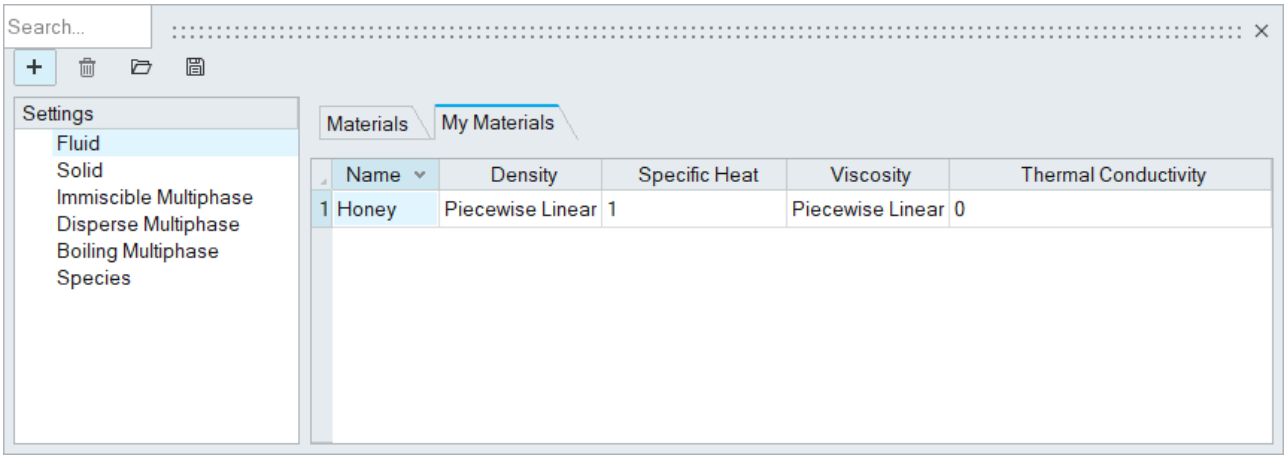


Figure 917:

7. From the Flow ribbon, click the **Material** tool.

Figure 918:

8. Select both model volumes.

9. Select **Honey** from the Material drop-down menu.

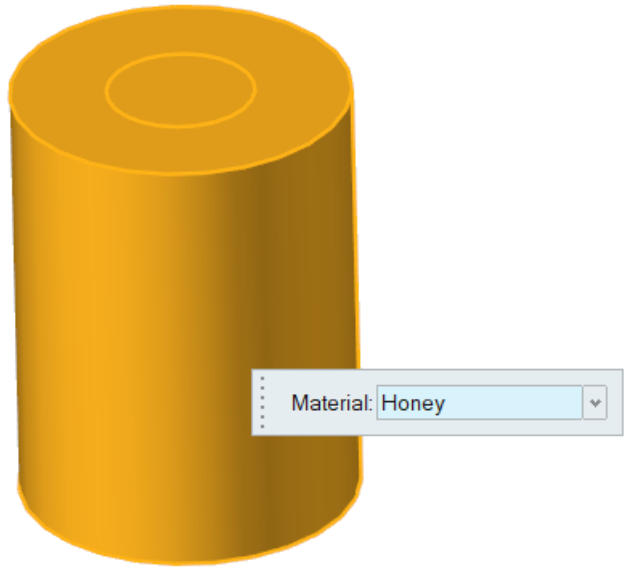


Figure 919:

10. On the guide bar, click ✓ to execute the command and exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, click the **Slip** tool.

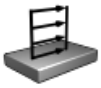


Figure 920:

2. Select the top two surfaces highlighted in the figure below then click ✓ on the guide bar.

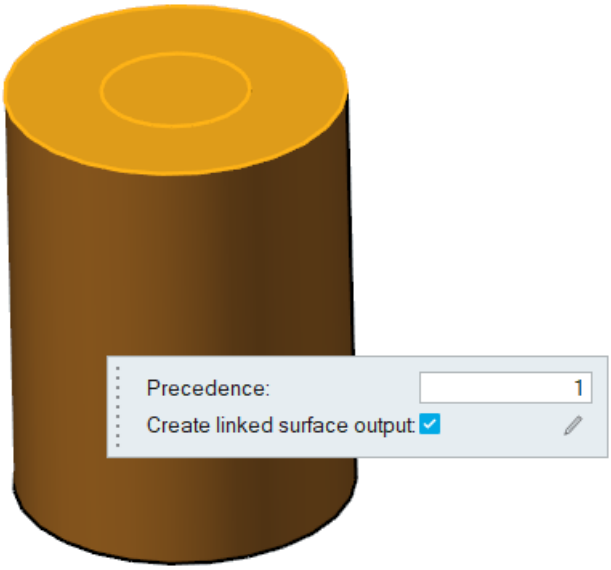


Figure 921:

Define Nodal Outputs and Nodal Initial Conditions

In this step, you will define the nodal output frequency and then specify the nodal initial conditions for honey

1. Define nodal output frequency.
   a) From the Solution ribbon, click the **Field** tool.

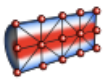


Figure 922:

   The **Field Output** dialog opens.
   b) Click the **Solution variables** setting.
   c) Check the **Write initial conditions** option.
   d) Verify that the **Write results at time step interval** option is checked.
   e) Set the Time step interval to 2.

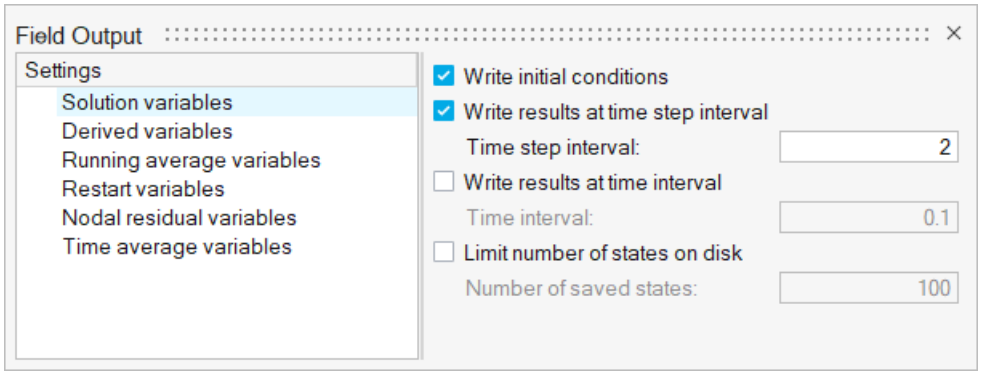


Figure 923:

2. Define the nodal initial conditions.

a) From the Solution ribbon, click the **Part** tool.

Figure 924:

b) Select the volume highlighted below.

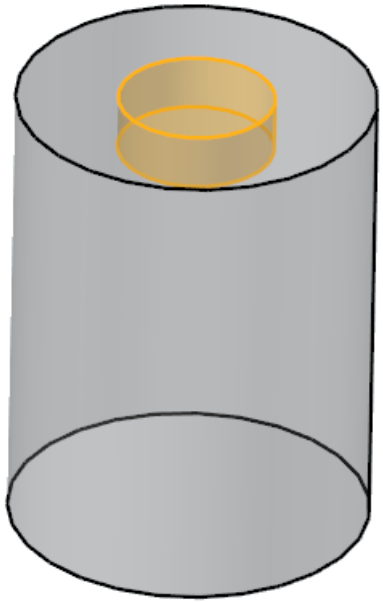


Figure 925:

c) In the microdialog, click + and select **Species 1** from the list.

d) Set the Type to **Constant** and the Value to `1`.

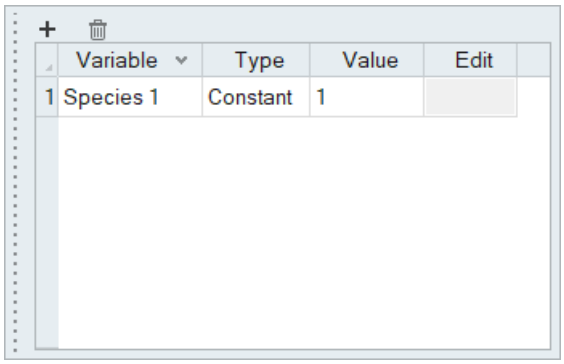


Figure 926:

3. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 927:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Uncheck the **Automatically define pressure reference** option.
5. Expand **Default initial conditions** and uncheck the **Pre-compute flow** and **Pre-compute turbulence** options.
6. Set all the velocity components to `0` m/s.
7. Leave the remaining options as default and click **Run** to launch AcuSolve.

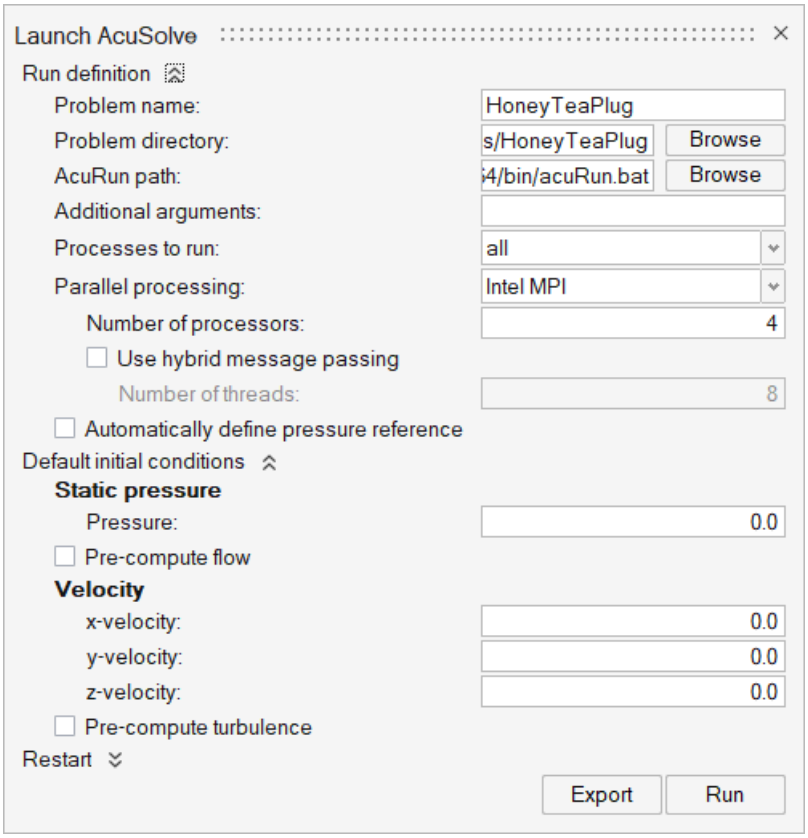


Figure 928:

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

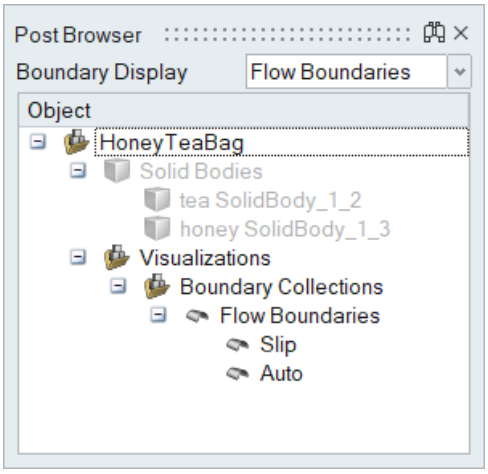


Figure 929:

4. From the Post ribbon, click the **Slice Planes** tool.

Figure 930:

5. Select the plane shown in the figure below.

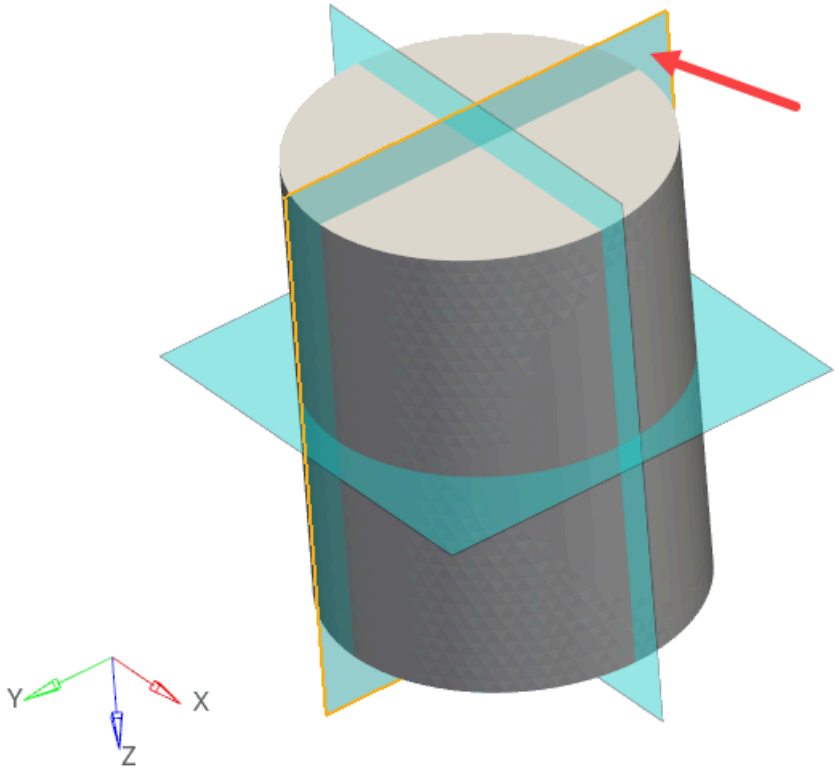


Figure 931:

6. In the Post Browser, hide the **Slip** and **Auto** surfaces.
7. In the slice plane microdialog, click to create the slice plane.
8. In the display properties microdialog, change the Display option to **species 1**.
9. Activate the **Legend** radio button then click and set the Colormap name to **Rainbow Uniform**.

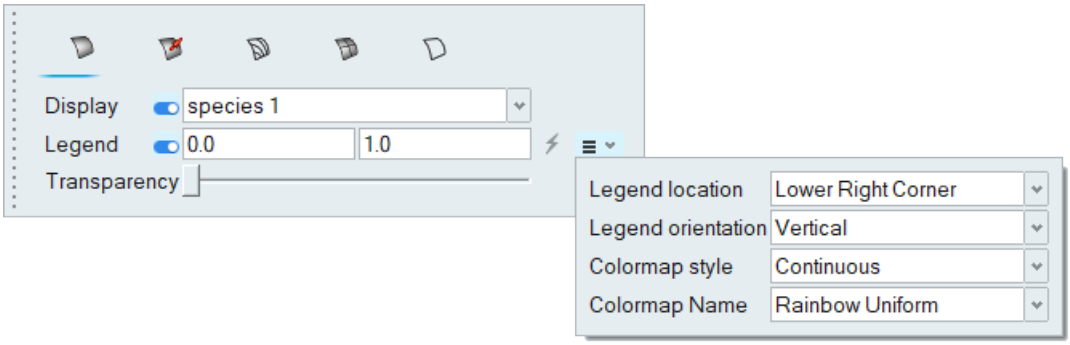


Figure 932:

10. Click on the guide bar.
11. Make sure that the time scale at bottom of the modeling window is at 1 sec.

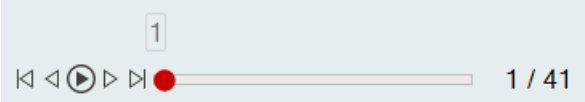


Figure 933:

12. Adjust the time scale to show 2 sec, 4, sec, 6 sec, 8 sec, and 10 sec.
The distribution of species 1 (honey) at those times is shown below.

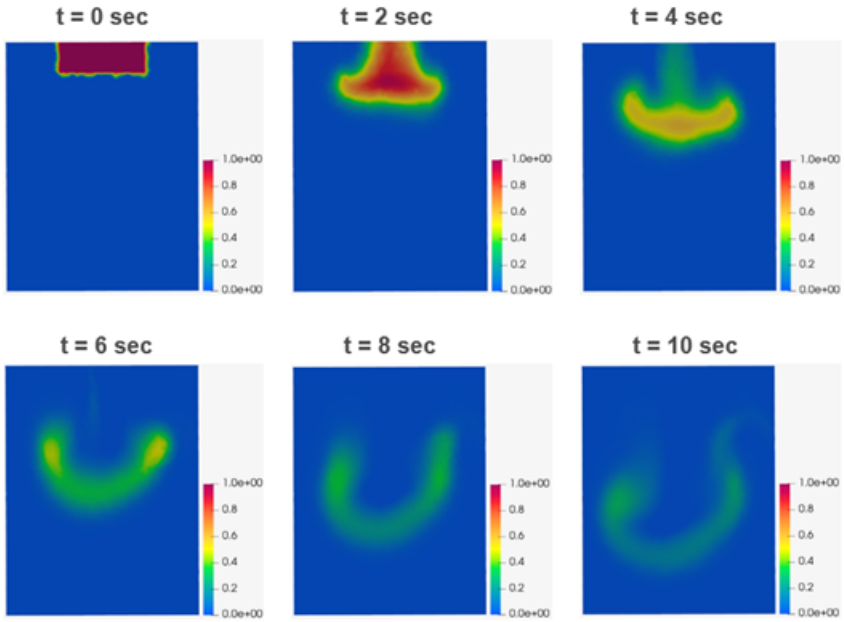


Figure 934:

13. In the Post Browser, hide the slice plane and show the **Auto** and **Slip** surfaces.

14. Click the **Boundary Groups** tool.

Figure 935:

15. Select all the surfaces on the model to create a new boundary group.

16. In the microdialog, move the Transparency slider to the middle to add a transparency effect.

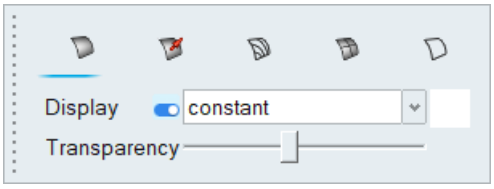


Figure 936:

17. Click ✓ on the guide bar.
18. Click the **Iso-Surfaces** tool.

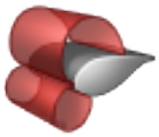


Figure 937:

19. In the Iso-function microdialog, change the Iso Variable to **species 1** and set the Iso Value to `0.5`

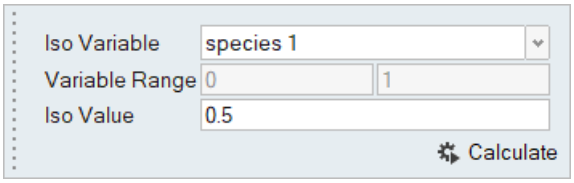


Figure 938:

20. Click **Calculate** then click ✓.

Figure 939: Distribution of species at 3 sec

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving species transport using HyperMesh CFD. You started by opening the HyperMesh input file with the geometry and then defined the simulation parameters, fluid material, and boundary conditions. Once the solution was computed, you visualized the results of species transport with help of planes and an iso-surface.

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 2 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from creating CAD, generating mesh, and setting up the simulation to visualizing the results. No files are needed to start from here.
2. Solution Setup - Mesh to Results

You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/4mc3ieunzc

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 4400 Mixing Flows Using the Gas Kinetics Model

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/kbb813d1b7

Additional Resources

For more tutorials, visit the SimLab Learning Center.

Moving Body

5

This chapter covers the following:

- ACU-T: 5000 Blower - Steady (Rotating Frame) (p. 542)
- ACU-T: 5001 Blower - Transient (Sliding Mesh) (p. 560)
- ACU-T:5002 Centrifugal Blower – Hybrid (MRF + MM) (p. 575)
- ACU-T: 5100 Modeling of a Fan Component Using the Fan Component - Coefficient Method (p. 576)
- ACU-T: 5101 Modeling of a Fan Component Using the Fan Component - PQ Method (p. 599)
- ACU-T: 5200 Rigid-Body Dynamics of a Check Valve (p. 619)
- ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve (p. 651)
- ACU-T: 5202 Flow Closing Valve (p. 684)
- ACU-T: 5400 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (P-FSI) (p. 685)
- ACU-T: 5403 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (p. 686)

ACU-T: 5000 Blower - Steady (Rotating Frame)

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5000_BlowerSteady.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 940 and Figure 941. It consists of a centrifugal blower with a wheel of forward curved blades, and a housing with inlet and outlet ducts. The fluid through the inlet plane enters the hub of the blade wheel, radially accelerates due to centrifugal force as it flows over the blades, and then exits the blower housing through the outlet plane. Because they're relatively cheaper and simpler than axial fans, centrifugal blowers have been widely used in HVAC (heating, ventilating, and air conditioning) systems of buildings.

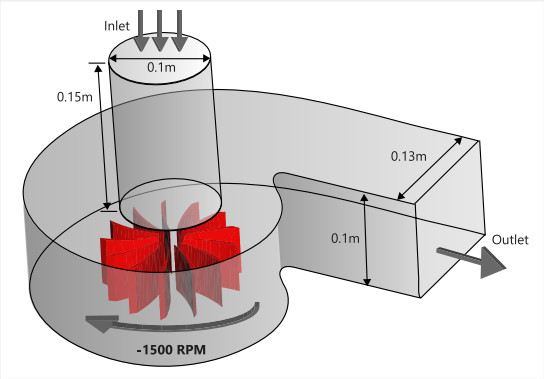


Figure 940: Schematic of Centrifugal Blower

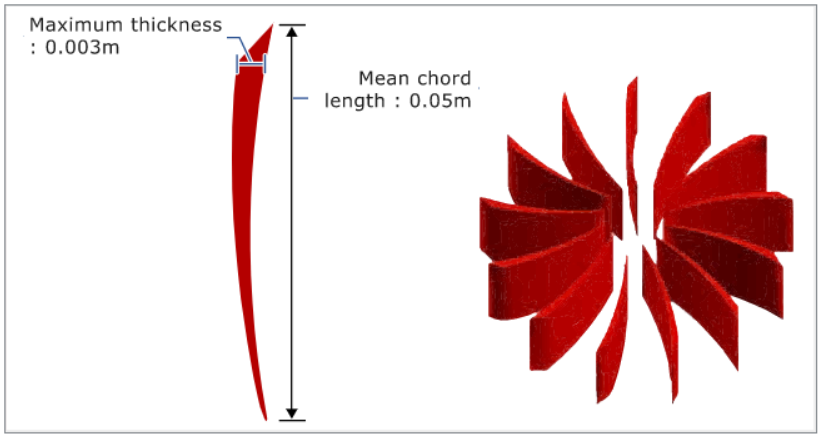


Figure 941: Schematic of Fan Blades

The boundary condition at the inlet is taken as stagnation pressure rather than mass flow rate so that AcuSolve calculates pressure rise based on impeller rotation.

The fluid in this problem is air, which has a density (ρ) of 1.225 kg/m^3 and a viscosity (μ) of 1.781 x 10^{-5} kg/m-sec.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

   Figure 942:

   The **Open File** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5000_BlowerSteady.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `CentrifugalBlower` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Blower_Steady` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 943:

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

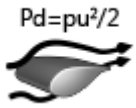
   

   Figure 944:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Verify that the **Incompressible** option is selected under Single phase flow.
   b) Set the Time marching to **Steady**.
   c) Select **Spalart-Allmaras** as the Turbulence model.

      The robustness and accuracy of the Spalart Allmaras turbulence model makes it an excellent choice for simulation of steady state flows.

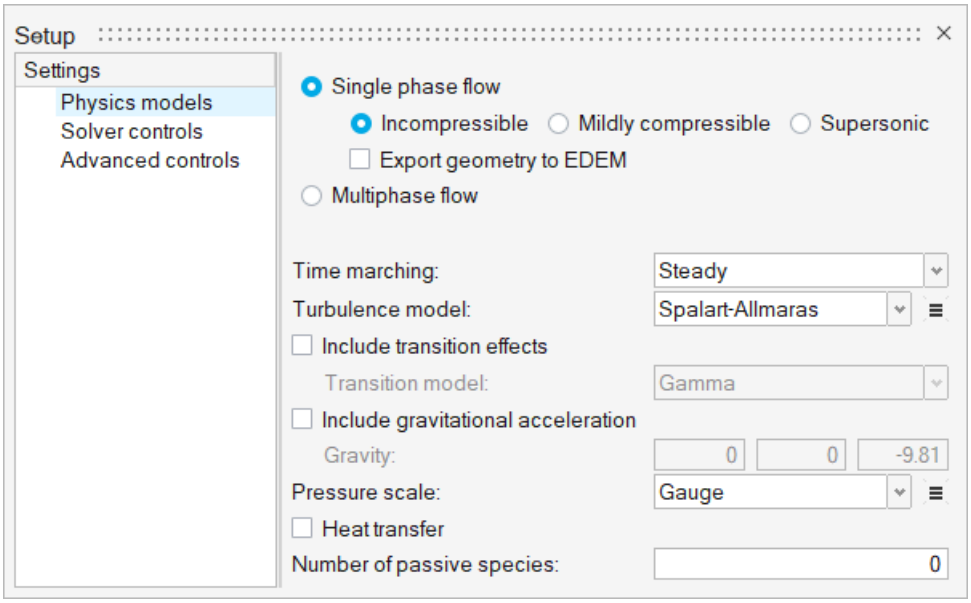


Figure 945:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

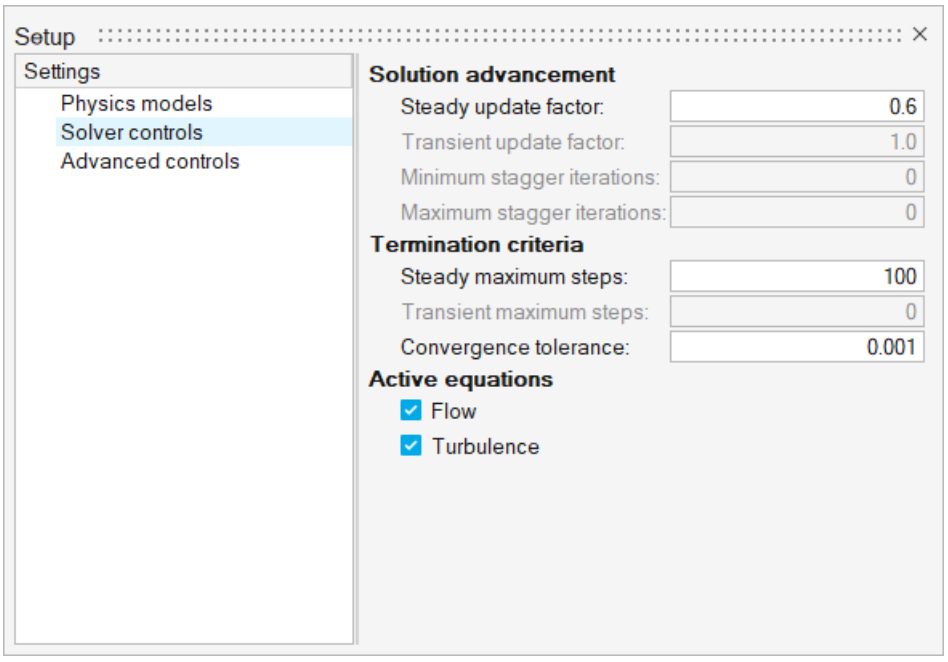


Figure 946:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 947:

2. Using window selection, draw a box around the entire model.
Both the centrifugal blower and the housing solids are selected.

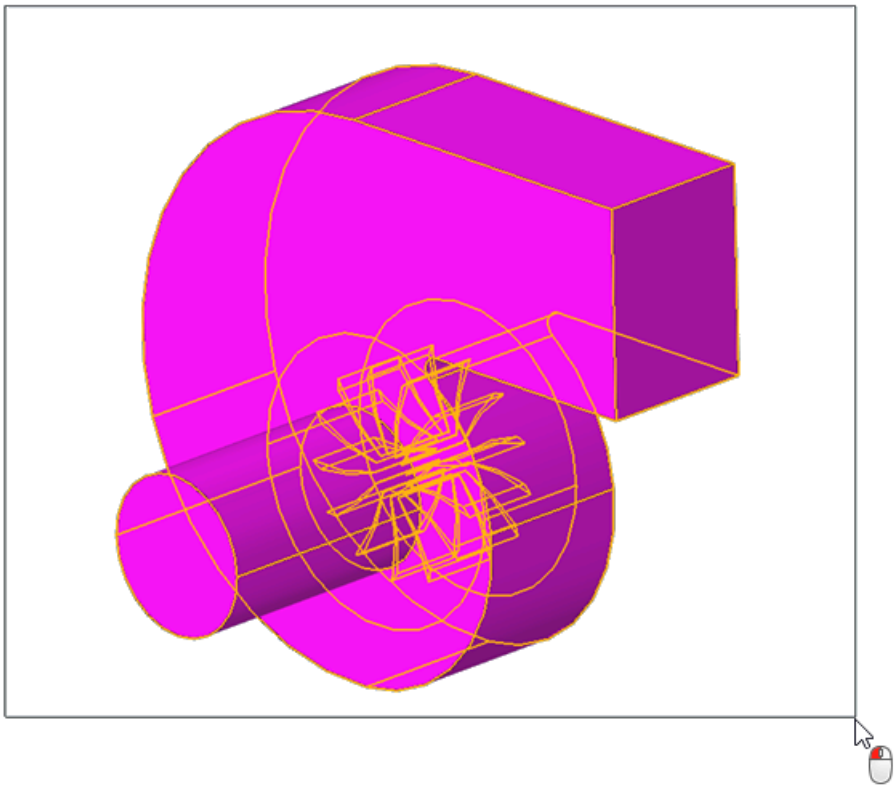


Figure 948:

3. In the microdialog, click the drop-down menu next to Material and select **Air**.
4. On the guide bar, click ✓ to execute the command and exit the tool.

Define the Reference Frame

In this step, you will create a rotating reference frame for the fluid in the impeller region so that the elements in those regions are solved in the given rotating reference frame and rotational body forces are added to that volume set.

1. Hide the housing solid.

a) Set the entity selector to **Solids**.
b) Select the centrifugal housing.
c) Right-click and select **Hide** form the context menu or press **H**.

Only the solid for the centrifugal blower displays in the modeling window.

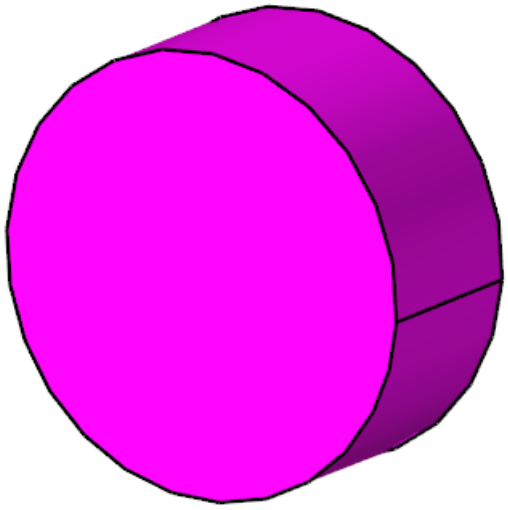


Figure 949:

2. From the Flow ribbon, click the **Reference Frame** tool.

Figure 950:

3. Make sure the Include bounding surfaces option is active on the guide bar.
4. Select the solid in the modeling window.
   Bounding surfaces are automatically selected.
5. On the guide bar, click **Axis**.
6. Define the axis of rotation.
   a) Use the Surf Center snap point to place the axis in the middle of the centrifugal blower.

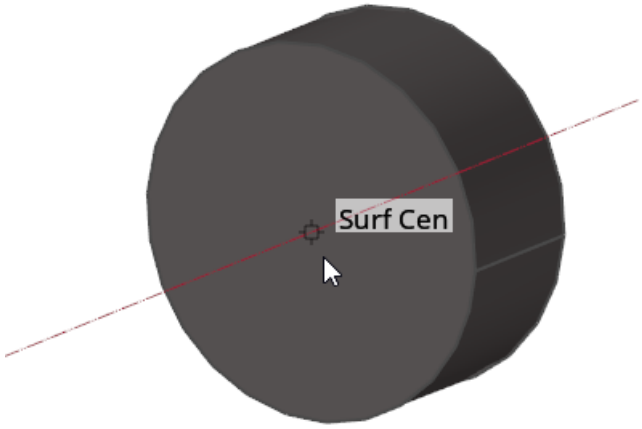


Figure 951:

b) In the microdialog, click **Z** to align the axis with the global z axis.

c) Click ⁺∕₋ to flip the spin direction.

d) Enter a value of `157.09` in the text field.

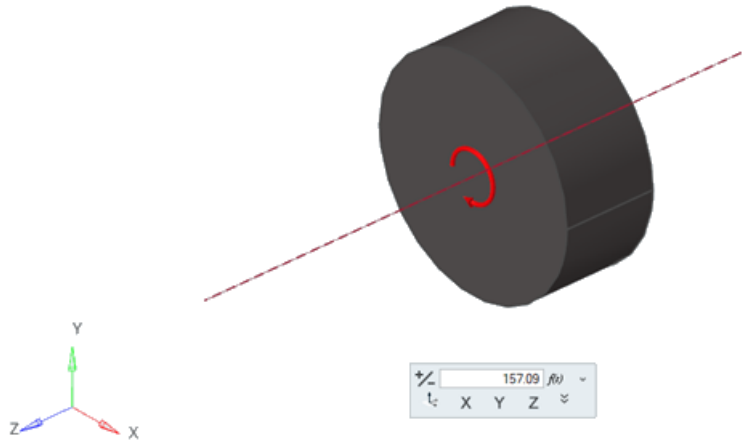


Figure 952:

7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Right-click in the modeling window and select **Show All** from the context menu or press **A** to return to the full model display.

Define Flow Boundary Conditions

1. From the Flow ribbon, Pressure tool group, click the **Stagnation Pressure** tool.

Figure 953:

2. Click the face of the inlet.

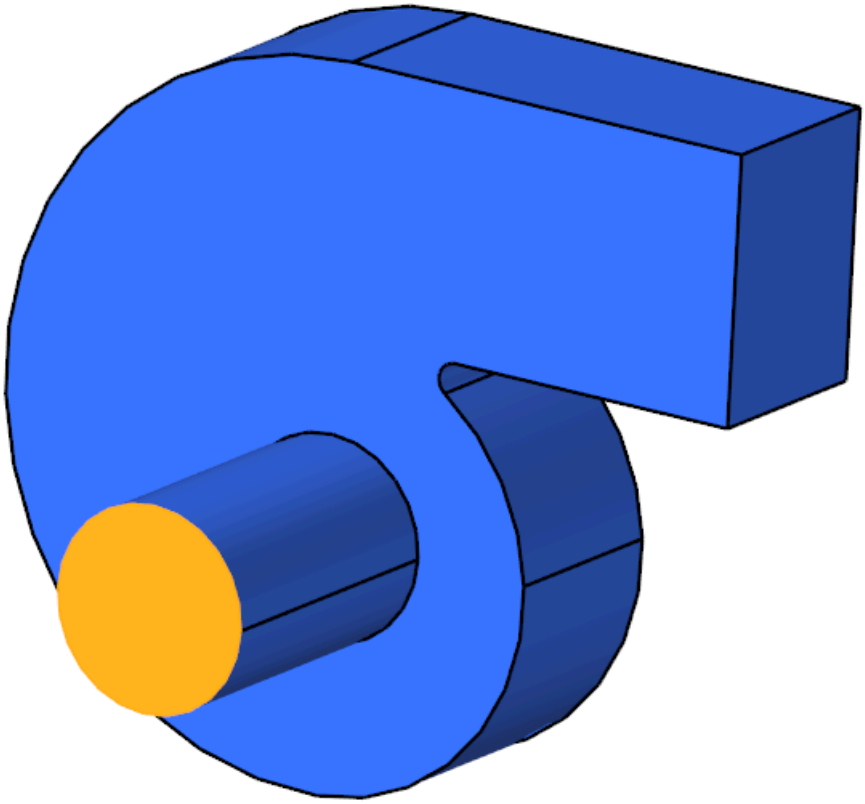


Figure 954:

3. In the microdialog, click the **Turbulence** tab.
4. Set the Turbulence input type to **Viscosity Ratio**.
5. Set the Turbulence viscosity ratio to `10`.

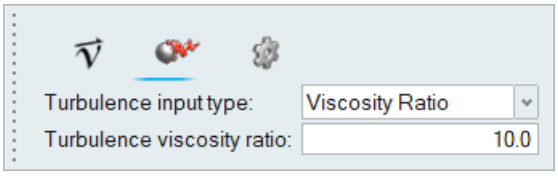


Figure 955:

6. Rename the inlet.

a) From the legend on the left side of the modeling window, double-click on **Stagnation pressure**.

b) Type `Inlet` and press **Enter**.

7. On the guide bar, click ✓ to execute the command and exit the tool.

8. Click the **Outlet** tool.

Figure 956:

9. Click the face of the outlet.

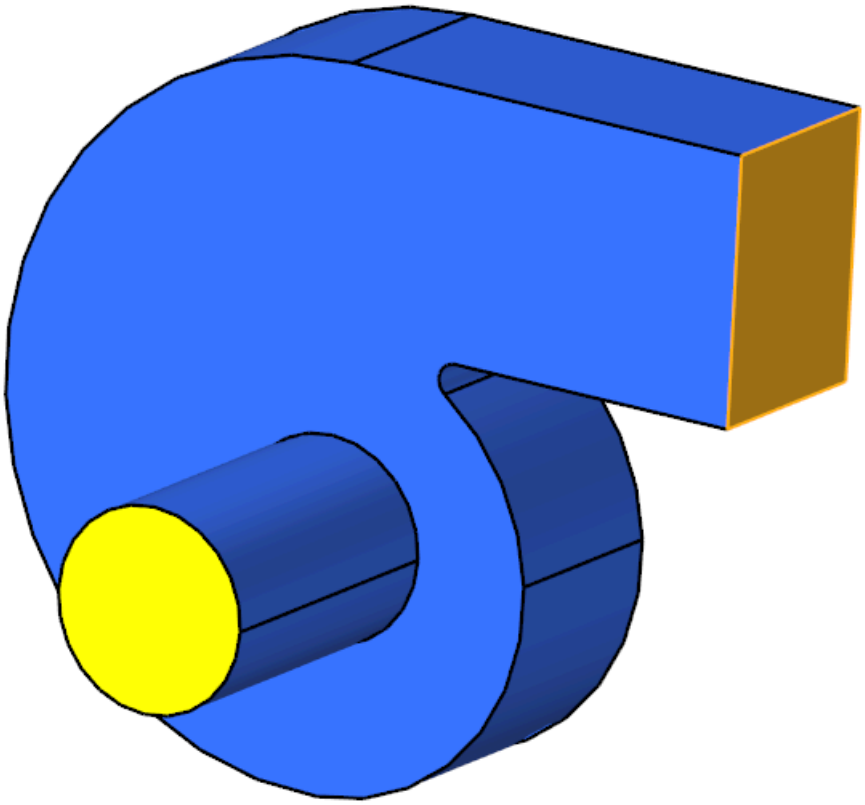


Figure 957:

10. In the microdialog, make sure both Static pressure and Pressure loss factor are 0.

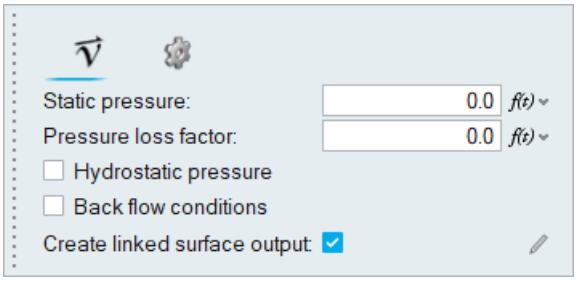


Figure 958:

11. On the guide bar, click ✓ to execute the command and exit the tool.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

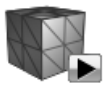


Figure 959:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.01061.

3. Accept all other default parameters.

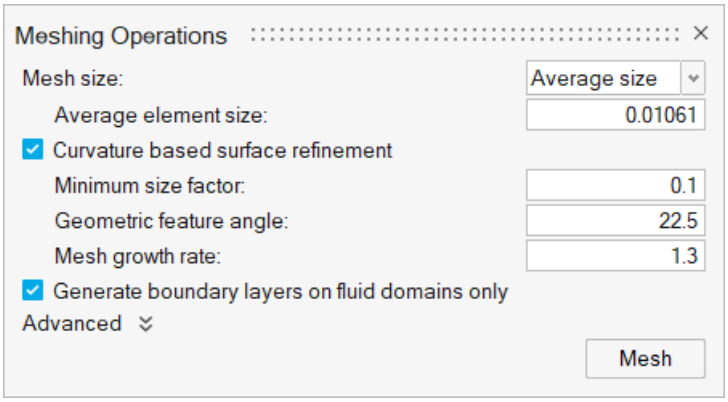


Figure 960:

4. Click **Mesh**.

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Define a Surface Monitor and Run AcuSolve

1. Set the entity selector to **Solids**.
2. Select the centrifugal housing.
3. Right-click and select **Hide** form the context menu or press **H**.
   The solid for the centrifugal blower should be displayed in the modeling window

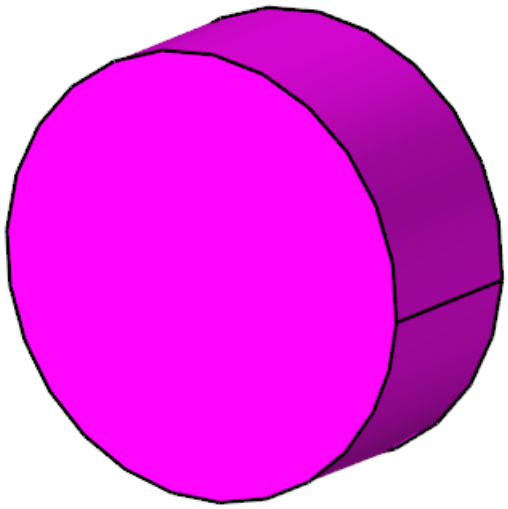


Figure 961:

4. From the Solution ribbon, click the **Surfaces** tool.

Figure 962:

5. Select the blower interface and verify that the arrow is heading toward the blower, as shown in figure below.

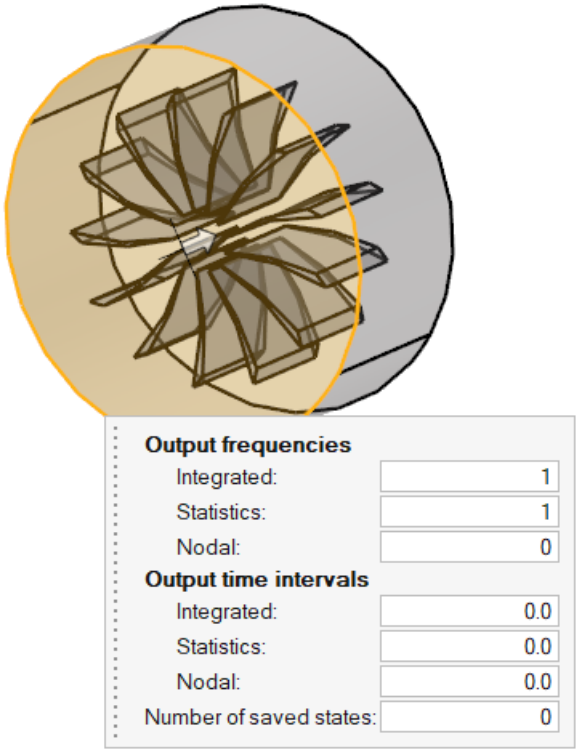


Figure 963:

6. On the guide bar, click ✓ to execute the command and exit the tool.
7. Rename surface_output to `blowerInterfaceFront`.
8. From the Solution ribbon, click the **Run** tool.

Figure 964:

The **Launch AcuSolve** dialog opens.

9. Set the Parallel processing option to **Intel MPI**.
10. Optional: Set the number of processors to **4** or **8** based on availability.
11. Deactivate the **Automatically define pressure reference** option.
12. Leave the remaining options as default and click **Run** to launch AcuSolve.

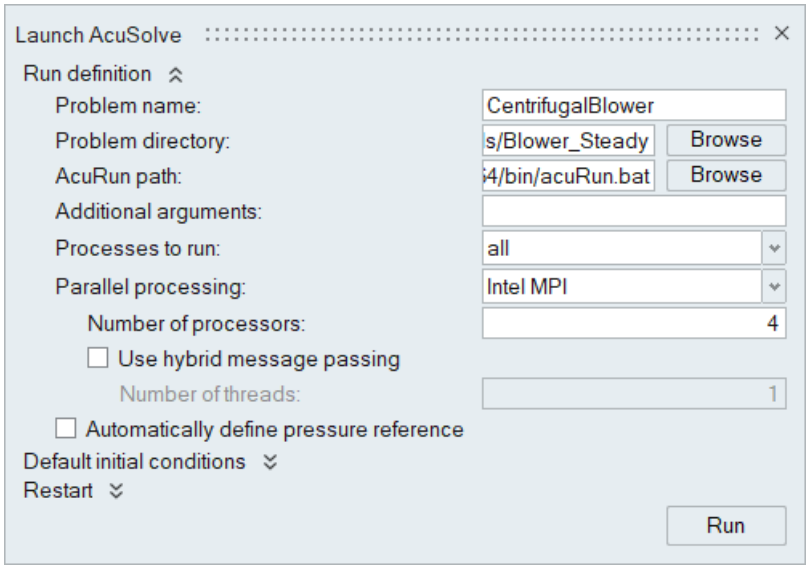


Figure 965:

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

Plot Pressure on a Slice Plane

1. Navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Slice Planes** tool.

Figure 966:

5. Select the x-y plane in the modeling window.
6. In the microdialog, click and move the plane along its normal direction a distance of `-0.07`.

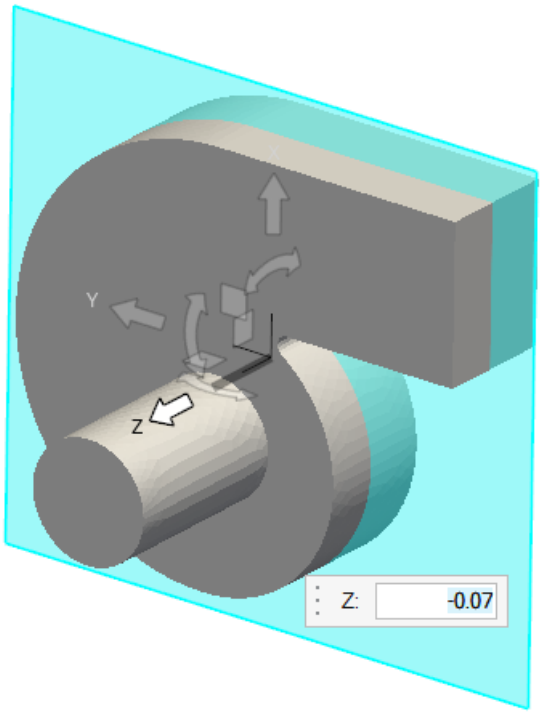


Figure 967:

7. Press **Esc** to exit the Move tool.
8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, toggle the **Legend** radio button.
10. Click and set the Colormap Name to **Rainbow Uniform**.

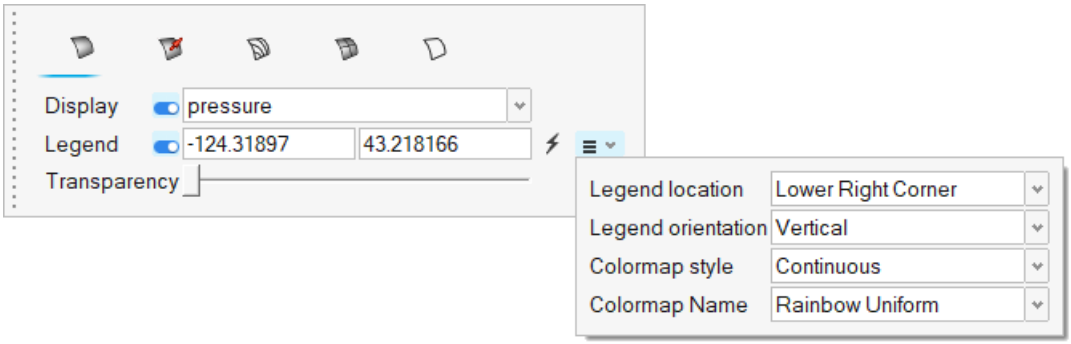


Figure 968:

11. On the guide bar, click ✓ to execute the command and exit the tool.
12. In the Post Browser, hide all the Parts and Flow Boundaries.

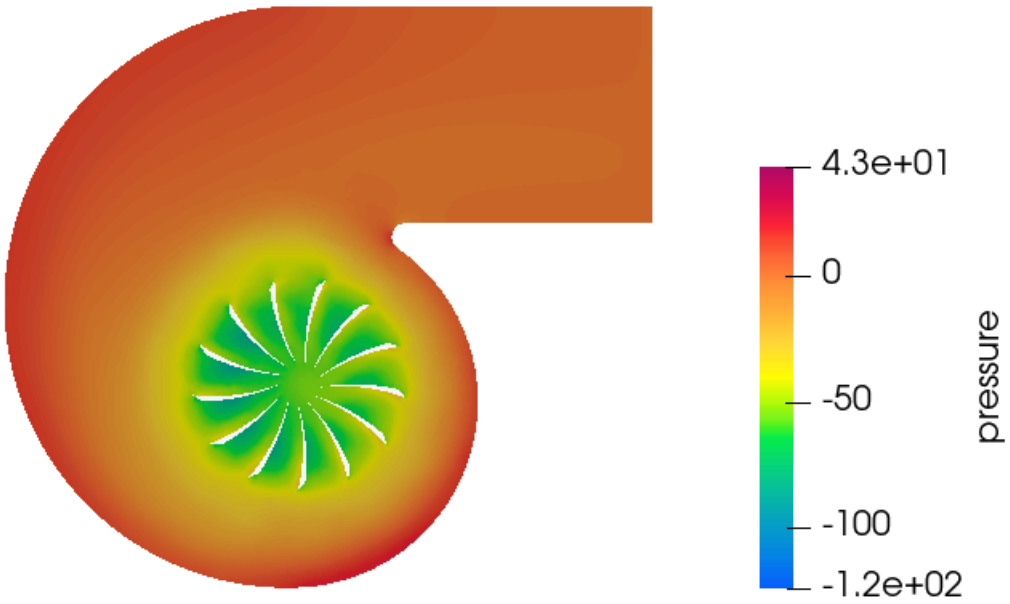


Figure 969:

Plot Velocity Streamlines

1. Hide the slice plane and show the **Inlet** and **Outlet** boundaries in the Post Browser.
2. Click the **Streamlines** > **Circle** tool.

Figure 970:

3. Select the x-y plane in the modeling window.
4. In the microdialog, click and move the plane along its normal direction a distance of `-0.07`.

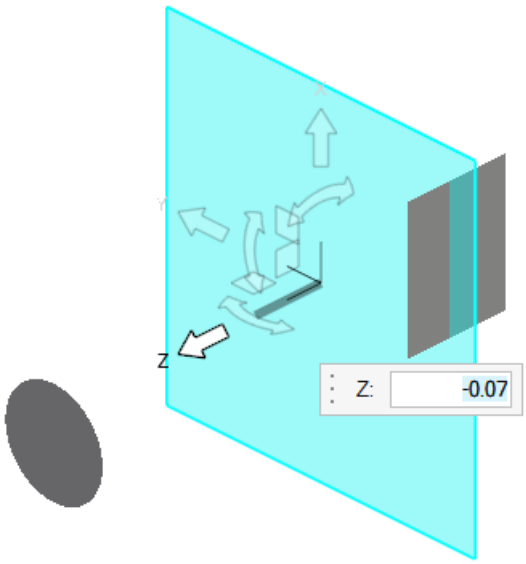


Figure 971:

5. Press **Esc** to exit the Move tool.
6. In the slice plane microdialog, click ⌄⌄ and set the circle radius to `0.1`.
7. On the guide bar, click **Seeds**.
8. In the microdialog, set the Generation method to **Uniform** and set the Point count to `100`.
9. Click **Calculate**.

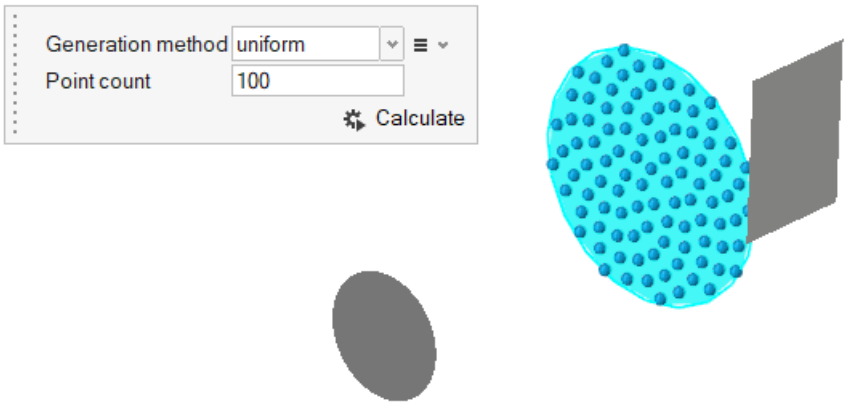


Figure 972:

10. In the display properties microdialog, set the Display to **Velocity** and set the Colormap Name to **Rainbow Uniform**.
11. On the guide bar, click ✓ to execute the command and exit the tool.

12. In the Post Browser, show the **Auto** boundary condition, then right-click on **Auto** and select **Edit**.
13. In the microdialog, adjust the transparency slider until you can see the velocity streamlines.
14. Click ✓ on the guide bar.

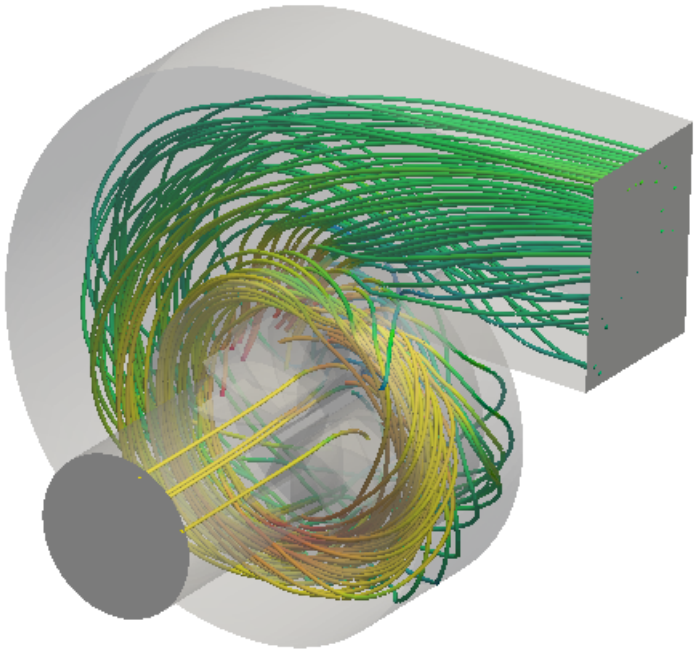


Figure 973:

Summary

In this tutorial, you successfully learned how to set up a steady state simulation involving a rotating reference frame in a centrifugal blower. You started by importing the mesh and then once the case was set up, you generated a solution using AcuSolve. Then, you created a contour plot for pressure and velocity streamlines using HyperMesh CFD post.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/hwpeba01q3

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5001 Blower - Transient (Sliding Mesh)

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial provides the instructions for setting up, solving, and viewing results for a transient simulation of a centrifugal air blower utilizing the sliding mesh approach. In order to run this tutorial, you should have already run through ACU-T: 5000 Blower - Steady (Rotating Frame) and kept the solution in your working directory. It is assumed that you have some familiarity with HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5001_BlowerTransient.hm`

Since the HyperMesh CFD database (.hm file) uses the ACU-T:5000 Blower-Steady model, this tutorial skips the geometry clean-up and inlet boundary conditions as well as the mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 974 and Figure 975. It consists of a centrifugal blower with forward curved blades. The fluid through the inlet plane enters the hub of the blade wheel, radially accelerates due to centrifugal force as it flows over the blades, and then exits the blower housing through the outlet plane.

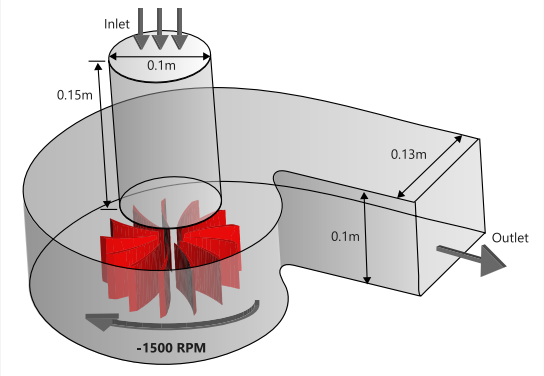


Figure 974: Schematic of Centrifugal Blower

The fan blades have a mean chord length and width of 0.05 m. The maximum thickness of the blades is 0.003 m.

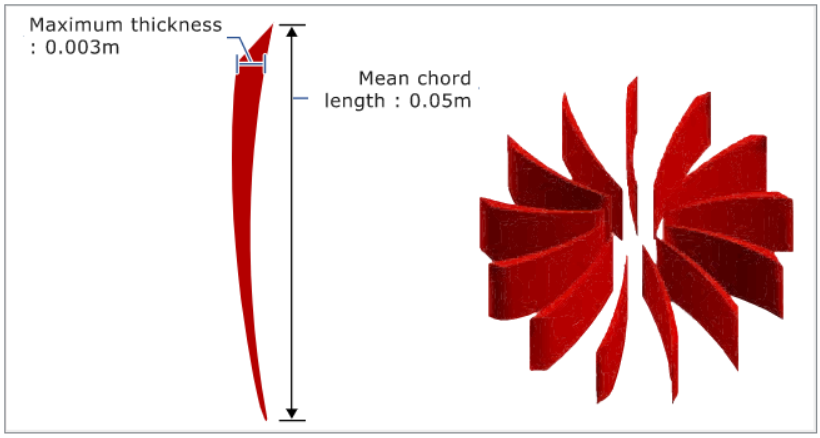


Figure 975: Schematic of Fan Blades

To capture the dynamic motion of the impeller blades, the simulation must be transient. The time step size is determined by:

$$\omega = 1500 \text{RPM} = 25 \text{ RPS} = 9000 \text{ deg/s} \tag{4}$$

$$\Delta t = \frac{\text{blade rotation angle per time step}}{\omega} = \frac{4.5}{9000} = 0.0005 \text{sec} \tag{5}$$

where ω is the blower rotational speed. It should be noted that a time step size sensitivity study should always be performed to establish appropriate time step size when analyzing a new application.

The CFD analysis of this problem offers detailed information about the flow through a centrifugal blower. To investigate this behavior, it is necessary to select an appropriate set of boundary conditions to use.

There are two different methods that are commonly used. One approach is to specify the mass flow rate at the inlet of the blower and allow AcuSolve to compute the pressure drop, that is, flow forces simulation. Another option is to specify the stagnation pressure at the inlet and allow AcuSolve to compute the flow rate that results from this specified pressure change between the inlet and outlet. The boundary conditions used in this example are the latter. That is, the inlet is taken as stagnation pressure rather than mass flow rate so that AcuSolve calculates mass flow rates and pressure rise based on impeller rotation.

The fluid in this problem is air, which has a density of 1.225 kg/m^3 and a viscosity of 1.781 X 10^{-5} kg/m-s.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 976:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5001_BlowerTransient.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `BlowerTransient` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `BlowerTransient_solved` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

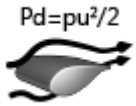


Figure 977:

The **Setup** dialog opens.

2. Under the Physics models settings:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.0005` and the Final time to `0.05`.

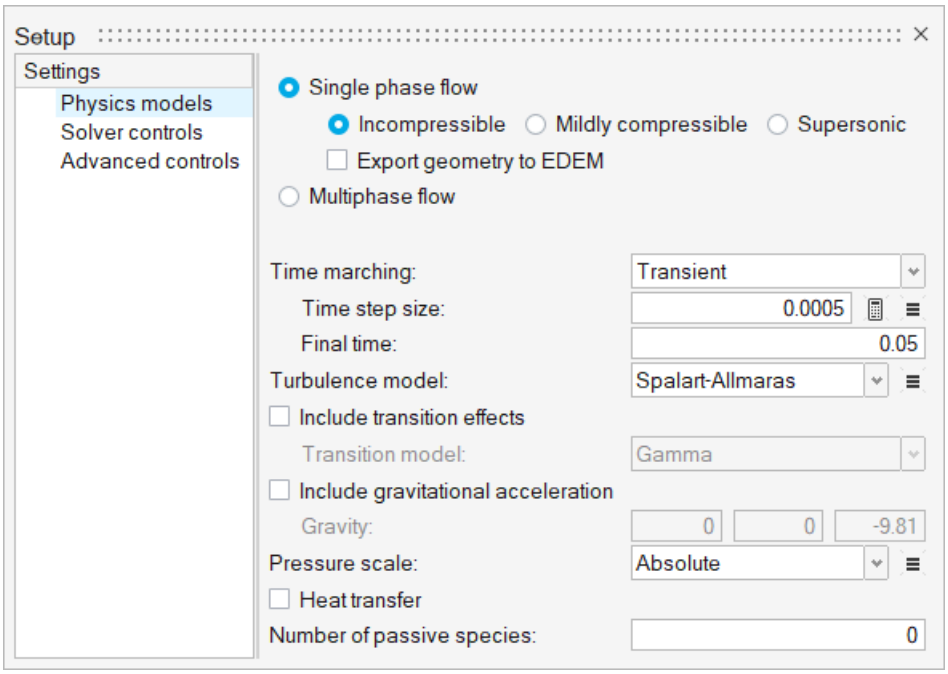


Figure 978:

3. Click **Solver controls**.
4. Set the Minimum stagger iterations and Maximum stagger iterations to `2` and `6`, respectively.

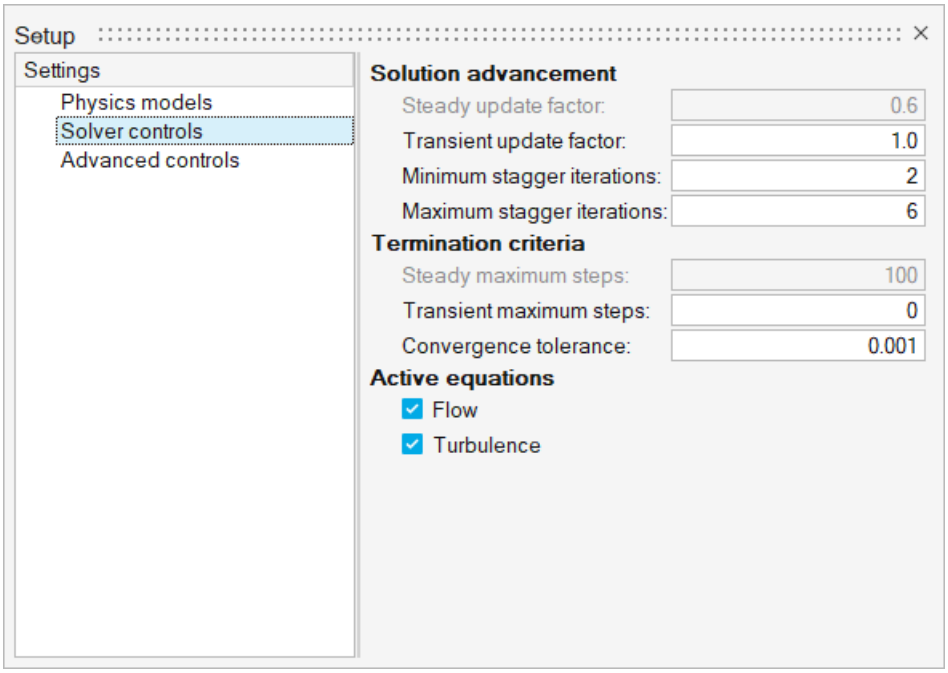


Figure 979:

5. Close the dialog and save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, Pressure tool group, click the **Stagnation Pressure** tool.

Figure 980:

2. Click the face of the inlet.

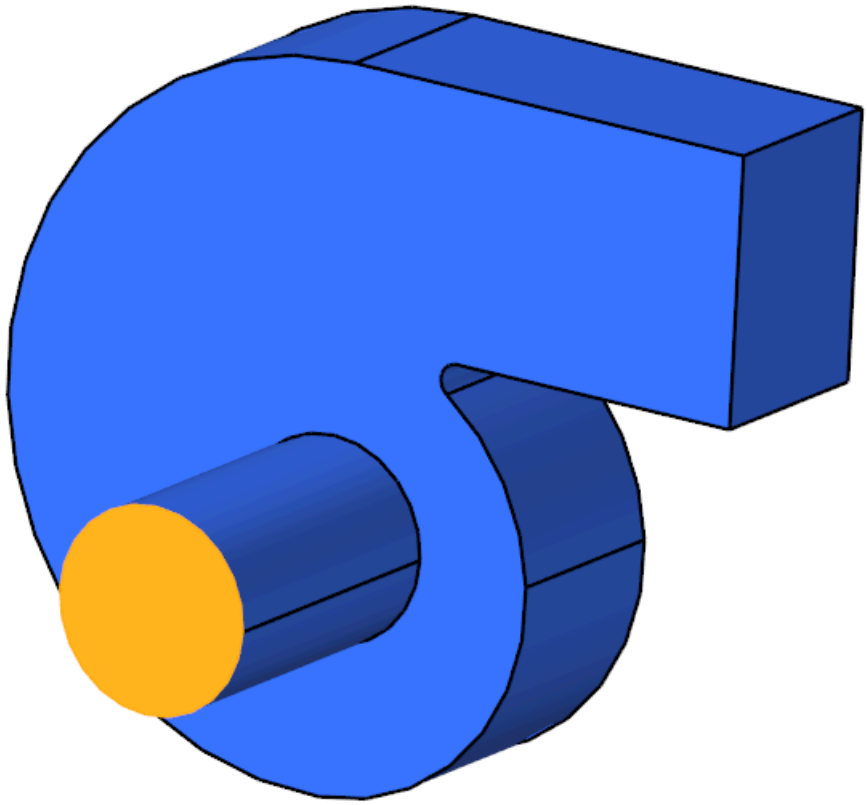


Figure 981:

3. In the microdialog, click the **Turbulence** tab.
4. Set the Turbulence input type to **Auto**.
5. Set the Turbulence flow type to **Internal**.

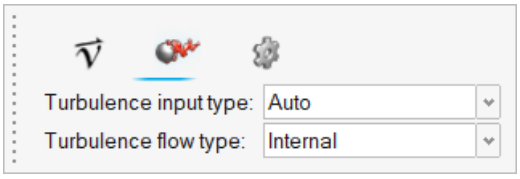


Figure 982:

6. Rename the inlet.
   a) From the legend on the left side of the modeling window, double-click on **Stagnation pressure**.
   b) Type `Inlet` and press **Enter**.
7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Click the **Outlet** tool.

Figure 983:

9. Click the face of the outlet.

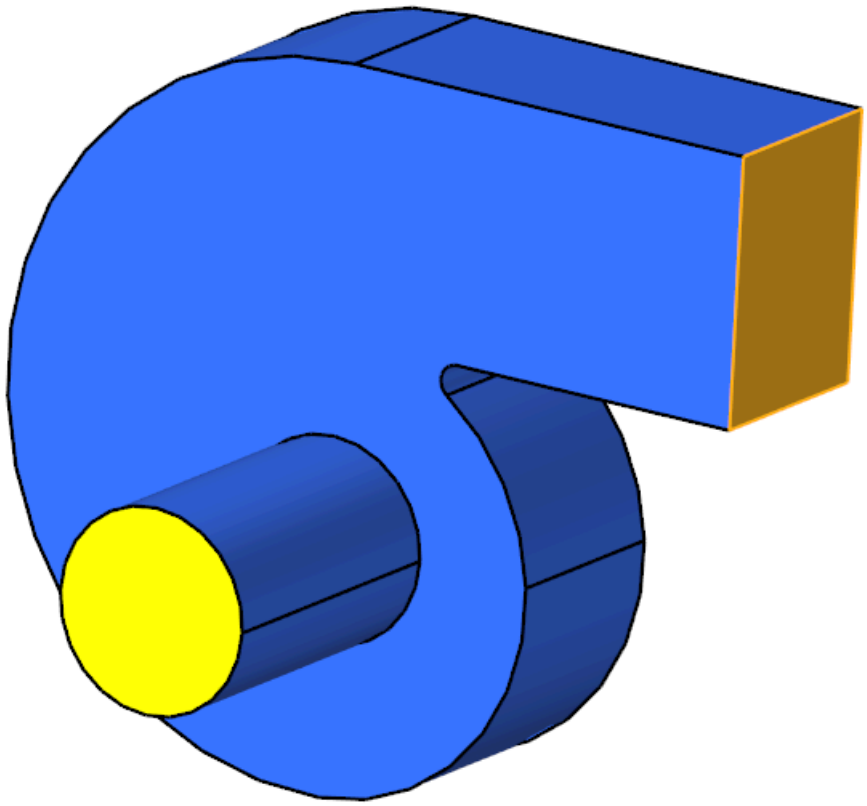


Figure 984:

10. In the microdialog, make sure both Static pressure and Pressure loss factor are 0.

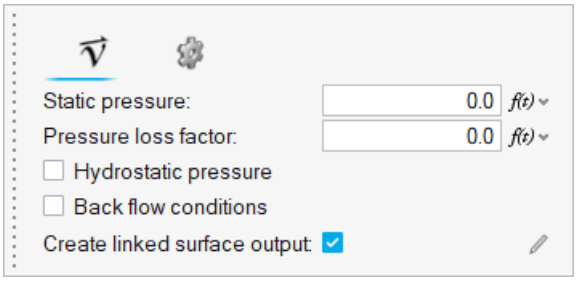


Figure 985:

11. On the guide bar, click ✓ to execute the command and exit the tool.

Set Up Mesh Motion

Define the Mesh Motion Type

1. From the Motion ribbon, click the **Settings** tool.

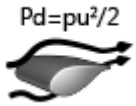


Figure 986:

The **Setup** dialog opens.

2. Change the Mesh motion type to **Specified**.
3. Click the **Solver controls** setting and make sure the **Mesh deformation** checkbox is checked.

Define the Rotating Mesh Motion

1. Hide the housing solid.
   a) Set the entity selector to **Solids**.
   b) Select the centrifugal housing.
   c) Right-click and select **Hide** or press **H** on your keyboard.

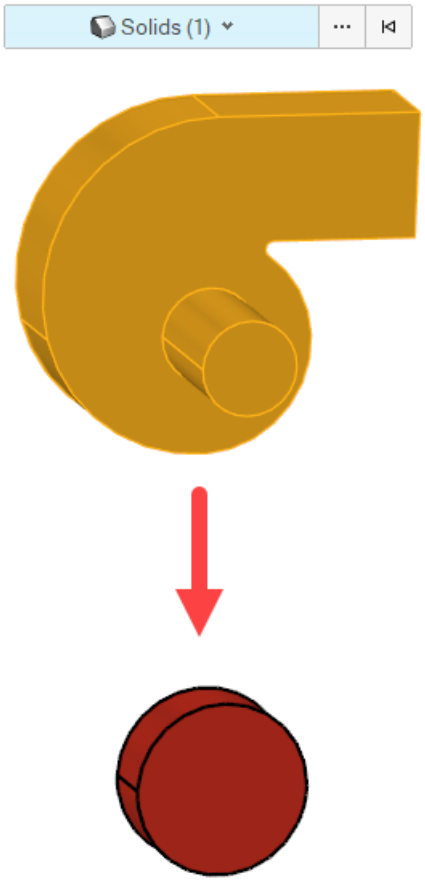


Figure 987:

2. Click the **Rotation** tool.

Figure 988:

3. Select the solid in the modeling window.
4. On the guide bar, click **Axis**.
5. Define the axis of rotation.
   a) Use the Surf Center snap point to place the axis in the middle of the centrifugal blower.

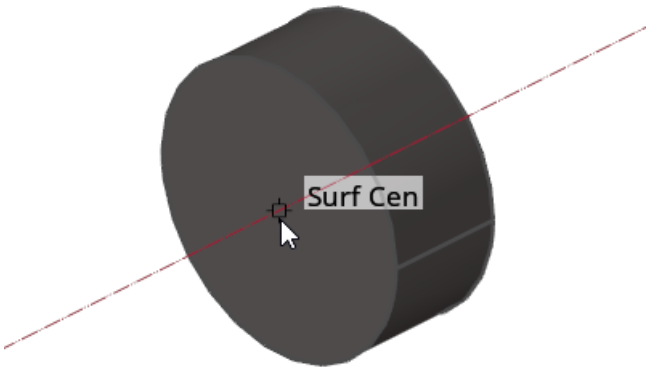


Figure 989:

b) In the microdialog, click **Z** to align the axis with the global z axis.

c) Click +/- to change the rotational direction.

d) Enter a value of `157.09` in the text field.

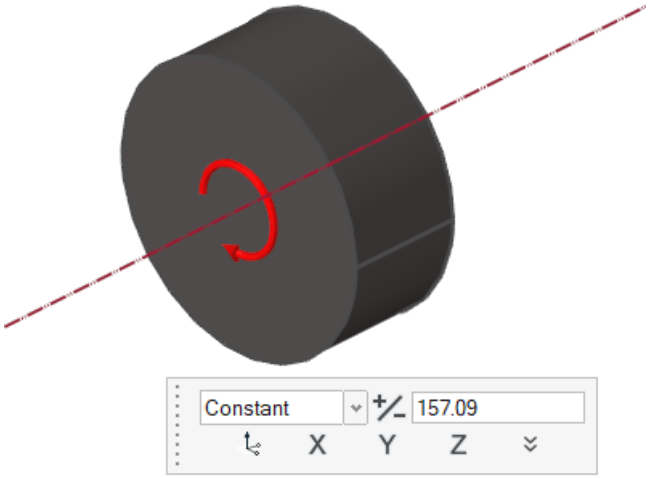


Figure 990:

6. On the guide bar, click ✓ to execute the command and exit the tool.
7. Right-click in the modeling window and select **Show All** or press **A** on your keyboard to return to the full model view.

Define Nodal Output Frequency

1. From the Solution ribbon, click the **Field** tool.

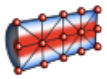


Figure 991:

The **Field Output** dialog opens.

2. Check the **Write initial conditions** checkbox.
3. Check the **Write results at time step interval** checkbox.
4. Set the Time step interval to `1`.

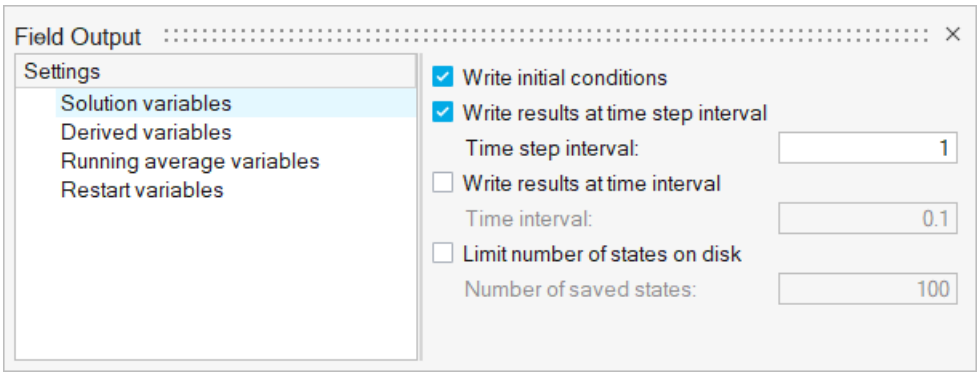


Figure 992:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 993:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the Automatically define pressure reference option.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

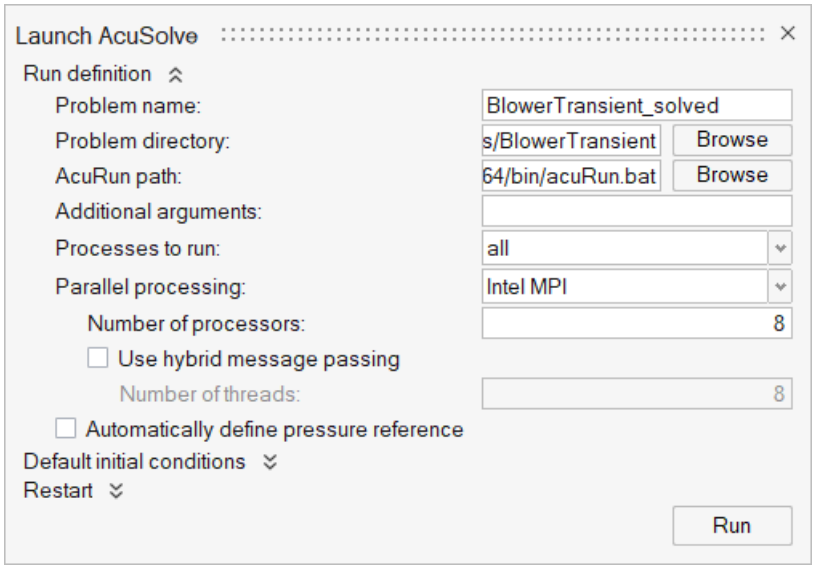


Figure 994:

> **Tip:** While AcuSolve is running, right-click on the AcuSolve job in the **Run Status** dialog and select **View Log File** to monitor the solution process.

Post-Process the Results with HM-CFD Post

In this step, you will plot the pressure on a slice plane.

1. Navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Slice Planes** tool.

Figure 995:

5. Select the x-y plane in the modeling window.
6. In the microdialog, click and move the plane along its normal direction a distance of `-0.07`.

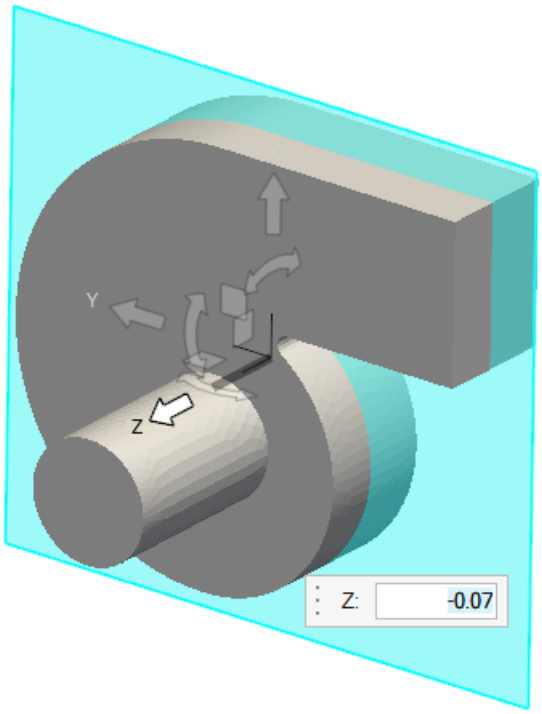


Figure 996:

7. Press **Esc** to exit the Move tool context.
8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, toggle the **Legend** radio button and set the bounds to `-120` and `43`.
10. Click and set the Colormap Name to **Rainbow Uniform**.

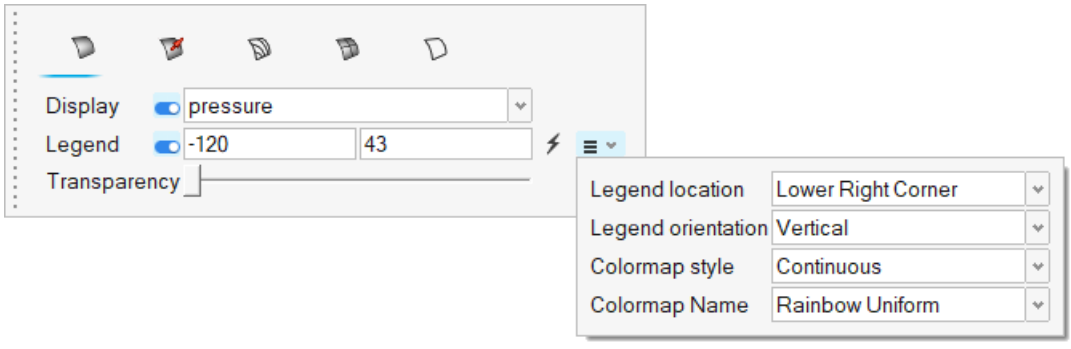


Figure 997:

11. On the guide bar, click ✓ to execute the command and exit the tool.
12. In the Post Browser, hide all the Parts and Flow Boundaries.
13. Click the play button on the Animation toolbar.

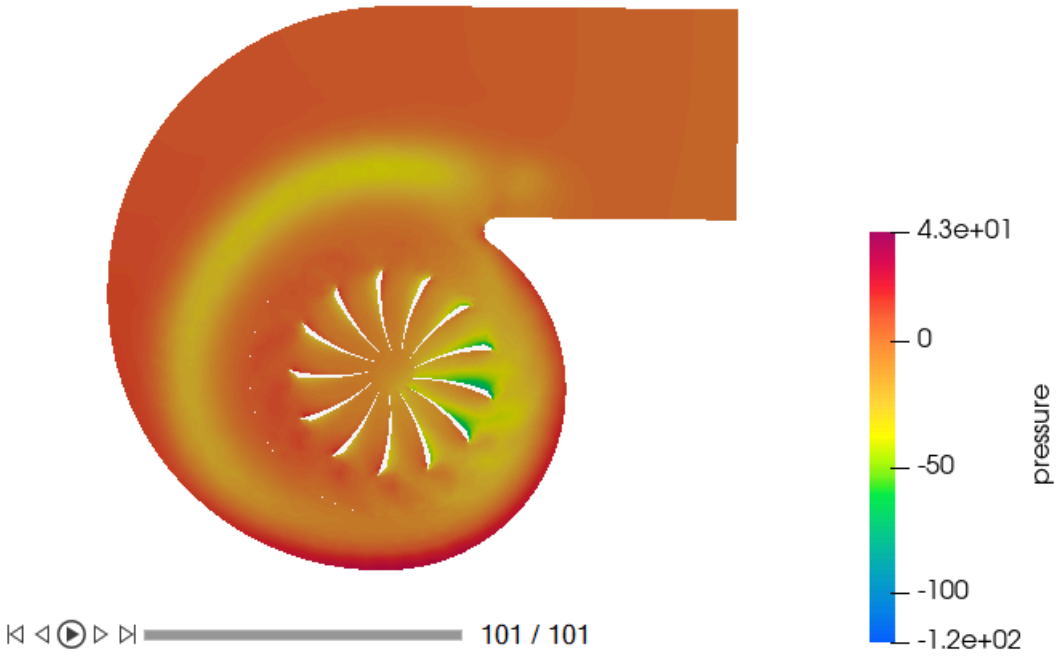


Figure 998:

14. Go to **File** > **Export** > **Results** to save the pressure contour animation.

Summary

In this tutorial, you worked through a basic workflow to set up a transient simulation with a sliding mesh in a centrifugal blower. You started by importing the mesh; once the case was set up, you generated a solution using AcuSolve. Then, you computed the blower momentum using the Plot tool and created a contour plot for pressure on a cut plane using HyperMesh CFD post.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve.

This tutorial demonstrates how to set up a transient centrifugal air blower simulation and visualize the results. Before you begin, download the SimLab database file.

You will need an internet connection to watch the video tutorials and download the files.

Tutorial Video

https://fast.wistia.net/embed/iframe/lxsozcufgv

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T:5002 Centrifugal Blower – Hybrid (MRF + MM)

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

This tutorial works directly from the solution setup. Before you begin download the SimLab database

Tutorial Video

https://fast.wistia.net/embed/iframe/on4yxcfyzu

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5100 Modeling of a Fan Component Using the Fan Component - Coefficient Method

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This simulation provides instructions for modeling a fan using the classic fan component. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5100_CoefficientFanComponent.hm`

`FanAxialCoefficients.csv`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be solved in this tutorial is shown schematically in the figure below. It consists of an interior fan which rotates at a speed of 377 rad/sec (~3600 RPM) and has a thickness of 0.06 m and a tip radius of 0.11 m. The volumetric flow rate at the inlet is 0.322675 m^3/sec (~1161.63 m^3/hr). The problem is simulated as a steady state run and the pressure rise across the fan region is computed.

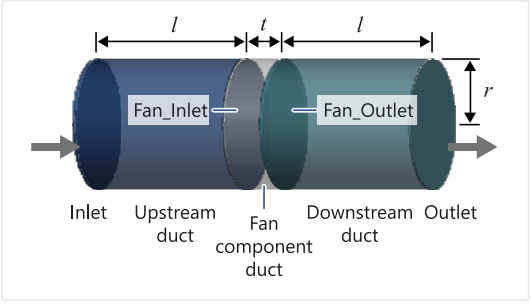


Figure 999:

Figure 999 shows a simple axial fan component problem where the fan is an interior fan with thickness "t" and tip radius "r". In this simulation, flow is passed from the pipe inlet. It enters the fan in an axial

direction and exits at the outlet causing pressure rise. This fan pressure rise can be simulated for a given volume flow rate at the inlet surface which will be assigned as the inflow boundary condition. The middle portion of the pipe is the Fan Component which has both Fan_Inlet and Fan_Outlet.

The Fan Component directly computes the body force term to yield the pressure rise within the volume of interest. The body force is applied directly to the volume and not the surface. In order to yield the required pressure rise based on fan performance curve that the you have input, you need to evaluate the volume flow rate correctly at the inlet surface of the domain.

A fan component is modeled by adding the body force to the momentum equations, which increases the pressure across the component by:

$$\Delta P_{axial} = \frac{1}{2} a_{axial} \rho u_{tip}^2 + \frac{1}{2} \rho \bar{u}^2$$

Where:

a_{axial} - axial coefficient of the fan component

u_{tip} - tip velocity (m/sec) = ωr_{tip}

ω - fan angular rotational speed (rad/sec)

r_{tip} - fan tip radius (m)

$\bar{u}$ - mass averaged velocity (m/sec) through inlet

The fan pressure rise is determined by the fan performance (P-Q) curve which is mainly dependent on the volume flow rate and fan speed for the required fan pressure rise (ΔP). Since piecewise_bilinear curve fit values used in the classic FAN_COMPONENT approach are functions of the normalized flow rate and axial coefficient, you need to convert them from the fan performance curve.

Normalized flow rate $\left(Q'\right) = \frac{Q}{A\, u_{tip}}$

Where:

Q - flow rate

A - area

Axial coefficient $(a_{axial}) = \frac{2\Delta P - \rho \bar{u}^2}{\rho u_{tip}^2}$

For example, evaluate the axial coefficients and normalized flow rate from the fan performance data. The following tables are inputs for the calculations.

Table 3: Fan characteristics

Fluid Density	1.225 kg/m^3
Tip Radius (r_{tip} - fan tip radius (m))	0.11m
Rotational Speed (ω)	3600 RPM = 376.99 rad/sec
Inlet Area, A_i	0.03801 m^2

Tip Velocity (u_{tip})	41.47 m/sec

Table 4: Fan performance parameters

S.No	Volume flow rate (Q), m^3/hr	Pressure rise (dP), Pa
1	525.35	494.91
2	890.21	474.63
3	1161.63	424.90
4	1272.76	389.11
5	1356.57	350.42
6	1431.84	308.18
7	1494.69	268.35
8	1551.39	230.89

You can calculate the normalized flow rate and axial coefficient for first two volume flow rates (Q) from Table 4. The same procedure is followed for the other volume flow rates.

1. For Q = 525.35 m^3/hr:

$$Q^1 = \frac{Q}{A\,u_{tip}} = 0.0926$$

$$\alpha_{axial} = \frac{2\Delta P - \rho \bar{u}^2}{\rho u_{tip}^2} = 0.4613$$

2. For Q = 890.21 m^3/hr:

$$Q^1 = \frac{Q}{A\,u_{tip}} = 0.1569$$

$$\alpha_{axial} = \frac{2\Delta P - \rho \bar{u}^2}{\rho u_{tip}^2} = 0.426$$

In this manner we calculate Q^l and α_{axial} for the remaining volume flow rates, shown in the following table.

Table 5: Normalized flow rates and axial coefficients

S.No	Normalized flow rate (Q^l)	Axial Coefficients (α_{axial})
1	0.0926	0.4613

S.No	Normalized flow rate (Q^I)	Axial Coefficients (α_{axial})
2	0.1569	0.426
3	0.2047	0.3615
4	0.2243	0.3191
5	0.239	0.2755
6	0.2523	0.229
7	0.2634	0.1854
8	0.2734	0.1445

The same information is entered as input for the FAN_COMPONENT parameter as shown below in Figure 1000.

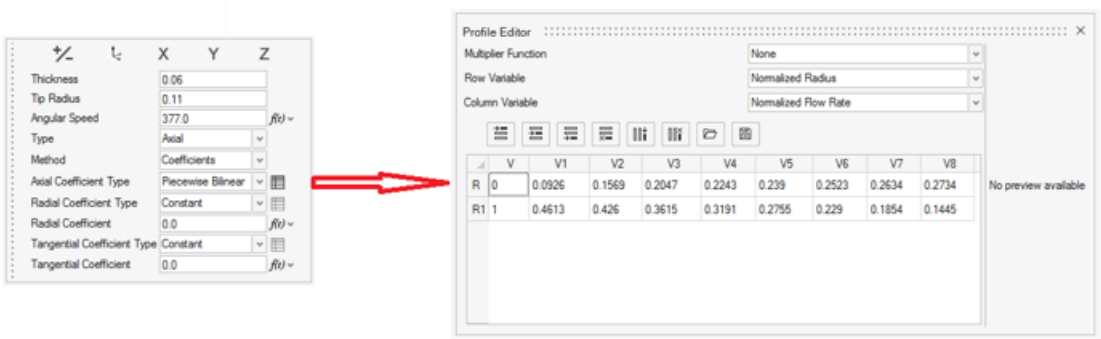


Figure 1000: Fan component Array Editor

The first column (0,1) is the normalized radius and the first row (0.0926,…,0.2734) defines the normalized flow rate. The second row data (0.4613,…,0.1445) are for the axial coefficient.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1001:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5100_CoefficientFanComponent.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Axial_Fan` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Axial_Fan` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1002:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

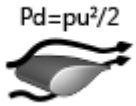
   

   Figure 1003:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.

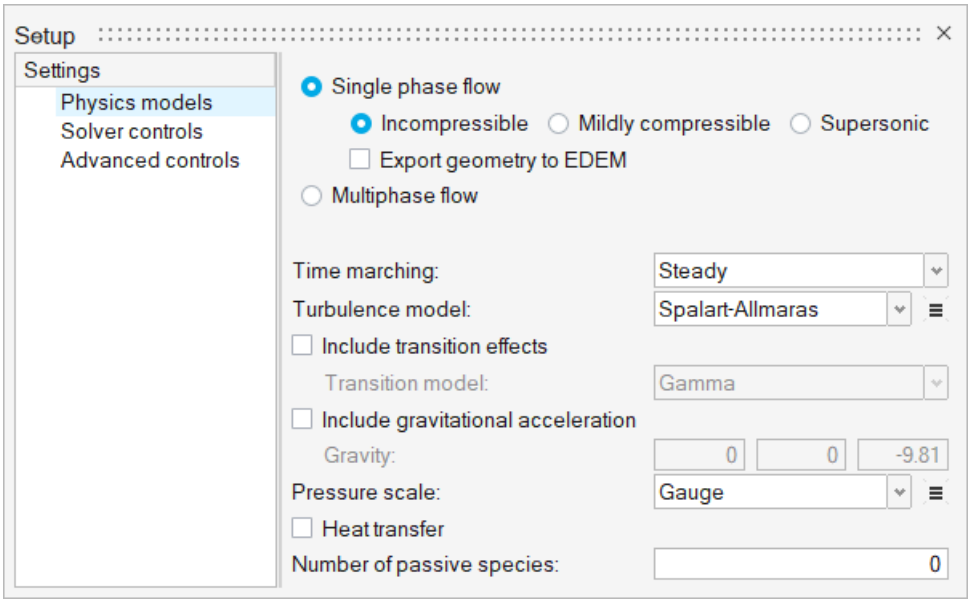


Figure 1004:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

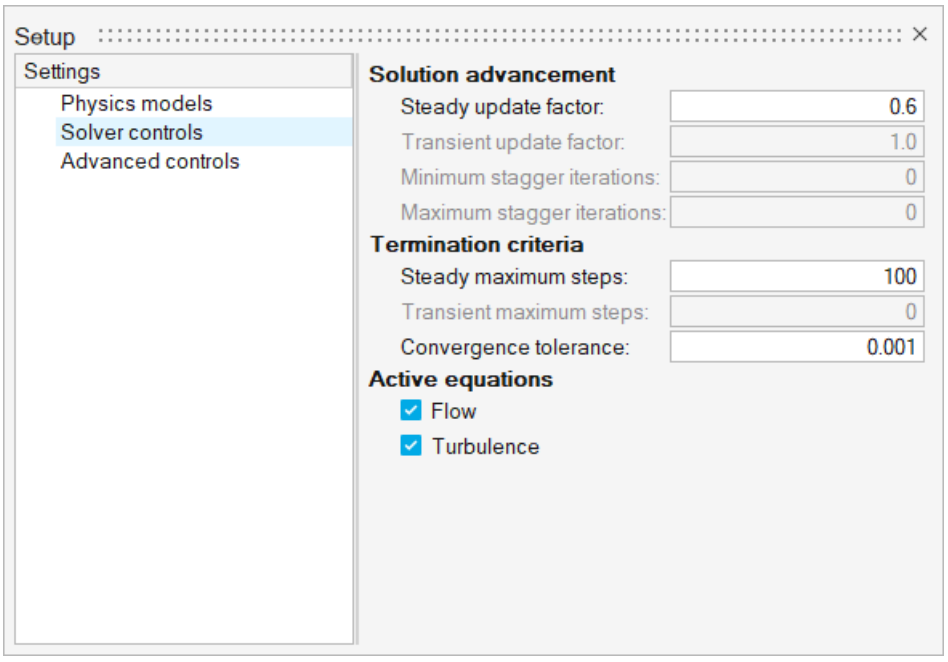


Figure 1005:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1006:

2. Verify that the Air material has been assigned to all three volumes.
3. Click ✓ on the guide bar to exit the tool.

Define the Fan Component

1. From the Flow ribbon, click the arrow next to the Domain tool set, then select **Fan Component**.

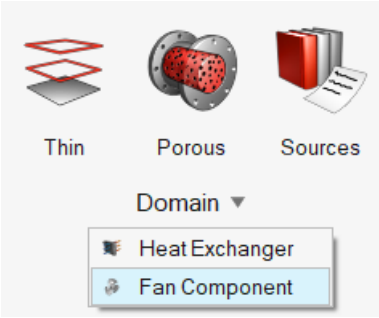


Figure 1007:

2. Select the middle solid as the fan component volume.

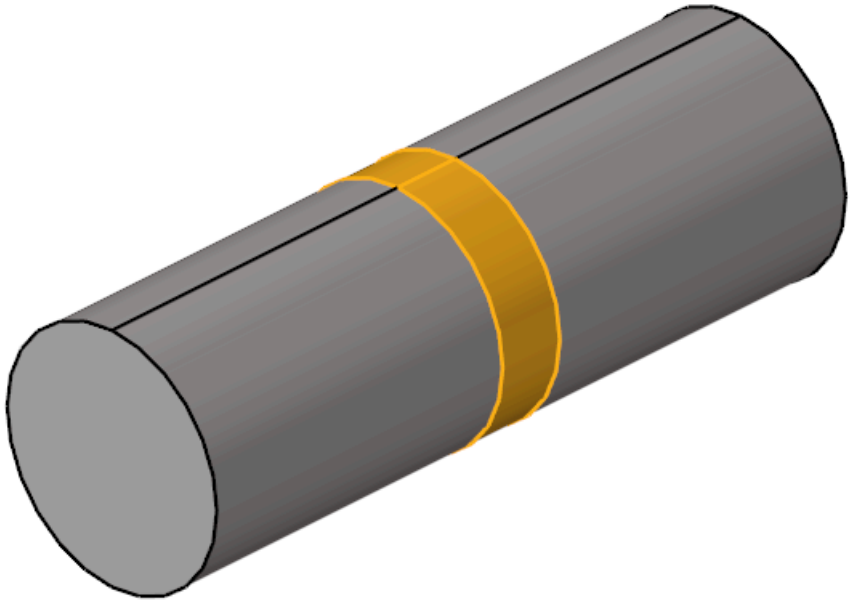


Figure 1008:

3. On the guide bar, click **Surfaces** then select the face shown below as the inlet of the fan component.

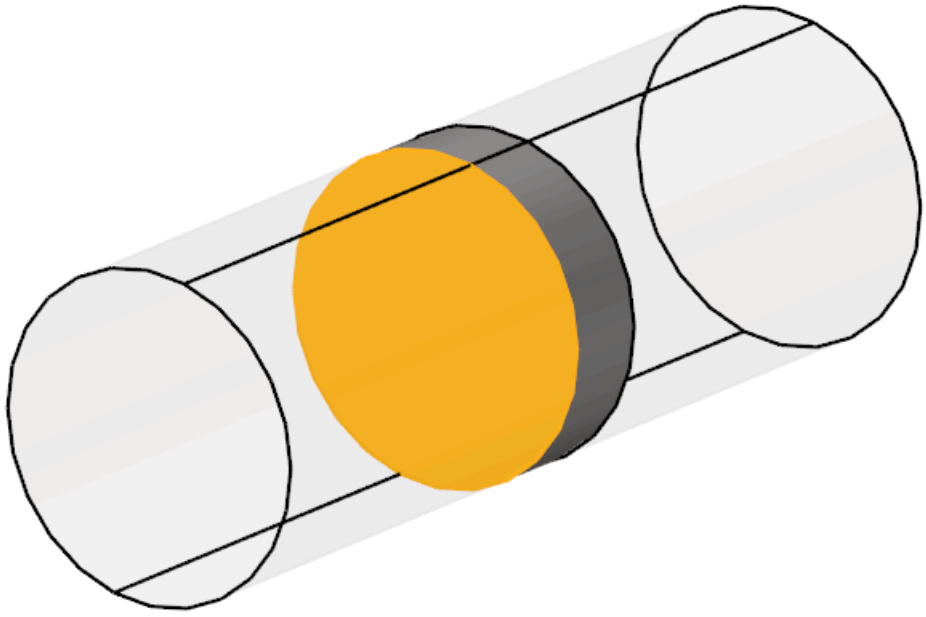


Figure 1009:

4. From the View Controls toolbar, change the geometry visualization mode from Shaded Geometry to **Transparent Geometry**.

This allows you to view the axis direction vector in the next step.

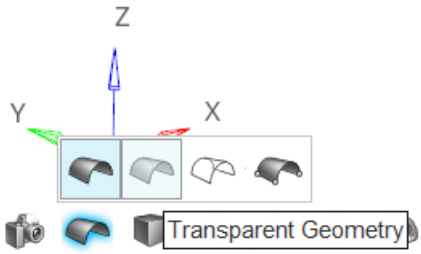


Figure 1010:

5. On the guide bar, click **Axis**.
   In the modeling window, you can see that the axis points in the -X direction.
6. Click +/- in the microdialog to flip the axis vector to the +X direction.

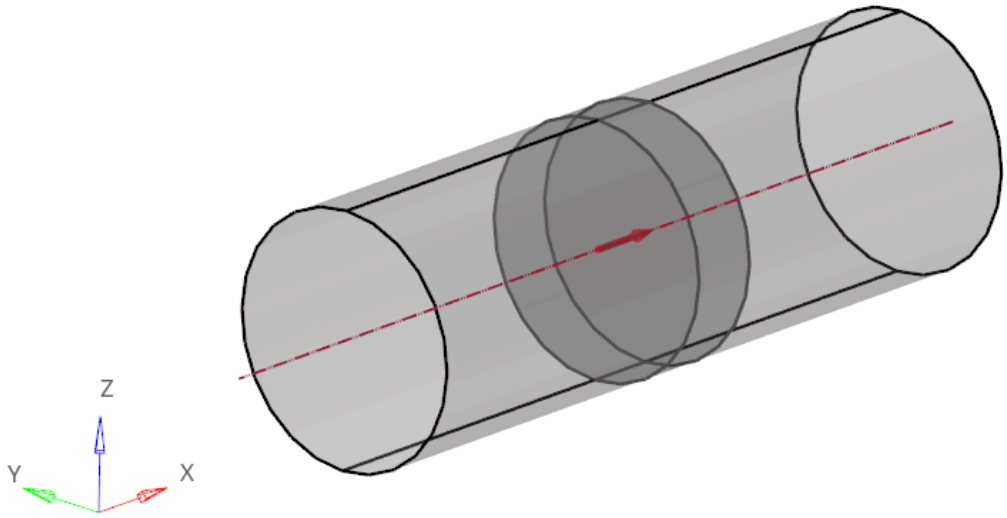


Figure 1011:

7. Enter `0.06` for Thickness (m).
8. Set the Type to **Axial** and set the Method to **Coefficients**.
9. Enter `0.11` for Tip Radius (m) and `377` for Angular Speed (rad/sec).
10. Set the Axial Coefficient Type to **Piecewise Bilinear**.

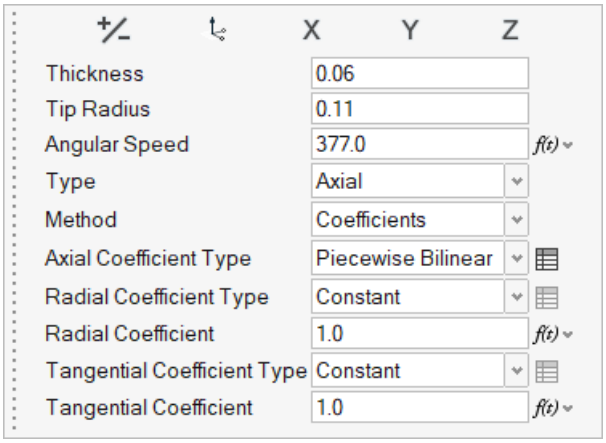


Figure 1012:

11. Click beside Piecewise Bilinear to open the Profile Editor.

12. Click , browse to the location where you saved `FanAxialCoefficients.csv`, and open it.

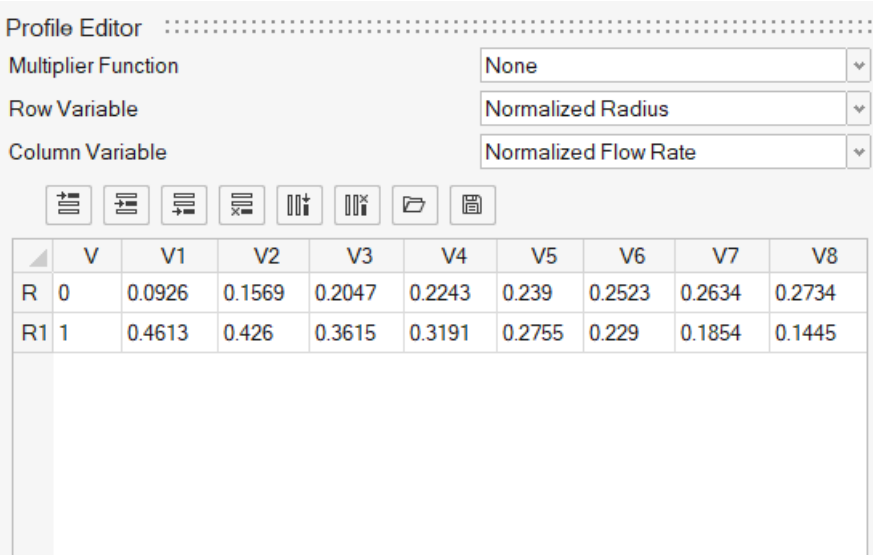


Figure 1013:

> **Note:** Both Normalized flow rate and Axial Coefficients have non-dimensionless units

13. Ensure that Row Variable is set to **Normalized Radius** and Column Variable to **Normalized Flow Rate**.

14. Close the Profile Editor.

15. Change the Radial Coefficient and Tangential Coefficient values from 1.0 to `0.0`.

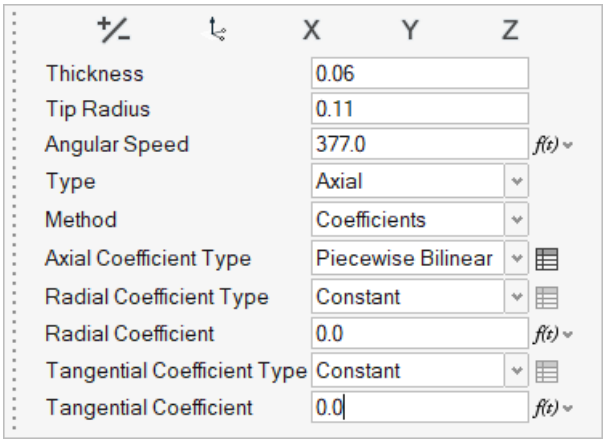


Figure 1014:

> **Note:** Make sure that both Radial and Tangential coefficient values are set to 0 since the fluid enters only in the axial direction.

16. On the guide bar, click ✓ to execute the command and exit the tool.
17. Save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Volumetric Flow Rate** tool.

Figure 1015:

2. Click the inlet face highlighted in the figure below.

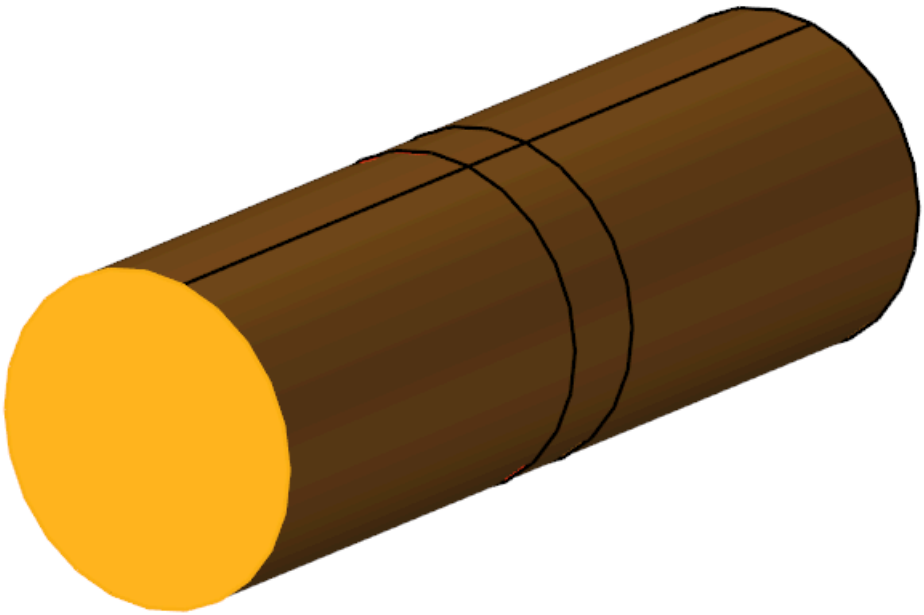


Figure 1016:

3. In the microdialog, enter `0.322675` for the volumetric flow rate (m^3/sec).

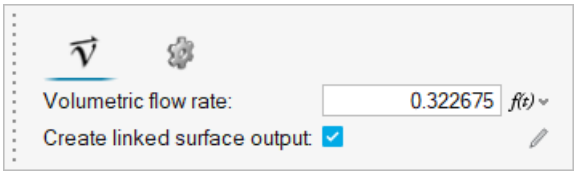


Figure 1017:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 1018:

6. Select the face highlighted in the figure below and then click ✓ on the guide bar.

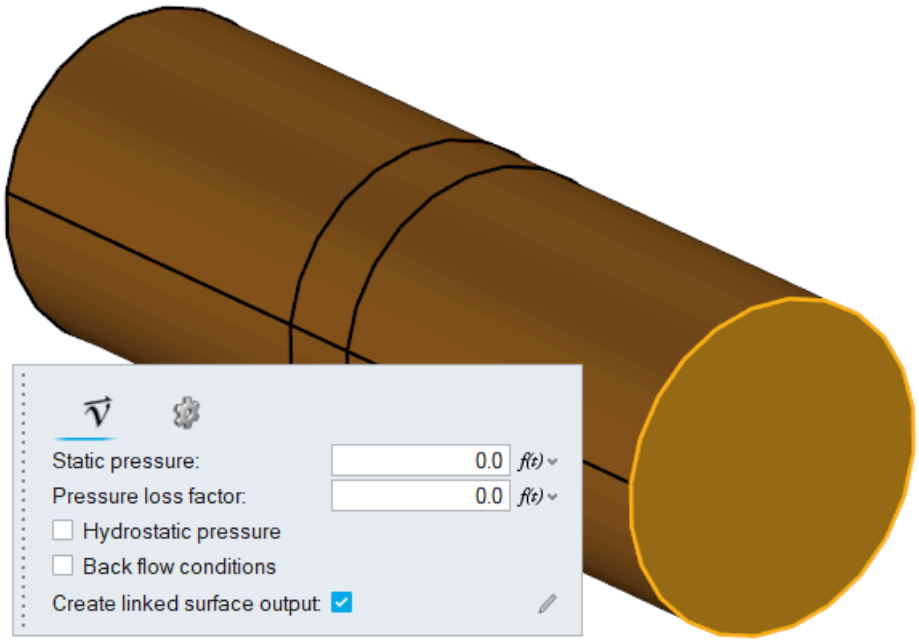


Figure 1019:

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

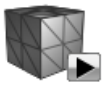


Figure 1020:

2. In the **Meshing Operations** dialog, set the Average element size to `0.01` and the Mesh growth rate to `1.1` (if not set already).

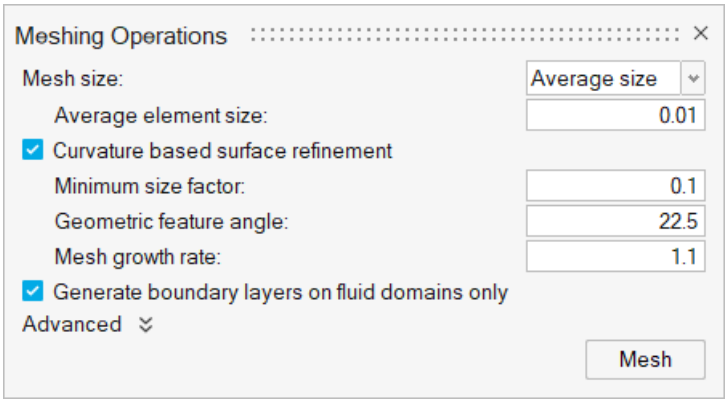


Figure 1021:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Surface Monitors and Run AcuSolve

1. Set the entity selector to **Solids**.

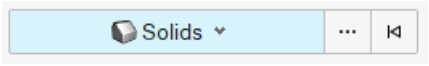


Figure 1022:

2. Select the fan component duct marked below to isolate it from front and aft ducts.

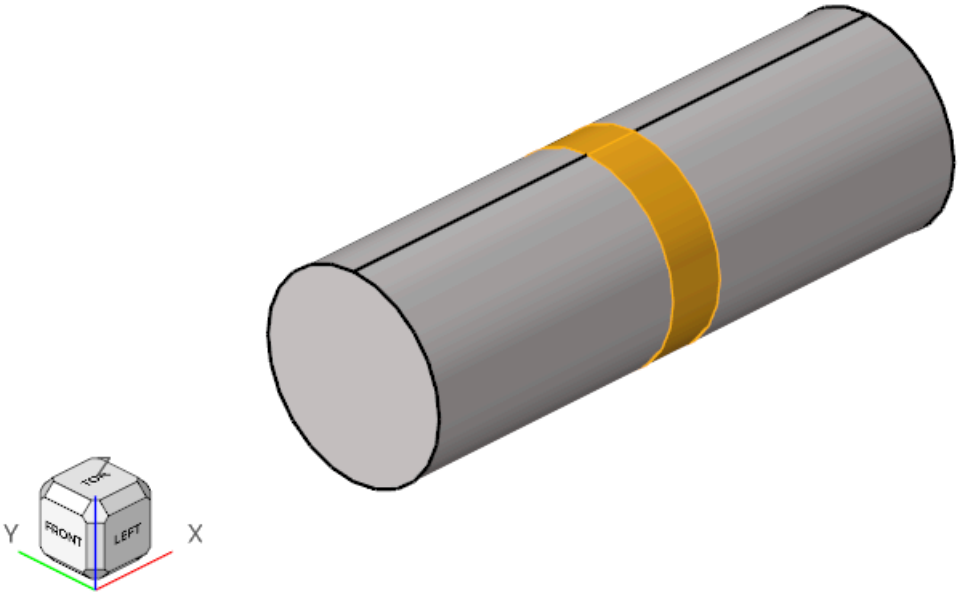


Figure 1023:

3. Right-click and select **Isolate** form the context menu or press **I**.
   The solid for the fan component should be displayed in the modeling window.

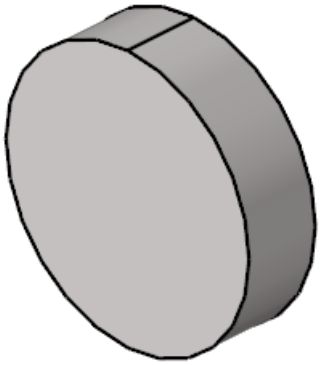


Figure 1024:

4. Form the Solution ribbon, click the **Surfaces** tool.

Figure 1025:

5. Select the front face of the fan and verify that the arrow is heading toward the fan component, as shown in figure below

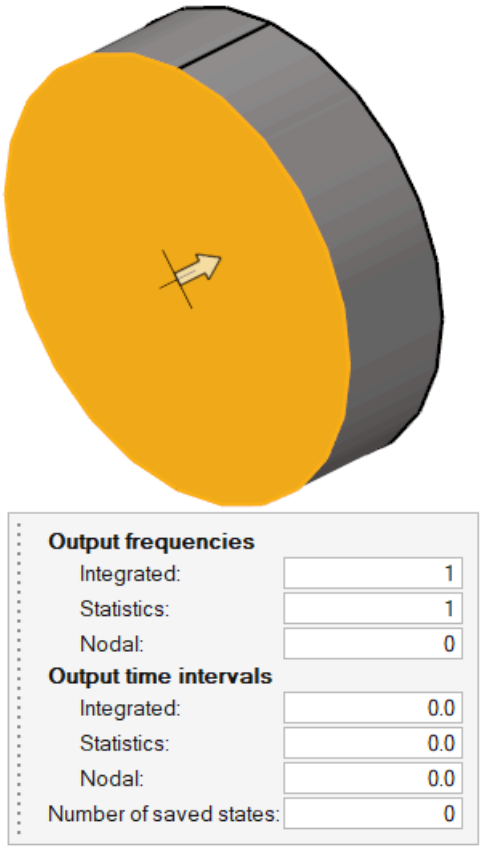


Figure 1026:

6. On the guide bar, click ▸ to execute the command and remain in the tool.
7. In the legend, rename surface_output to `FAN_inlet`.
8. Rotate the fan component and select the rear face.

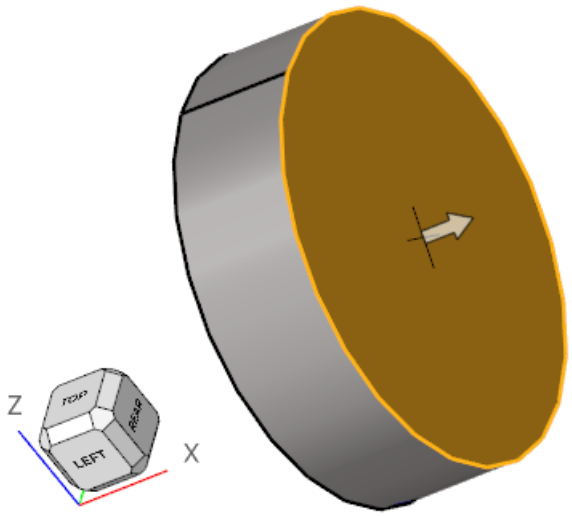


Figure 1027:

9. Click ▸, rename surface_output to `FAN_outlet` in the legend, then click ✓ to exit the tool.
10. From the Solution ribbon, click the **Run** tool.

Figure 1028:

The **Launch AcuSolve** dialog opens.

11. Set the Parallel processing option to **Intel MPI**.
12. Optional: Set the number of processors to **4** or **8** based on availability.
13. Leave the remaining options as default and click **Run** to launch AcuSolve.

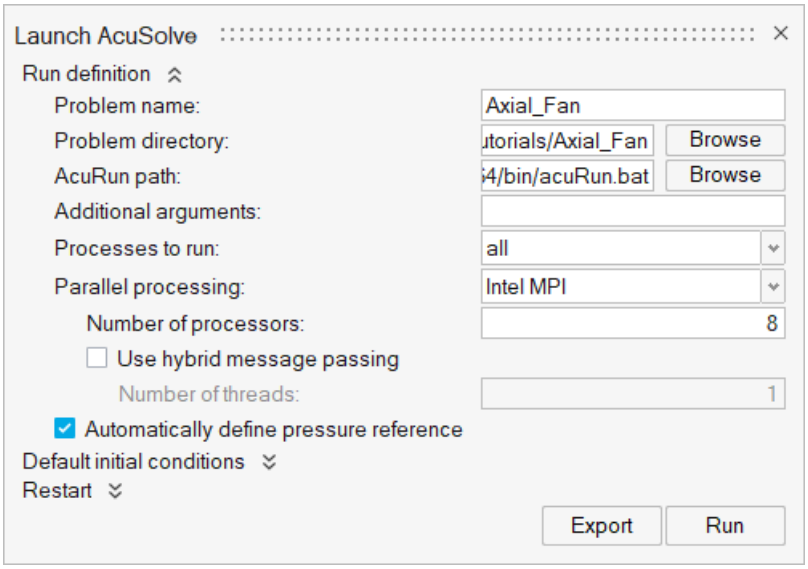


Figure 1029:

Post-Process with the Plot Tool

1. From the Solution ribbon, click the **Plot** tool.

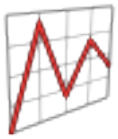


Figure 1030:

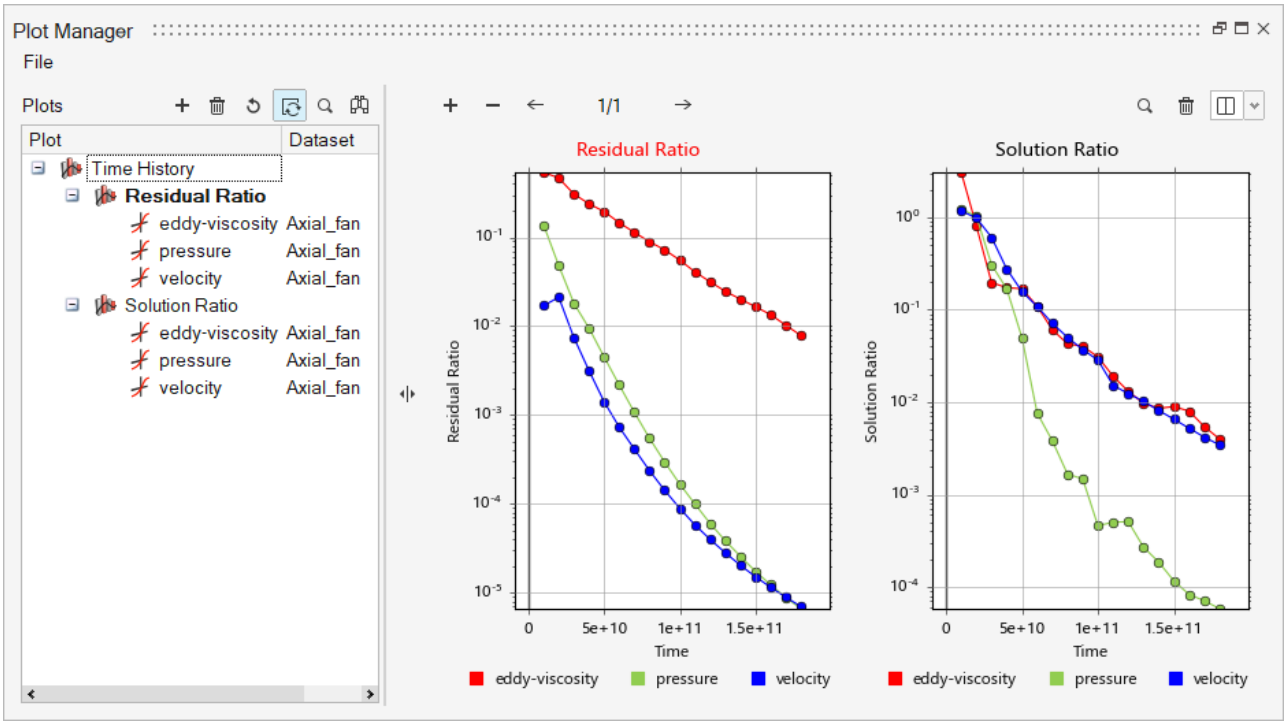


Figure 1031:

2. Click + to create a new plot.
3. In the Data Tree, expand **Integrated Surface Output** > **Pressure** then select **pressure**.
4. Under Parts, select **Fan_inlet** and **Fan_oultet** to display pressure at the inlet and outlet.

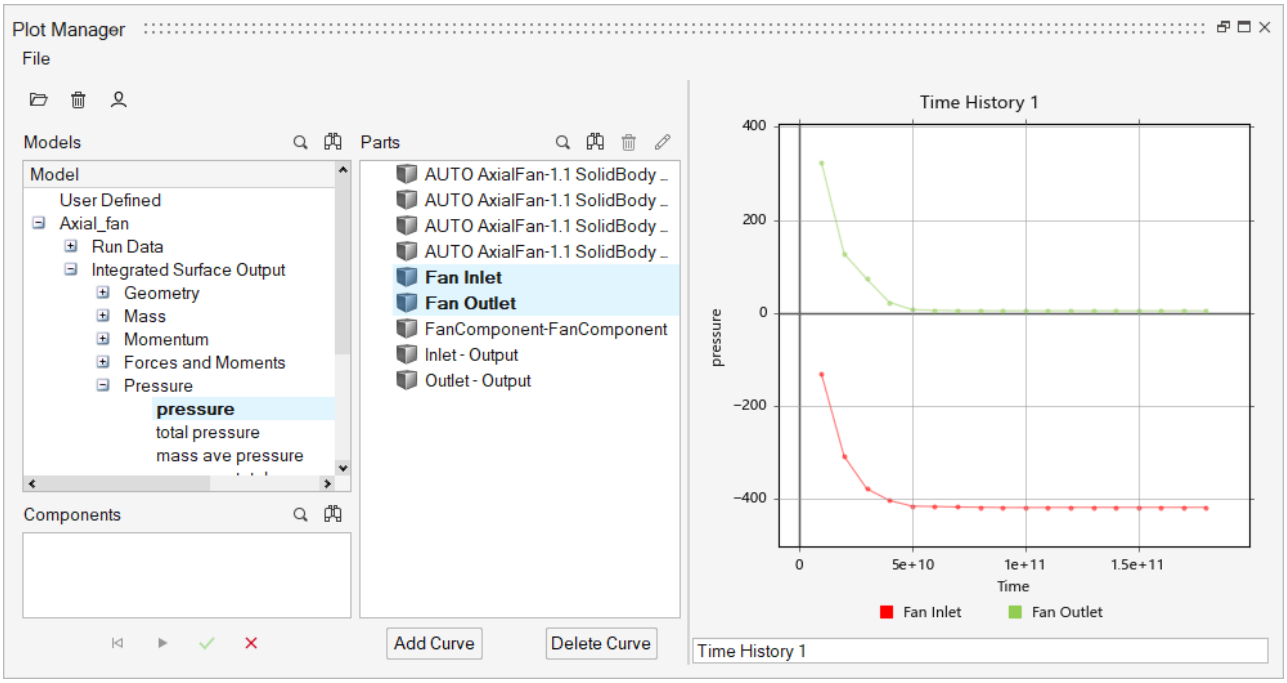


Figure 1032:

5. Right-click **User Defined** under the Model panel to create a user defined function.
6. In the Data Tree, expand **Integrated Surface Output** > **Pressure** then select **pressure**.
7. In the Name field, enter `dP`.
8. In the Value field, type `value=`.
9. Under Parts, choose **FAN outlet** then click to insert the field as part of the value, as shown below.

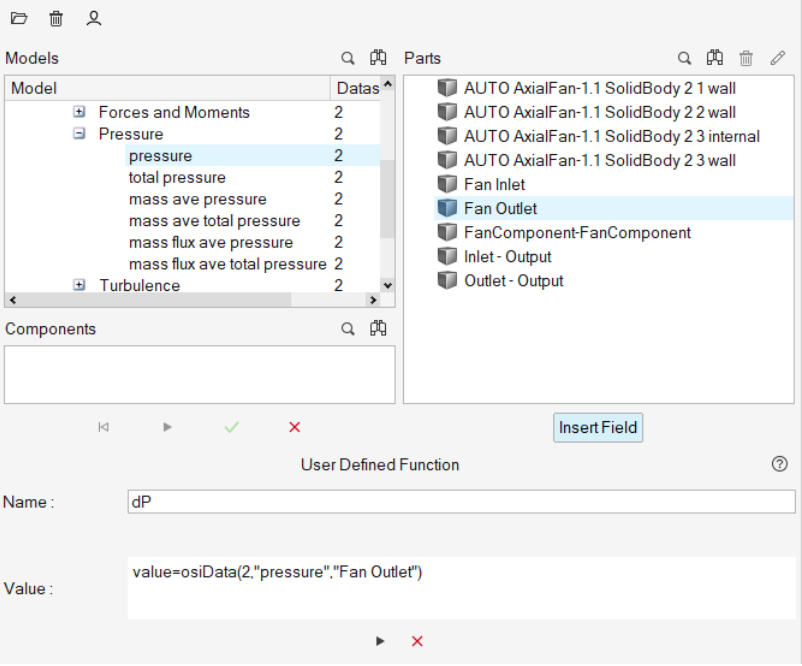


Figure 1033:

10. Type – at the end of the user defined function value.
11. Under Parts, choose **FAN inlet** then click to insert the field as part of the value.
12. Click ▸ to add the user defined function.

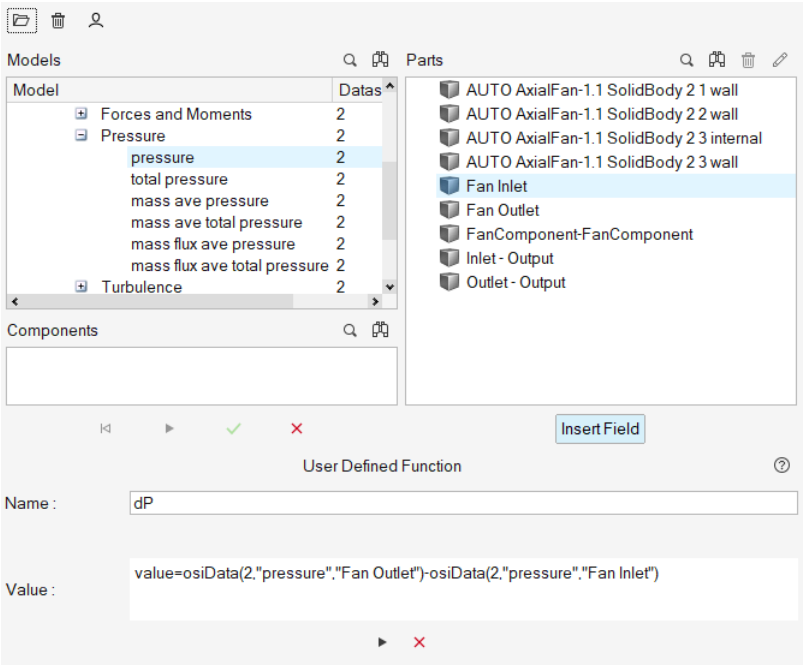


Figure 1034:

13. Click the **x axis** to switch to Time Steps.

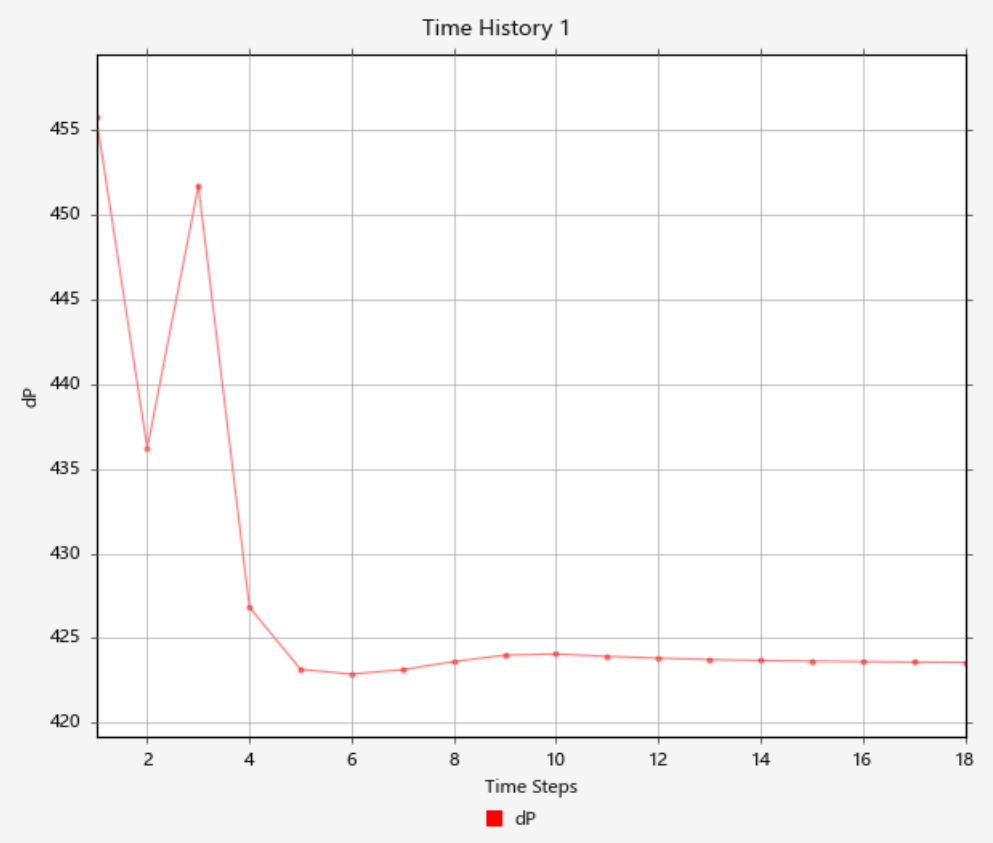


Figure 1035:

From the above figure, the pressure got stabilized at around the 6th iteration and remains constant with pressure difference between the FAN_inlet and FAN_outlet of 423.97 Pa for a given volume flow rate 0.322675 m^3/sec, which is very near compared to the reference pressure increase of 424.9 Pa.

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a coefficient fan component using HyperMesh CFD. You imported the geometry and then defined the simulation parameters, coefficient fan component, and flow boundary conditions. Once the solution was computed, you used the HM-CFD Plot tool to plot the pressure at fan inlet and fan outlet.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/aj6xim24kv

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5101 Modeling of a Fan Component Using the Fan Component - PQ Method

Perform this tutorial using the following user interface(s):

HyperMesh CFD

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This simulation provides instructions for running a steady state simulation of flow inside a pipe with an interior fan placed at the middle of the pipe. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5101_PQ_FanComponent.hm`

`fanPQcurve.csv`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be solved in this tutorial is shown schematically in the figure below. It consists of an interior fan which rotates at a speed of 377 rad/sec (~3600 RPM) and has a thickness of 0.06 m and a tip radius of 0.11 m. The volumetric flow rate at the inlet is 0.322675 m^3/sec (~1212.3 m^3/hr). The problem is simulated as a steady state run and the pressure rise across the fan region is computed.

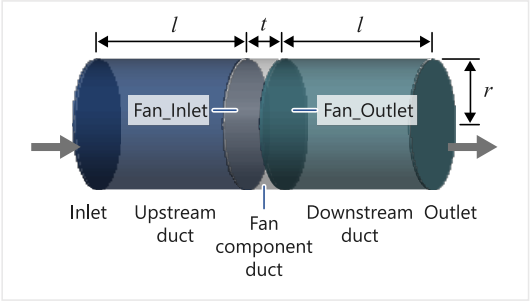


Figure 1036:

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1037:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5101_PQ_FanComponent.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `PQ_Fan` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `PQ_Fan` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1038:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

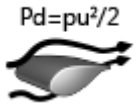


Figure 1039:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.

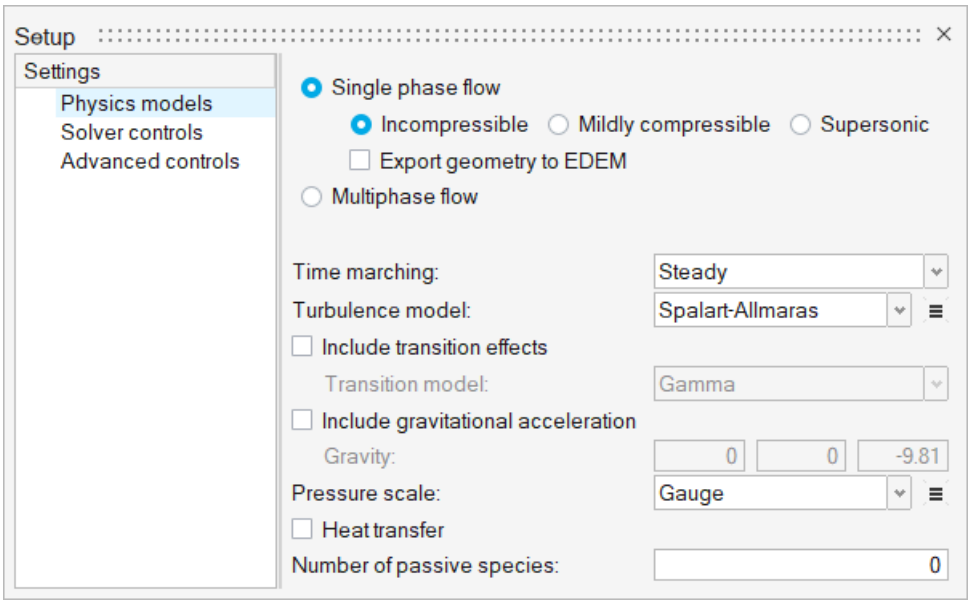


Figure 1040:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

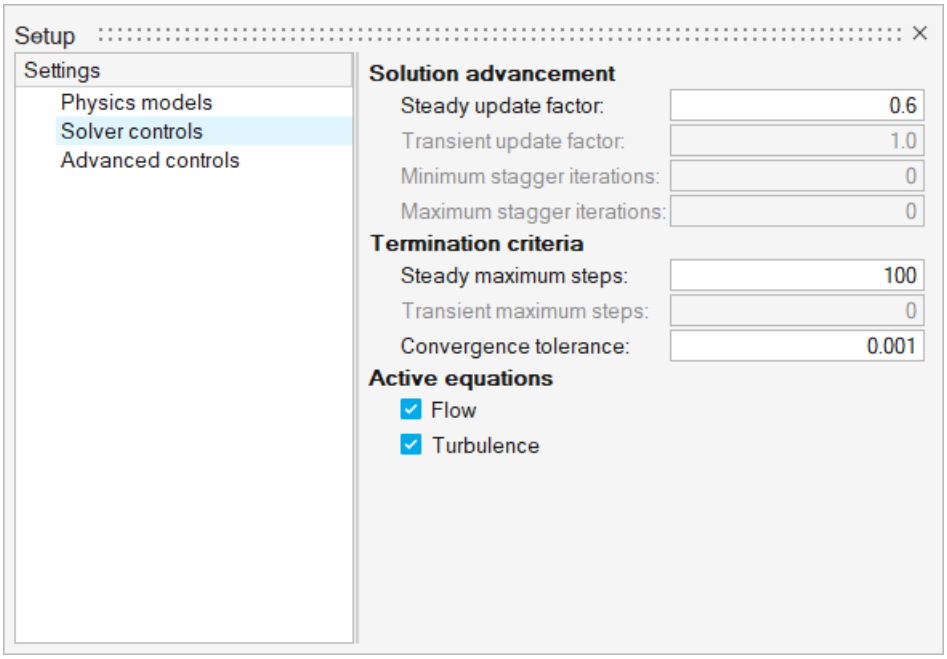


Figure 1041:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1042:

2. Verify that the Air material has been assigned to all three volumes.
3. Click ✓ on the guide bar to exit the tool.

Define the Fan Component

1. From the Flow ribbon, click the arrow next to the Domain tool set, then select **Fan Component**.

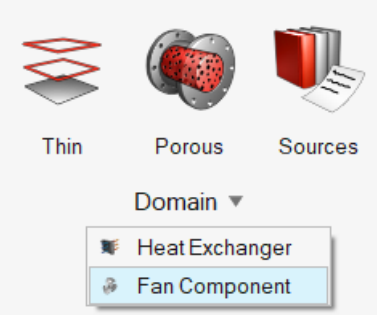


Figure 1043:

2. Select the middle solid as the fan component volume.

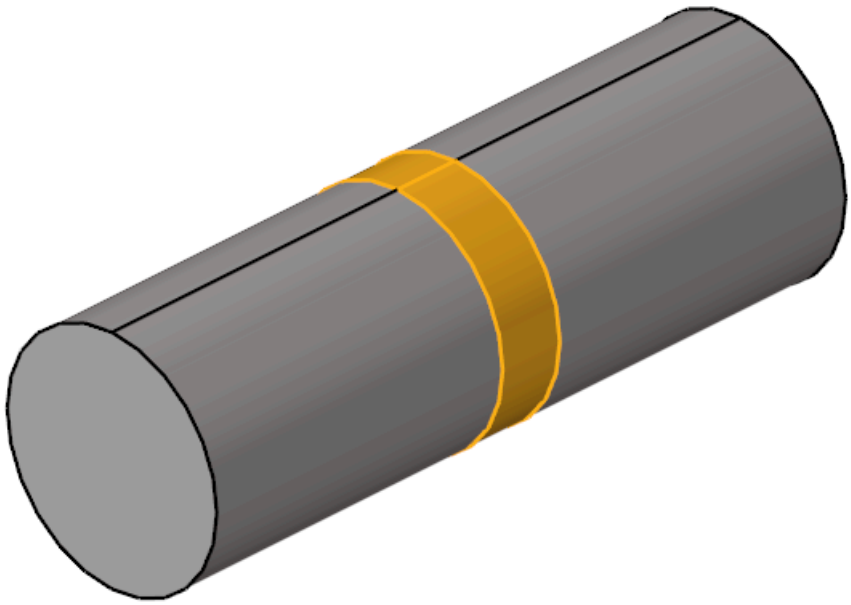


Figure 1044:

3. On the guide bar, click **Surfaces** then select the face shown below as the inlet of the fan component.

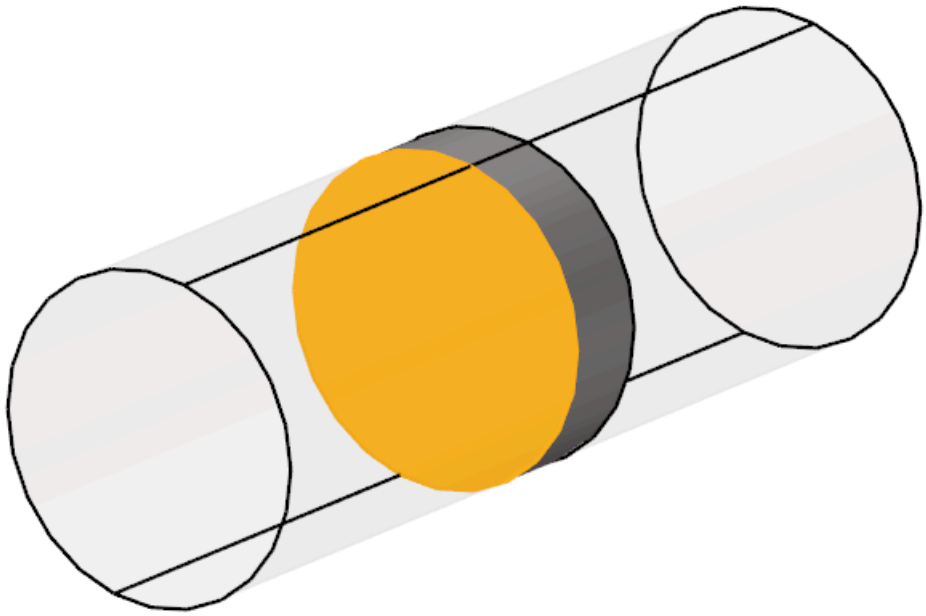


Figure 1045:

4. From the View Controls toolbar, change the geometry visualization mode from Shaded Geometry to **Transparent Geometry**.

   This allows you to view the axis direction vector in the next step.

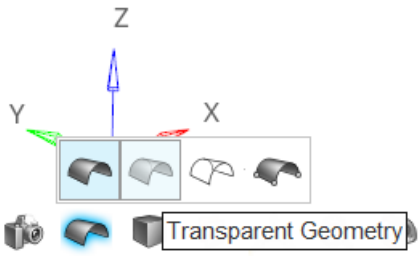


Figure 1046:

5. On the guide bar, click **Axis**.
   In the modeling window, you can see that the axis points in the -X direction.
6. Click +/- in the microdialog to flip the axis vector to the +X direction.

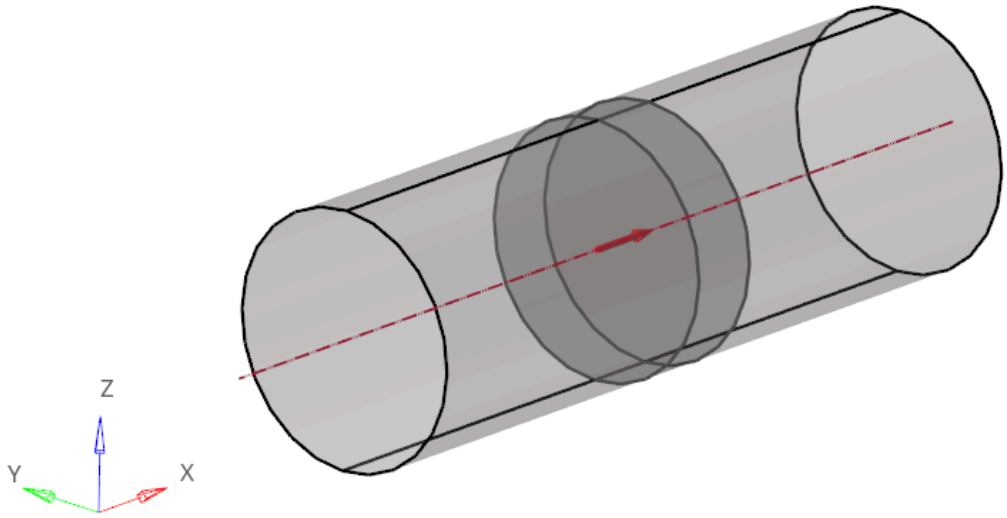


Figure 1047:

7. Enter `0.06` for Thickness (m).

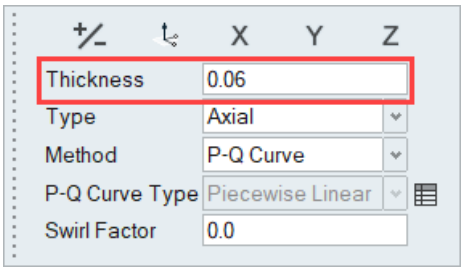


Figure 1048:

8. Click beside P-Q Curve Type to open the Profile Editor.

9. Click , browse to the location where you saved `fanPQcurve.csv`, and open it.

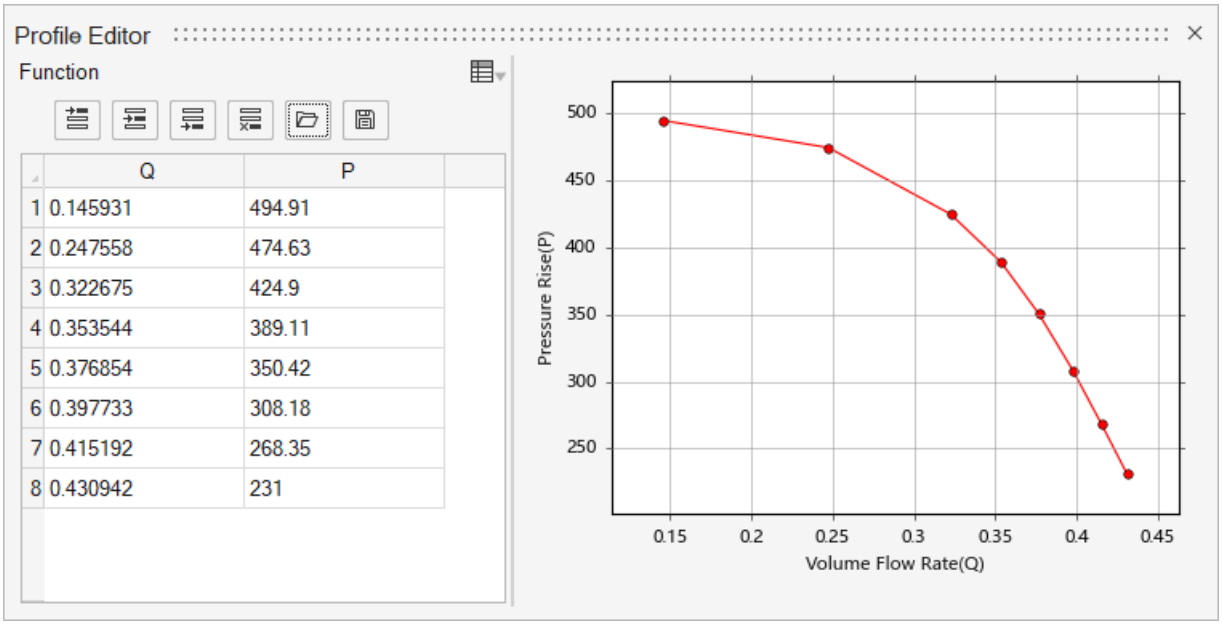


Figure 1049:

Note: Units for volume flow (Q) and Pressure Rise (P) are, respectively, m^3/sec and Pa.

10. On the guide bar, click ✓ to execute the command and exit the tool.
11. Save the model.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Volumetric Flow Rate** tool.

Figure 1050:

2. Click the inlet face highlighted in the figure below.

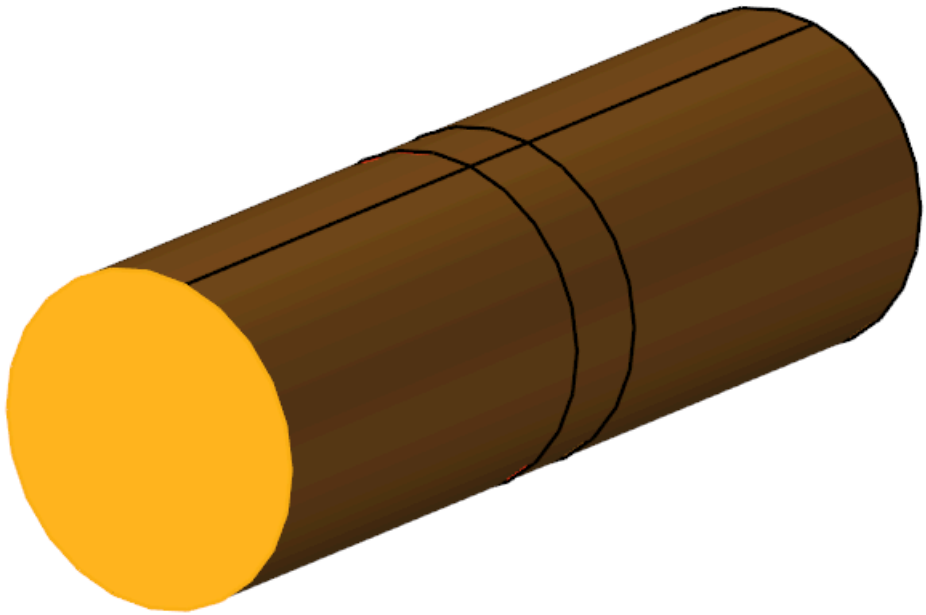


Figure 1051:

3. In the microdialog, enter `0.322675` for the volumetric flow rate (m^3/sec).

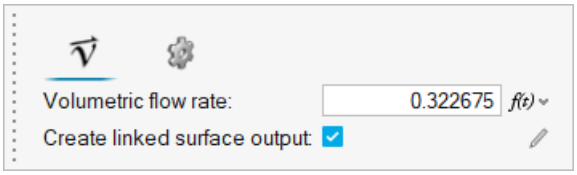


Figure 1052:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 1053:

6. Select the face highlighted in the figure below and then click ✓ on the guide bar.

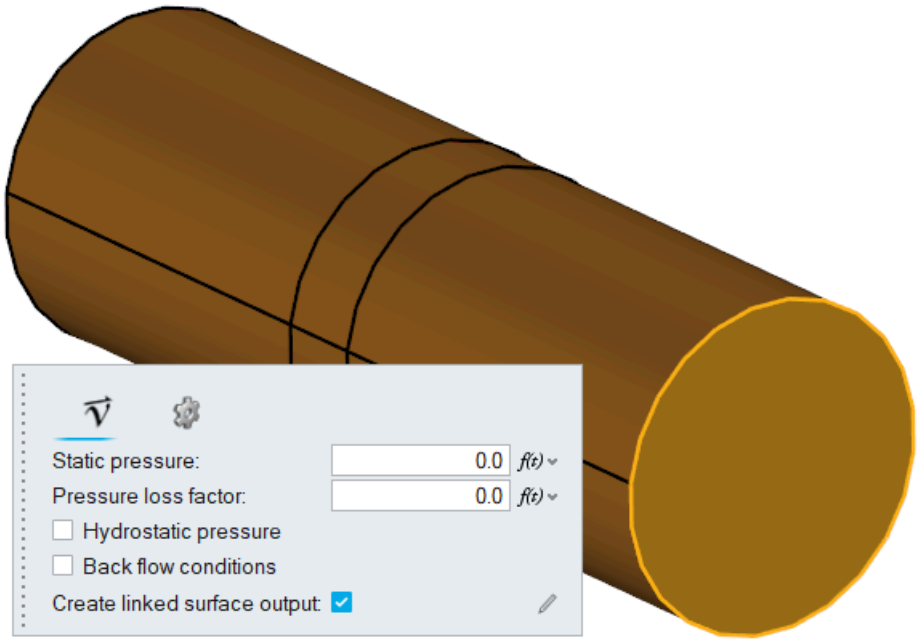


Figure 1054:

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

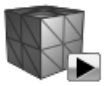


Figure 1055:

2. In the **Meshing Operations** dialog, set the Average element size to `0.01` and the Mesh growth rate to `1.1` (if not set already).

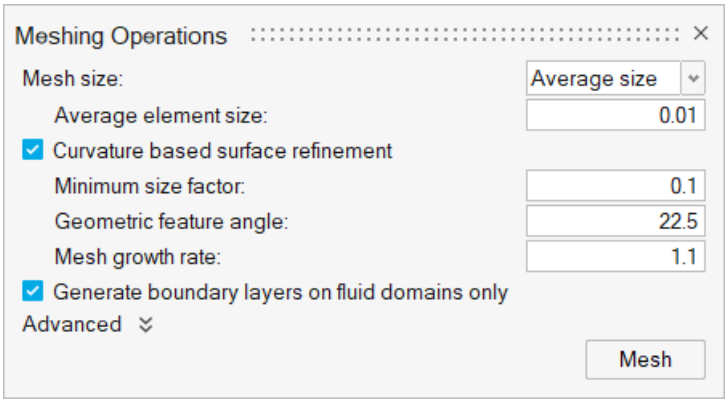


Figure 1056:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Surface Monitors and Run AcuSolve

1. Set the entity selector to **Solids**.

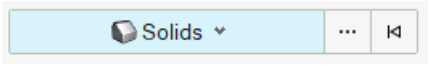


Figure 1057:

2. Select the fan component duct marked below to isolate it from front and aft ducts.

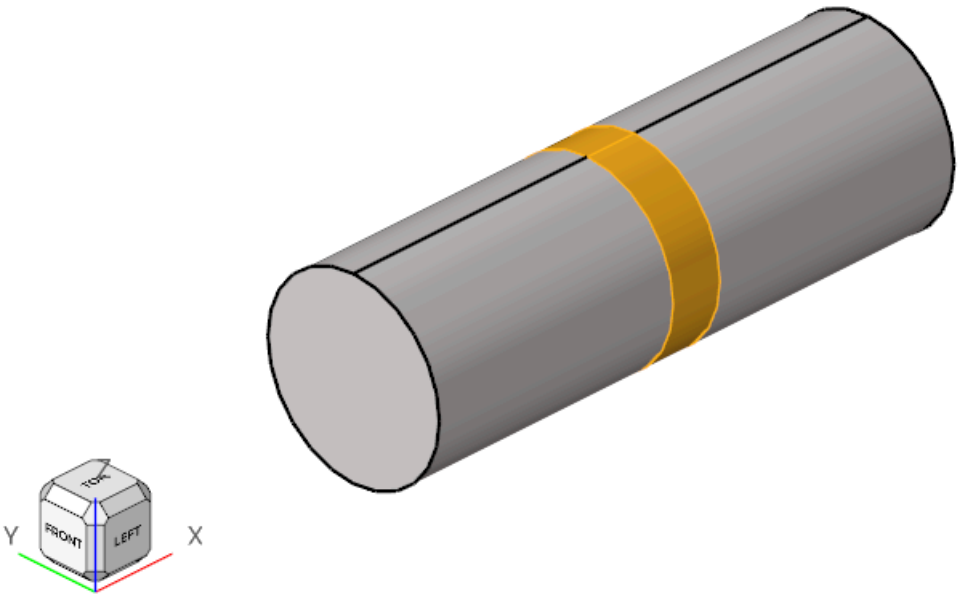


Figure 1058:

3. Right-click and select **Isolate** form the context menu or press **I**.
   The solid for the fan component should be displayed in the modeling window.

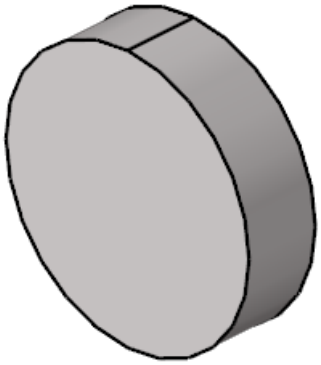


Figure 1059:

4. Form the Solution ribbon, click the **Surfaces** tool.

Figure 1060:

5. Select the front face of the fan and verify that the arrow is heading toward the fan component, as shown in figure below

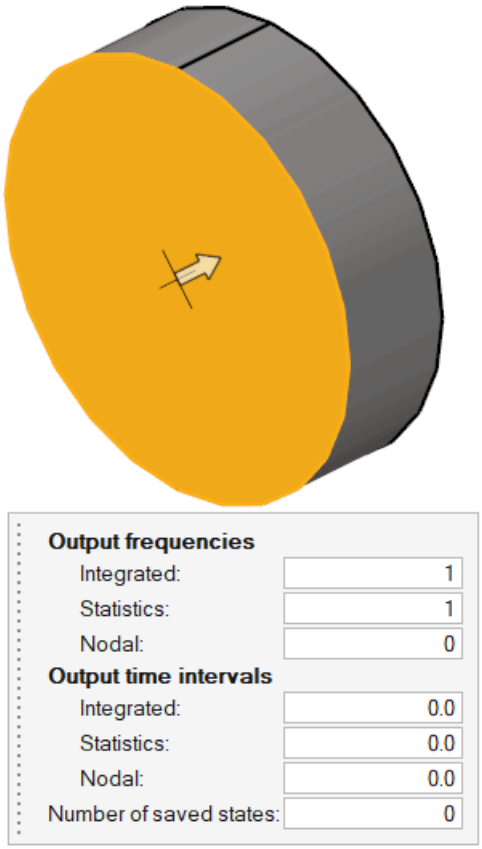


Figure 1061:

6. On the guide bar, click ▸ to execute the command and remain in the tool.
7. In the legend, rename surface_output to `FAN_inlet`.
8. Rotate the fan component and select the rear face.

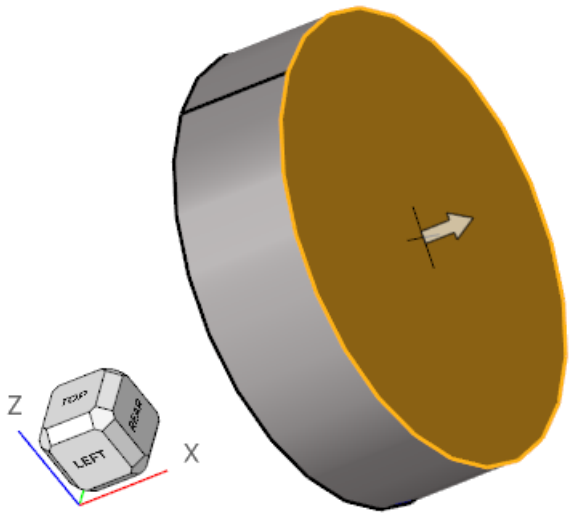


Figure 1062:

9. Click ▸, rename surface_output to `FAN_outlet` in the legend, then click ✓ to exit the tool.
10. From the Solution ribbon, click the **Run** tool.

Figure 1063:

The **Launch AcuSolve** dialog opens.

11. Set the Parallel processing option to **Intel MPI**.
12. Optional: Set the number of processors to **4** or **8** based on availability.
13. Leave the remaining options as default and click **Run** to launch AcuSolve.

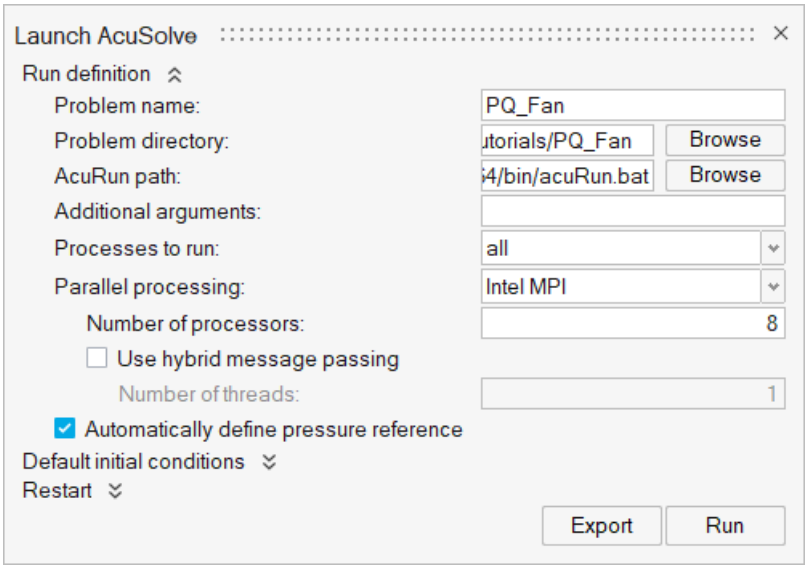


Figure 1064:

Post-Process with the Plot Tool

1. From the Solution ribbon, click the **Plot** tool.

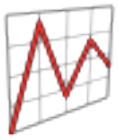


Figure 1065:

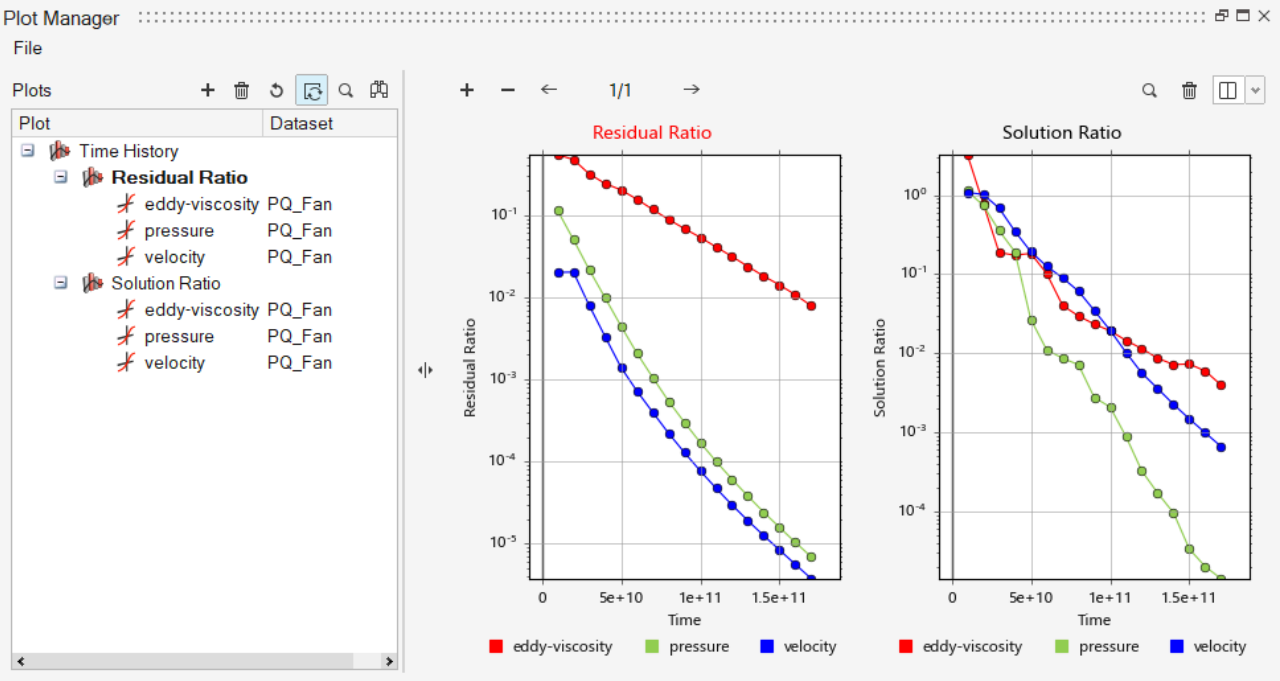


Figure 1066:

2. Click + to create a new plot.
3. In the Data Tree, expand **Integrated Surface Output** > **Pressure** then select **pressure**.
4. Under Parts, select **Fan_inlet** and **Fan_oultet** to display pressure at the inlet and outlet.

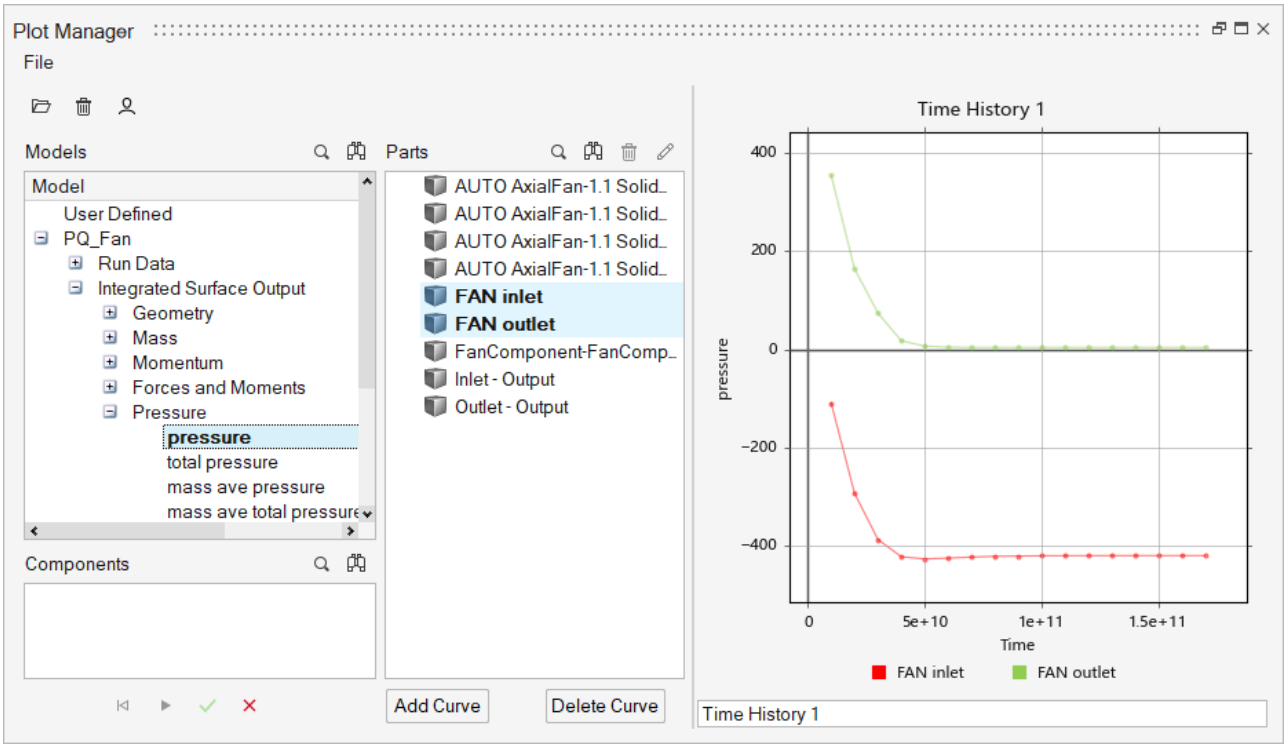


Figure 1067:

5. Right-click **User Defined** under the Model panel to create a user defined function.
6. In the Data Tree, expand **Integrated Surface Output** > **Pressure** then select **pressure**.
7. In the Name field, enter `dP`.
8. In the Value field, type `value=`.
9. Under Parts, choose **FAN outlet** then click **Insert Field** to insert the field as part of the value, as shown below.

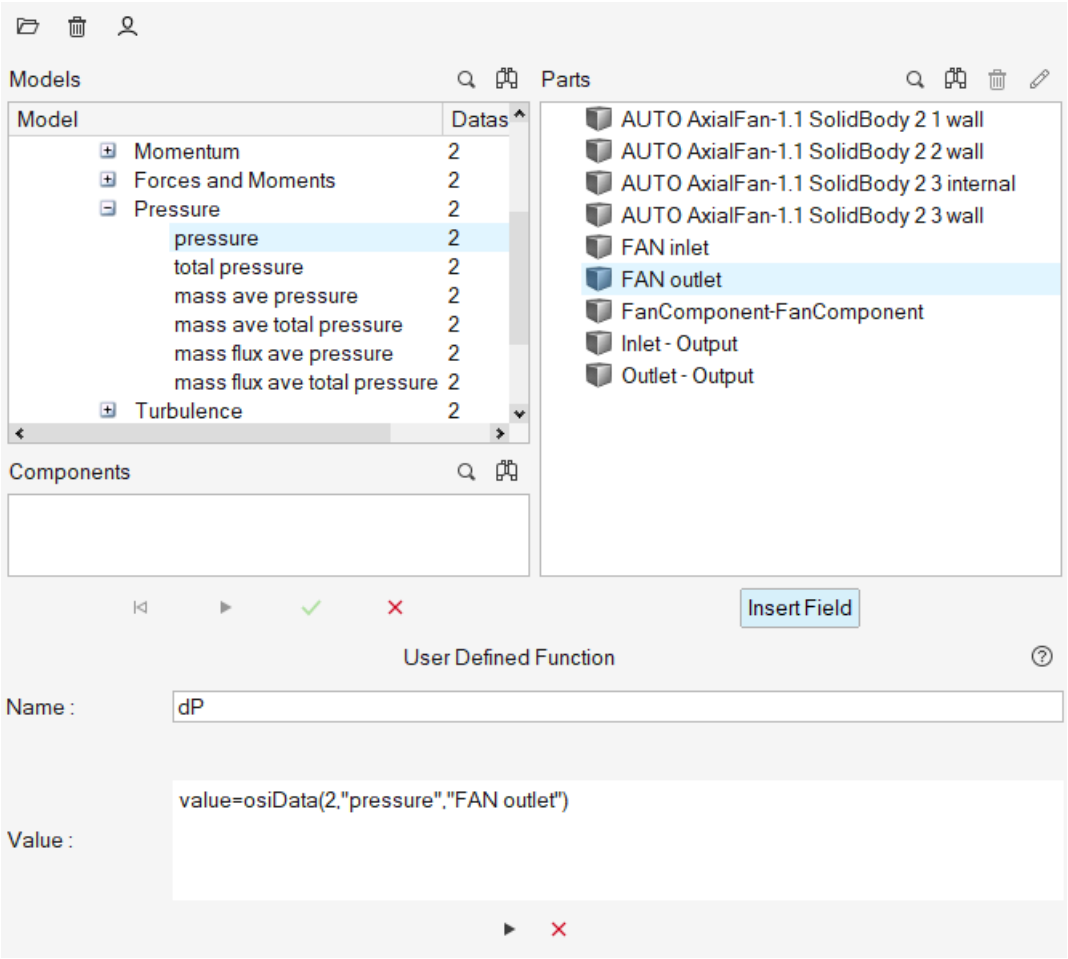


Figure 1068:

10. Type – at the end of the user defined function value.

11. Under Parts, choose **FAN inlet** then click **Insert Field** to insert the field as part of the value.

12. Click ▸ to add the user defined function.

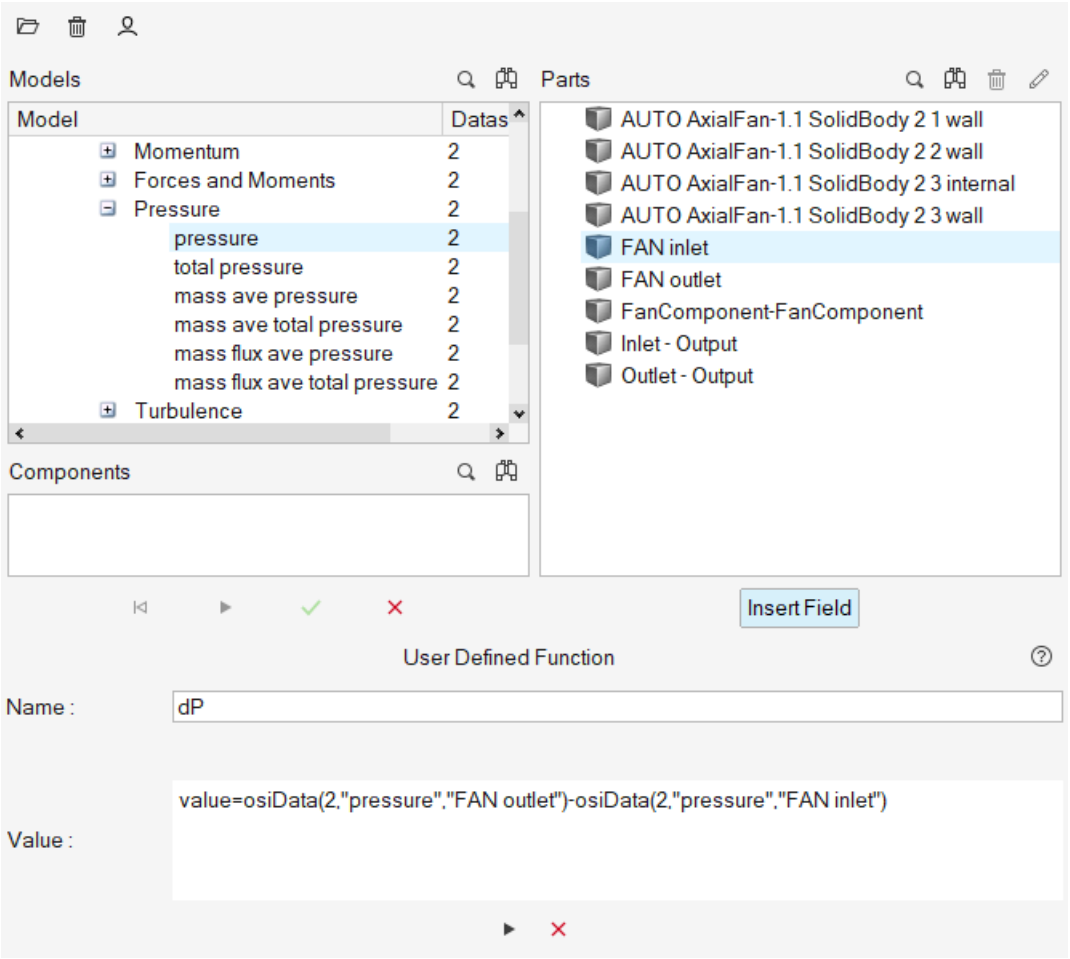


Figure 1069:

13. Click the **x axis** to switch to Time Steps.

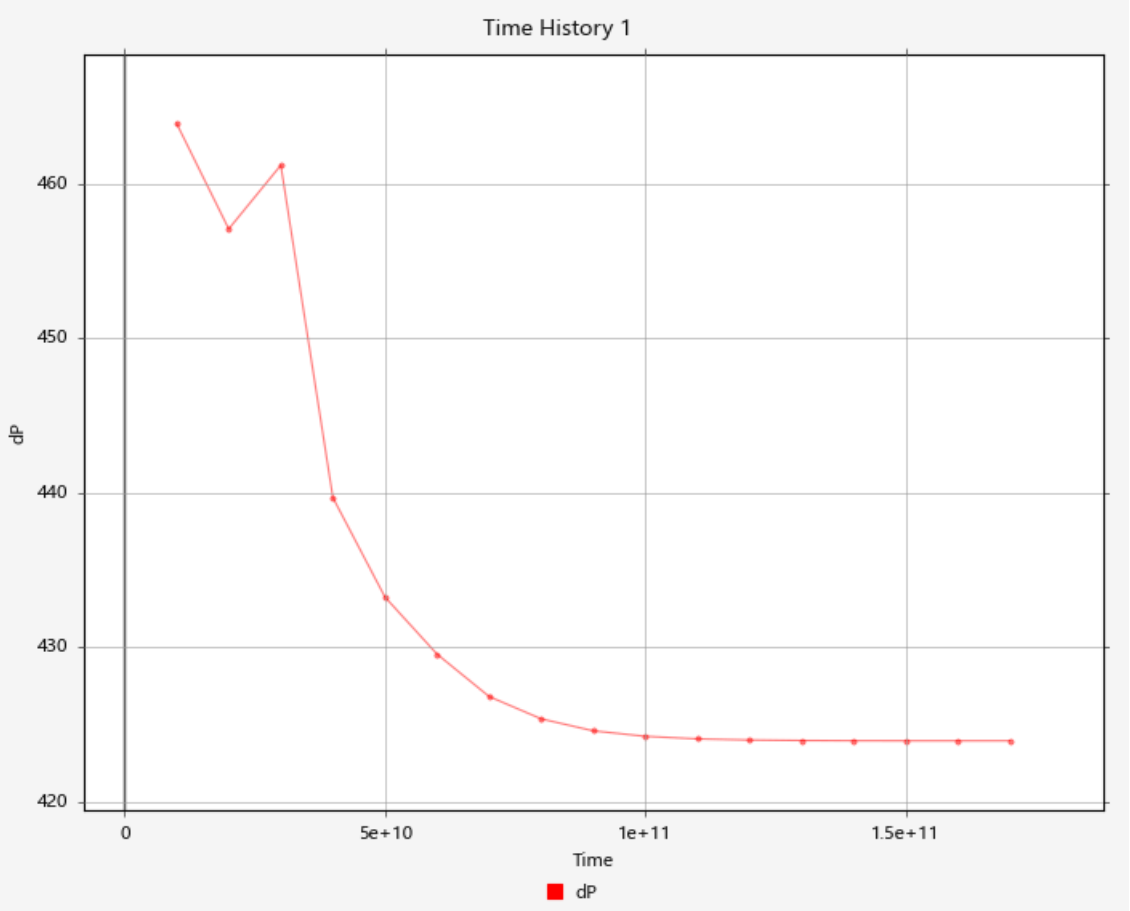


Figure 1070:

From the above figure, the pressure got stabilized at around the 6^{th} iteration and remains constant with pressure difference between the FAN_inlet and FAN_outlet of 423.97 Pa for a given volume flow rate 0.322675 m^3/sec, which is very near compared to the reference pressure increase of 424.9 Pa.

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a PQ fan component using HyperMesh CFD. You imported the geometry and then defined the simulation parameters, fan component, and flow boundary conditions. Once the solution was computed, you used the HM-CFD Plot tool to plot the pressure at fan inlet and fan outlet.

ACU-T: 5200 Rigid-Body Dynamics of a Check Valve

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

This tutorial provides the instructions for setting up, solving, and viewing results for a simulation of the opening of a pressure check valve. In this simulation, AcuSolve is used to compute the forces on the valve due to the time-varying inlet flow field and to compute the motion of the valve that results from these flow forces. This tutorial is designed to introduce you to a number of modeling concepts necessary to perform simulations of rigid-body dynamics.

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5200_pressureCheckValve.x_t`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1071. It consists of a cylindrical pipe containing water that flows past a check valve with a shutter attached to a virtual spring (not included in the geometry). The inlet pressure varies over time and the movement of the shutter will be determined as a function of the balance of the fluid forces against the reactive force of the spring. The problem is rotationally periodic at 30° increments about the longitudinal axis, and it is assumed that the resulting flow is also rotationally periodic, allowing for modeling with the use of a wedge-shaped section. For this tutorial, a 30° section of the geometry is modeled, as shown in the figure. Modeling a portion of an rotationally periodic geometry leads to reduced computation time while still providing an accurate solution.

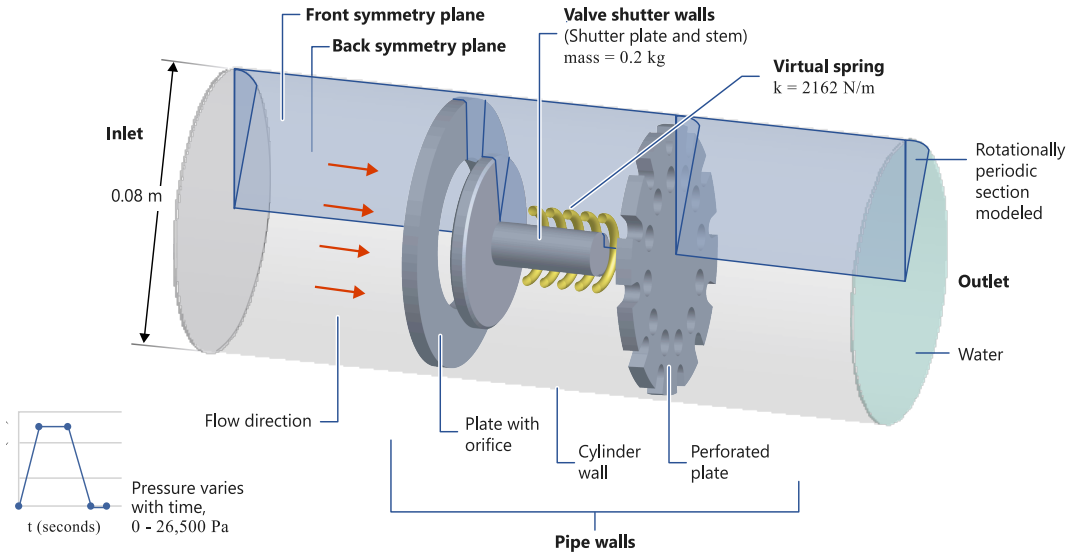


Figure 1071: Schematic of Check Valve with Spring-Loaded Shutter

The pipe has an inlet diameter of 0.08 m, and is 0.3 m long. The check-valve assembly is 0.085 m downstream of the inlet. It consists of a plate 0.005 m thick with a centered orifice 0.044 m in diameter and a shutter with an initial position 0.005 m from the opening, simulating a nearly closed condition. The shutter plate is 0.05 m in diameter and 0.005 m thick. The shutter plate is attached to a stem 0.03 m long and 0.01 m in diameter. The mass of the shutter and stem is 0.2 kg and its motion is affected by a virtual spring with a stiffness of 2162 N/m. The motion of the valve shutter is limited by a stop mounted on a perforated plate downstream of the shutter.

Note that AcuSolve's internal rigid-body-dynamics solver is not able to simulate contact. Therefore, this problem is formulated to avoid contact between the valve and the stop.

Modeling the geometry as a 30° section requires that the fluid model is set up to be consistent with the rigid-body model. Since only $^{1}/_{12}$ of the rigid body is modeled, the forces computed by AcuSolve that act on the valve shutter represent $^{1}/_{12}$ of the actual force on the device. Therefore, it is also necessary to account for this in the simulation. There are two methods that can be used to accomplish this:

1. Scale up the fluid forces calculated by AcuSolve by a factor of 12 to represent the full load on the device when the displacement of the body is computed.

   Using this approach, the full stiffness of the valve spring is used in the rigid-body solution, and the full mass of the valve is used.
2. Scale down the mass of the valve and the stiffness of the spring to by a factor of 12 to match the fraction of the valve geometry to be modeled.

   Using this approach, the loading passed to the rigid-body solver is not scaled.

This second approach is used in this tutorial; the scaled mass of 0.0167 kg and the scaled stiffness of 180.1667 N/m will be used .

The fluid in this problem is water, which has a density (ρ) of 1000 kg/m^3 and a molecular viscosity (μ) of 1 X 10^{-3} kg/m-sec.

At the start of the simulation the flow field is stationary. Flow is driven by the pressure at the inlet, which varies over time as a piecewise linear function shown in Figure 1072. As the pressure at the inlet rises, the flow will accelerate as the valve opens. The turbulence viscosity ratio is assumed to be 10.

The initial inlet pressure is 0 Pa. At 0.002 s, the pressure begins to ramp up, and reaches 26,500 Pa at 0.05 s. The pressure is held at 26,500 Pa and begins to ramp back down starting at 0.2 s, reaching the initial pressure at 0.25 s where it remains for the rest of the simulation.

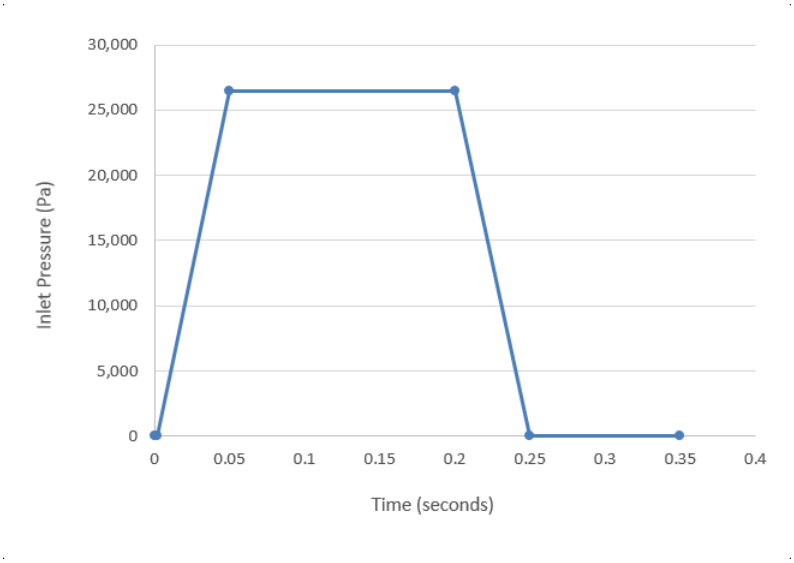


Figure 1072: Transient Pressure at the Inlet

Prior simulations of this geometry indicate that the average velocity at the inlet reaches a maximum of 0.9 m/s. At this velocity, the Reynolds number for the flow is 72,000. When the Reynolds number is above 4,000, it is generally accepted that flow should be modeled as turbulent.

Note that the initial conditions of the flow are actually laminar, however, the increase in flow velocity and flow around the valve shutter is expected to cause a rapid transition to turbulent conditions. Therefore, the simulation will be set up to model transient, turbulent flow. When performing a transient analysis, convergence is achieved at every time step based on the defined stagger criteria. Mesh motion will be modeled using arbitrary mesh movement (arbitrary Lagrangian-Eulerian mesh motion).

For this case, the transient behavior of interest occurs in the time it takes for the pressure to ramp up and ramp back down, which is given by the transient pressure profile. To allow time for the spring to recover, additional time will be simulated. For this tutorial, 0.1 s is added after the pressure drops back to initial conditions, for a total duration of 0.35 s.

Another critical decision in a transient simulation is choosing the time increment. The time increment is the change in time during a given time step of the simulation. It is important to choose a time increment that is short enough to capture the changes in flow properties of interest, but does not require unnecessary computation time.

The change in inlet pressure from initial conditions to maximum occurs over 0.048 s. A time increment of 0.002 s would allow for excellent resolution of the transient changes, without requiring excessive computational time.

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.
   - From the Home tools, Files tool group, click the **Save As** tool.

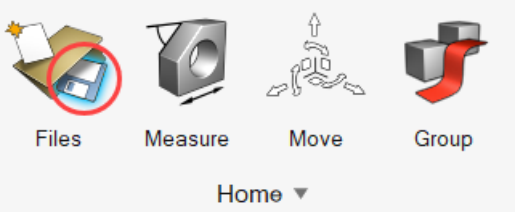


Figure 1073:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `CheckValve_RBD` as the name for the database then click **Save**.
   This will be your problem directory and all the files related to the simulation will be stored in this location.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T5200_pressureCheckValve.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

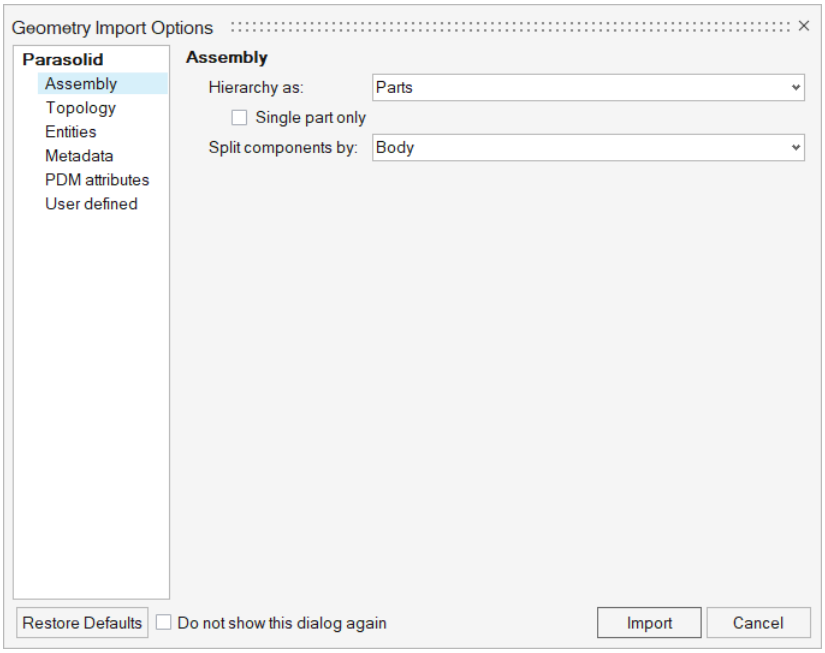


Figure 1074:

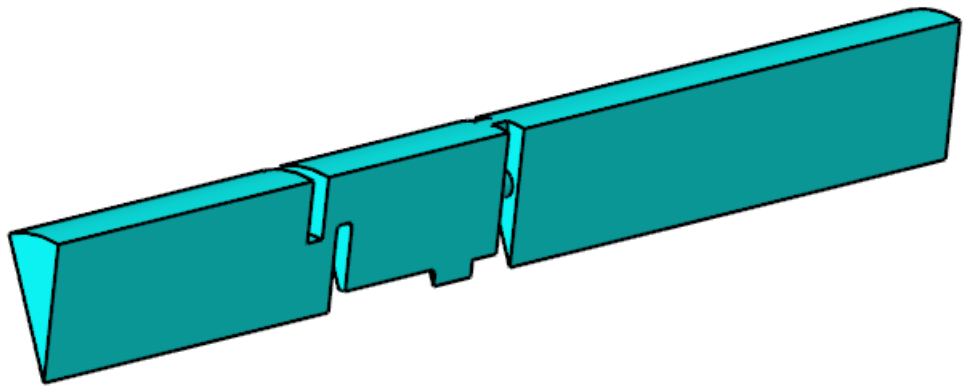


Figure 1075:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 1076:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 1077:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

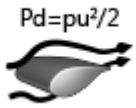


Figure 1078:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.002` and the Final time to `0.35`.
   c) Select **Spalart-Allmaras** as the Turbulence model.

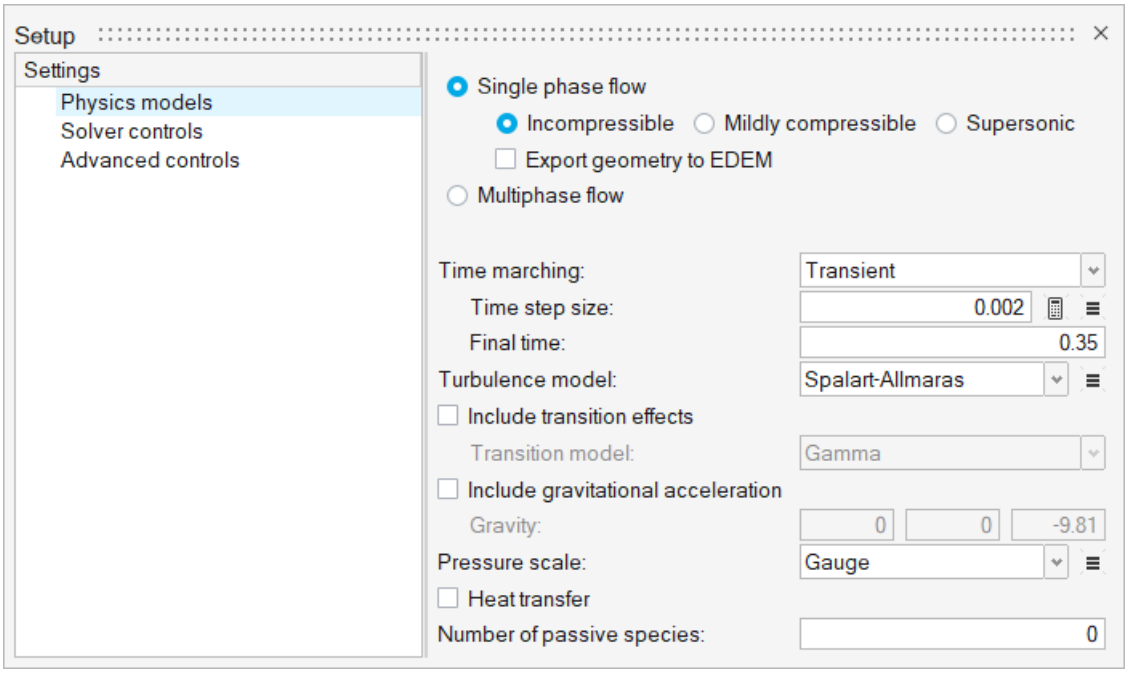


Figure 1079:

3. Click the **Solver controls** setting and set the Maximum stagger iterations to `3`.

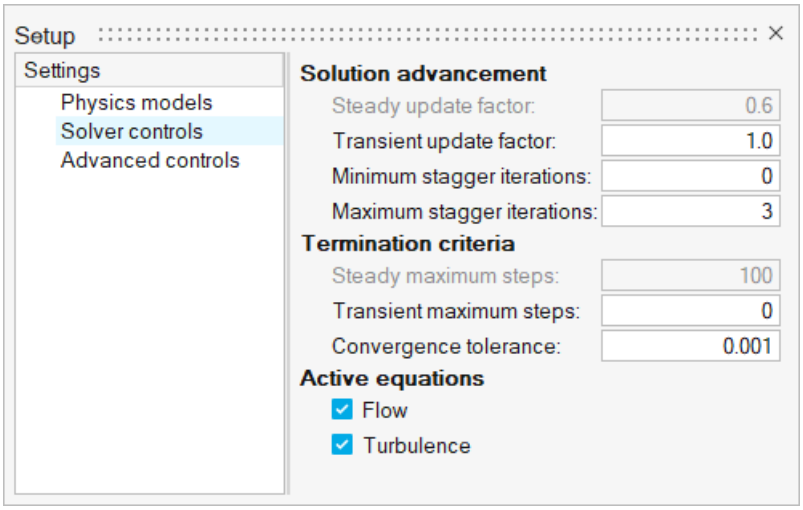


Figure 1080:

4. Close the dialog and save the model.

Create a Multiplier Function for the Inlet Pressure

1. From the Flow ribbon, click the arrow next to the Setup tool set, then select **Multipliers**.

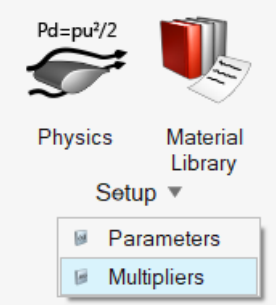


Figure 1081:

2. Click + in the **Multiplier Library** dialog.
3. In the multiplier creation dialog, change the name of the multiplier function to `Inlet Pressure` by clicking on the top-left corner.
4. Set the Type to **Piecewise Linear**.
5. Verify that the Variable is set to **Time** and Evaluation is set to **Time Step**.
6. Click four times to add four rows to the bottom of the table.
7. Enter the table values according to the image below.

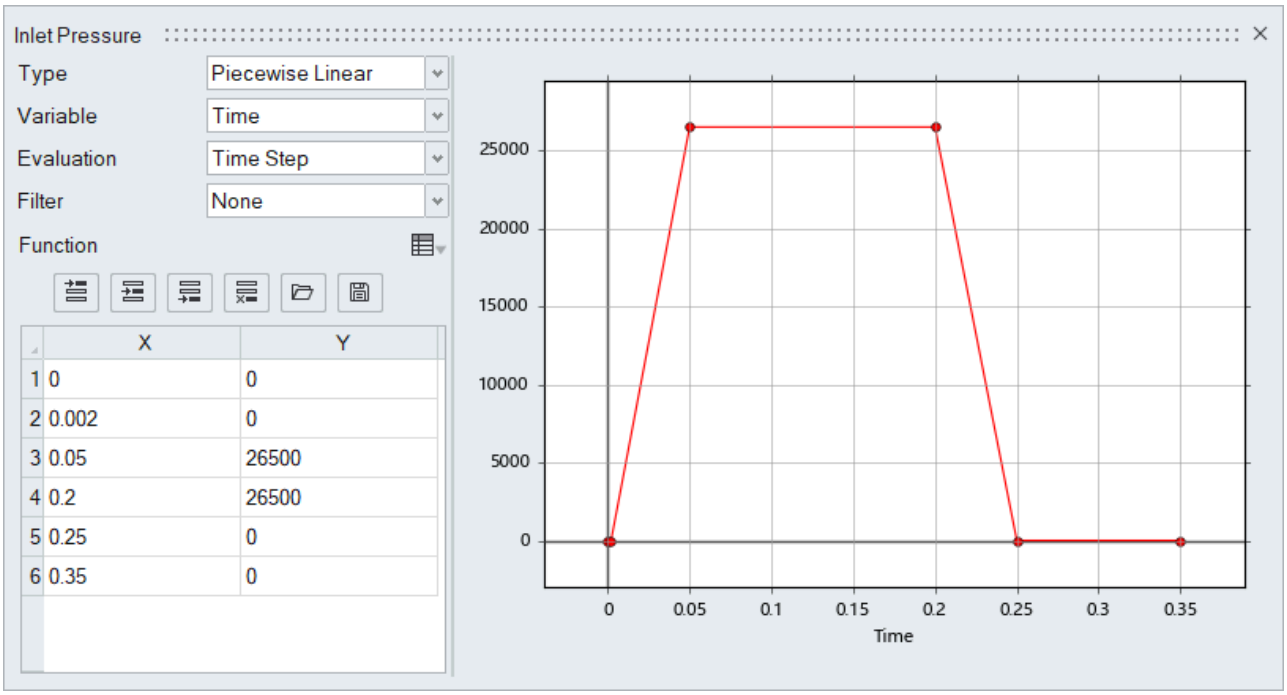


Figure 1082:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1083:

2. Click anywhere on the pipe geometry.
   The entire geometry is highlighted.

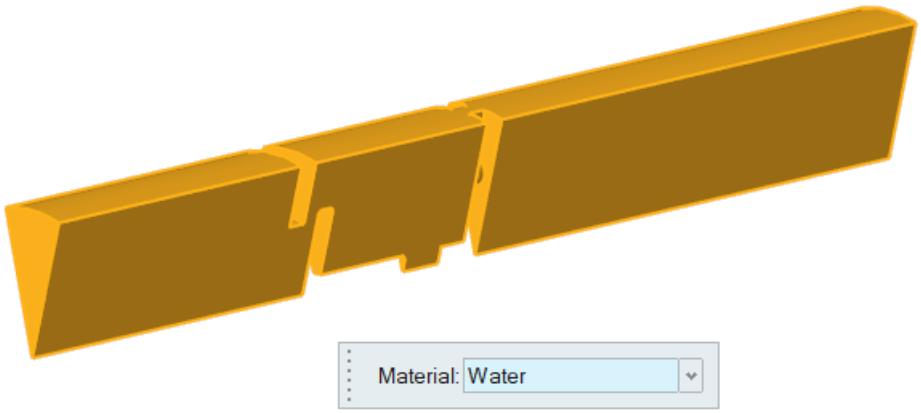


Figure 1084:

3. In the microdialog, select **Water** from the Material drop-down menu.
4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Save the model.

Define Flow Boundary Conditions

In this step, you will assign the boundary conditions for the inlet, outlet and the symmetry faces.

1. From the Flow ribbon, Pressure tool group, click the **Stagnation Pressure** tool.

Figure 1085:

2. In the modeling window, click on the inlet surface, as shown in Figure 1086.
3. In the microdialog, enter a value of `1` N/m^2 for the Stagnation pressure.
4. Click the Multiplier function drop-down and select the function **Inlet Pressure** from the list.

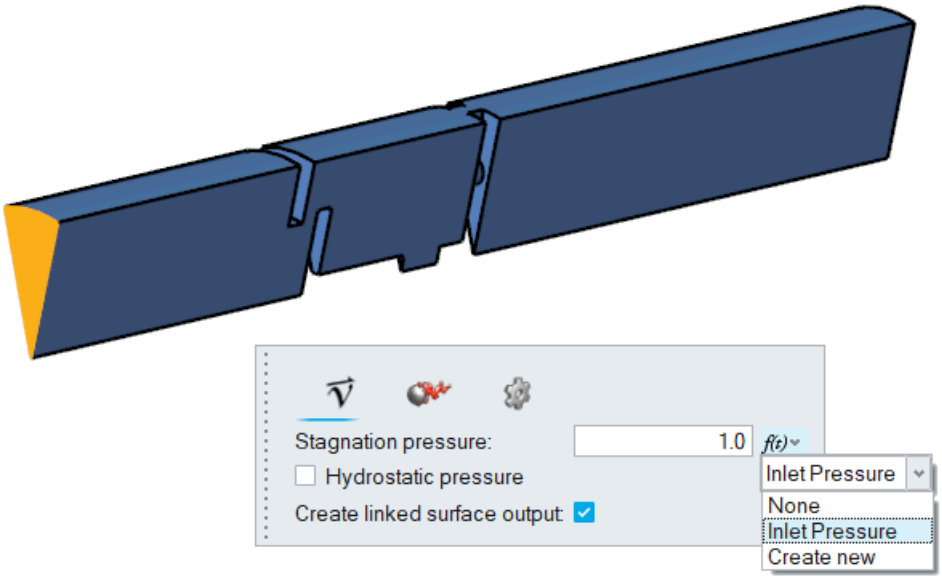


Figure 1086:

5. Click the Turbulence icon the microdialog, set the Turbulence input type to **Viscosity Ratio**, and the enter a value of `10` for the Turbulence viscosity ratio.

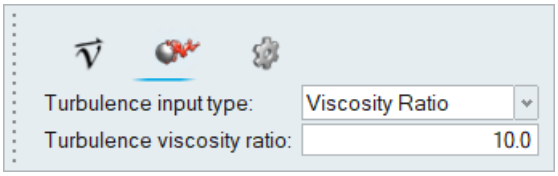


Figure 1087:

6. On the guide bar, click ▸ to execute the command and remain in the tool.
7. In the Boundaries legend, right-click on **Stagnation pressure** and rename it to `Inlet` then press **Enter**.
8. Click the **Outlet** tool.

Figure 1088:

9. Click the outlet surface shown in the figure below.

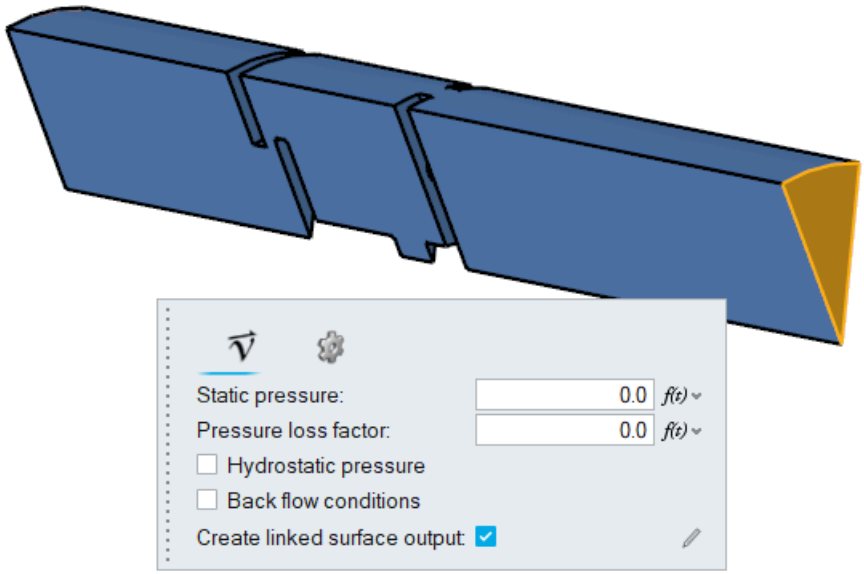


Figure 1089:

10. On the guide bar, click ✓ to execute the command and exit the tool.

11. Click the **Symmetry** tool.

Figure 1090:

12. Select the two surfaces shown in the figure below.

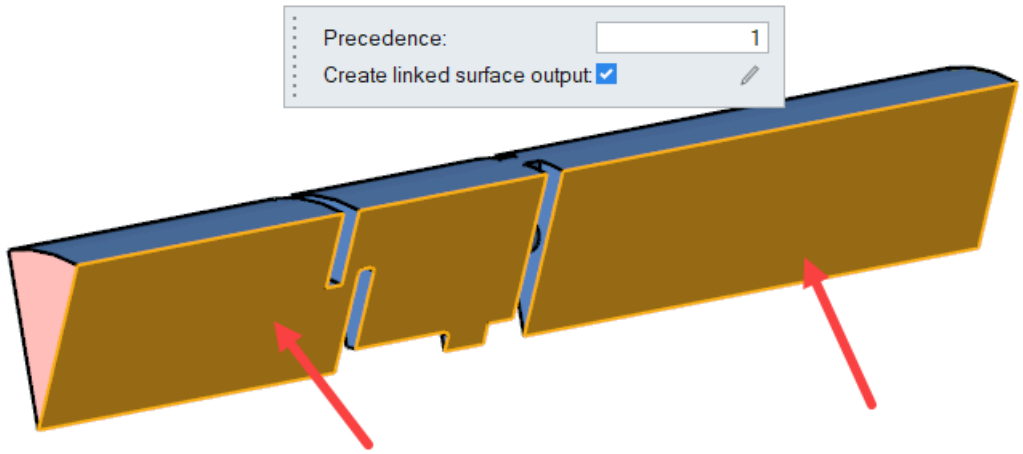


Figure 1091:

13. On the guide bar, click ▸ to execute the command and remain in the tool.
14. In the Boundaries legend, rename Symmetry to `Front_symmetry`.
15. Rotate the model and select the other two symmetry faces.

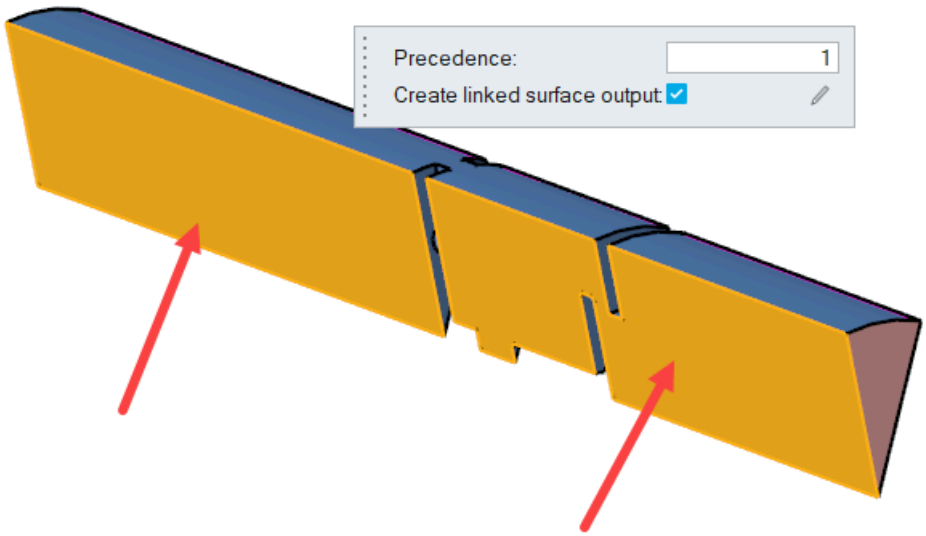


Figure 1092:

16. On the guide bar, click ▸ to execute the command and remain in the tool.
17. In the Boundaries legend, rename Symmetry to `Back_symmetry`.
18. Save the model.

Set Up Motion

In this step, you will activate the mesh motion and define the rigid body motion for the valve wall. Then, you will define the mesh displacement boundary conditions for the symmetry surfaces.

Define the Mesh Motion Type

1. From the Motion ribbon, click the **Settings** tool.

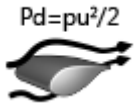


Figure 1093:

2. In the **Setup** dialog, change the Mesh motion to **Arbitrary**.

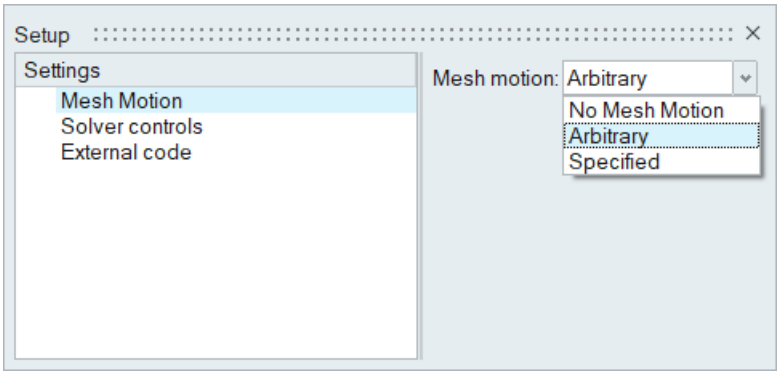


Figure 1094:

Define Rigid Body Motion

In this step, you will define the rigid body mesh motion on the valve walls.

1. From the Motion ribbon, click the **Rigid Body** tool.

Figure 1095:

2. On the guide bar, activate the **Surfaces** selector by clicking on it.

3. For convenience, hide the Front_symmetry and Back_symmetry faces in the modeling window by selecting the four surface and then pressing **H** on the keyboard or by right-click and selecting **Hide** from the context menu.

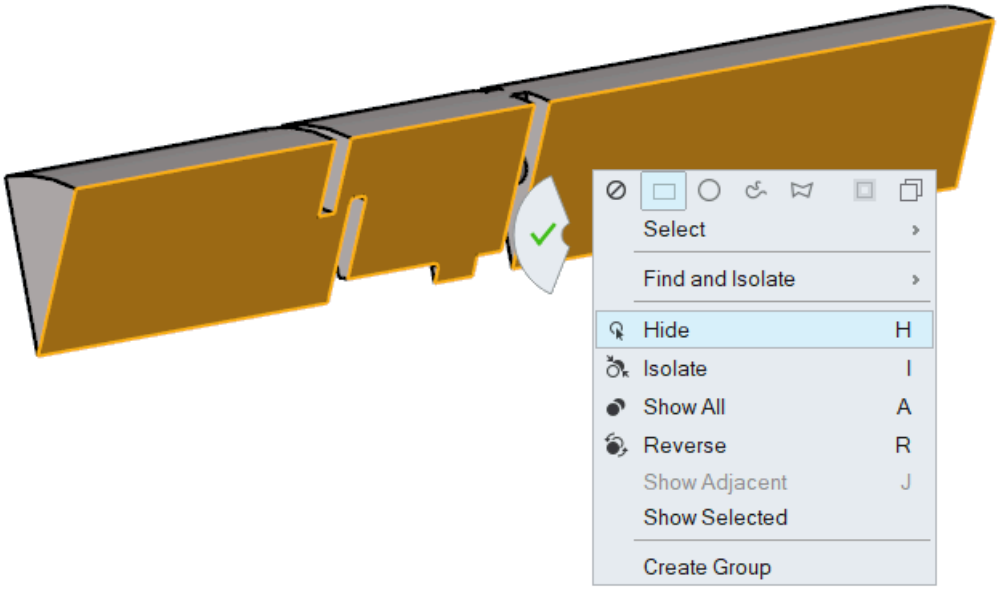


Figure 1096:

4. Select the valve wall surfaces, shown in the figure below, by using box selection (hold and drag the left mouse button).
The number of surfaces selected should be 8, which will show on the guide bar.

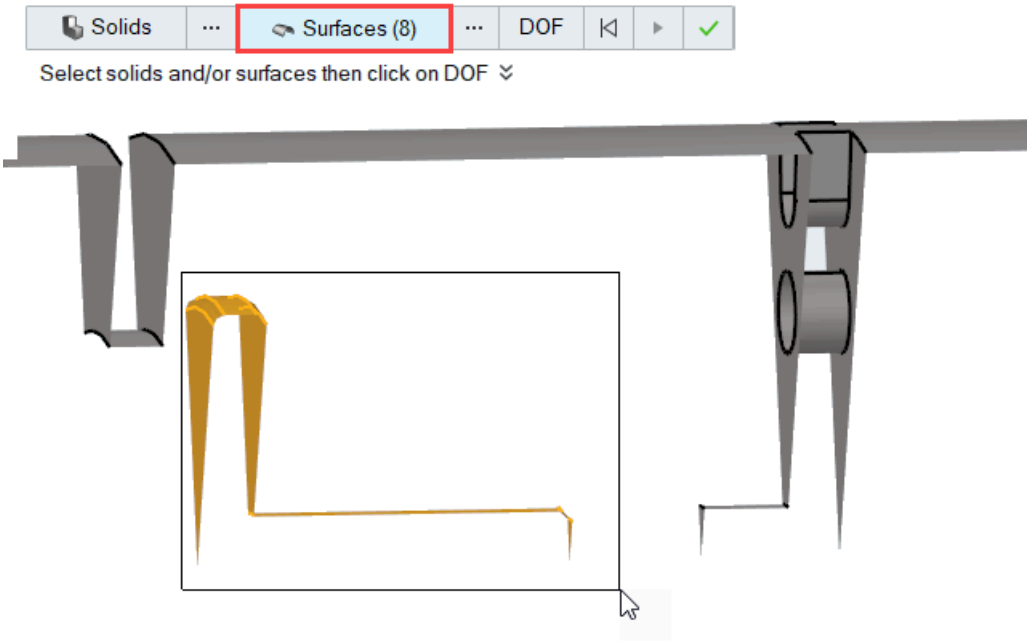


Figure 1097:

5. On the guide bar, click on **DOF** then click anywhere on the valve wall surfaces in the modeling window.
6. In the microdialog that appears, enter the following values:
   - Mass: `0.0167` kg
   - Center: `(0, 0, 0)`

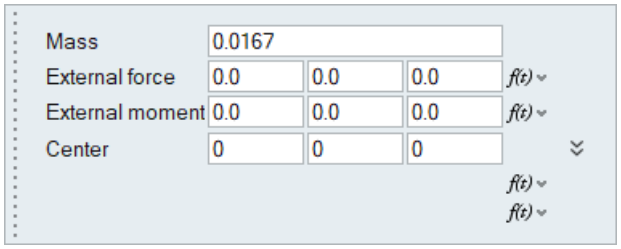


Figure 1098:

7. Click on the **X-displacement** arrow in the modeling window.
8. In the microdialog, enter a value of `180.1667` kg/sec^2 for Kxx, the stiffness of the spring.

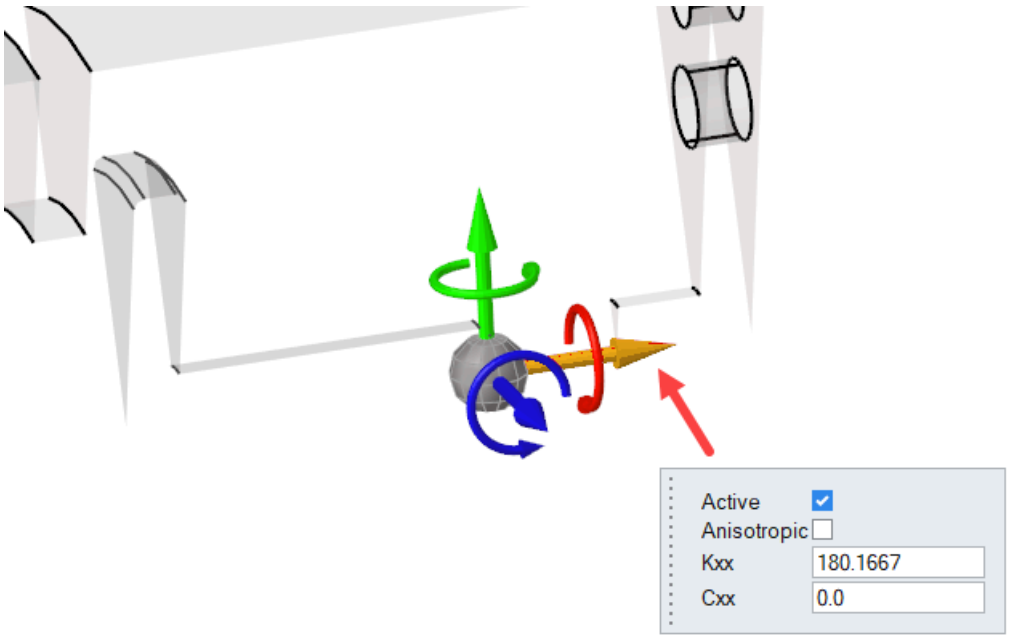


Figure 1099:

9. Click on the **X-rotation** arrow and de-activate it.

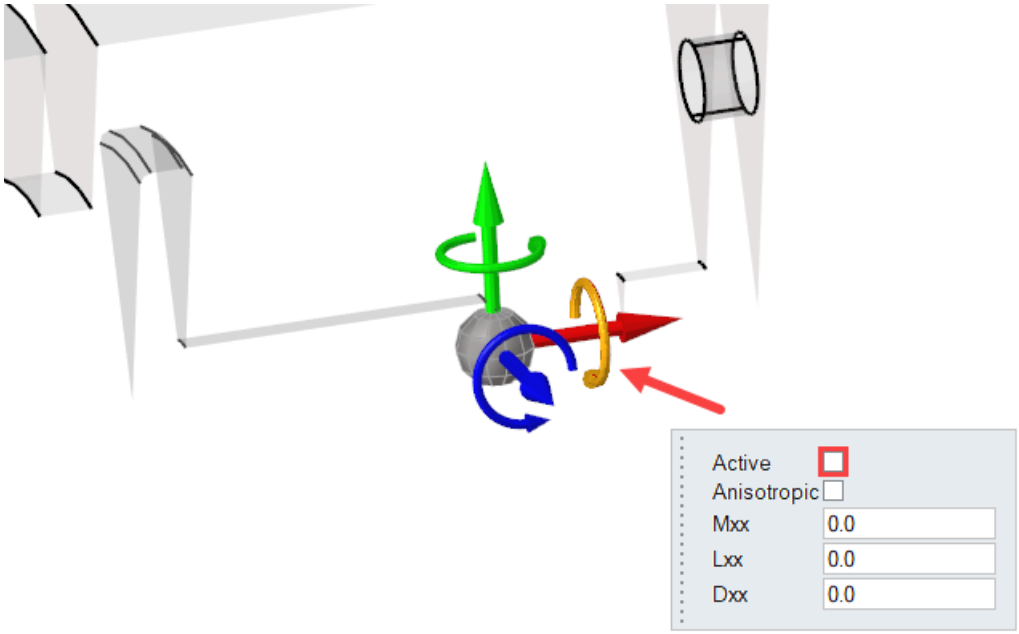


Figure 1100:

10. Similarly, click on the remaining four arrows (Y-displacement, Y-rotation, Z-displacement, and Z-rotation) and de-activate them.
    This will ensure that the valve body translates only in the x-direction.
11. On the guide bar, click ✓ to execute the command and exit the tool.
12. Turn on the display of all the surfaces and save the model.

Define the Mesh Displacement Boundary Conditions

1. From the Motion ribbon, click the **Planar Slip** tool.

Figure 1101:

2. Select the two front symmetry surfaces shown in Figure 1102.
3. In the Mesh Motion legend, rename Planar Slip to `Front_symmetry` by double-clicking on the name.

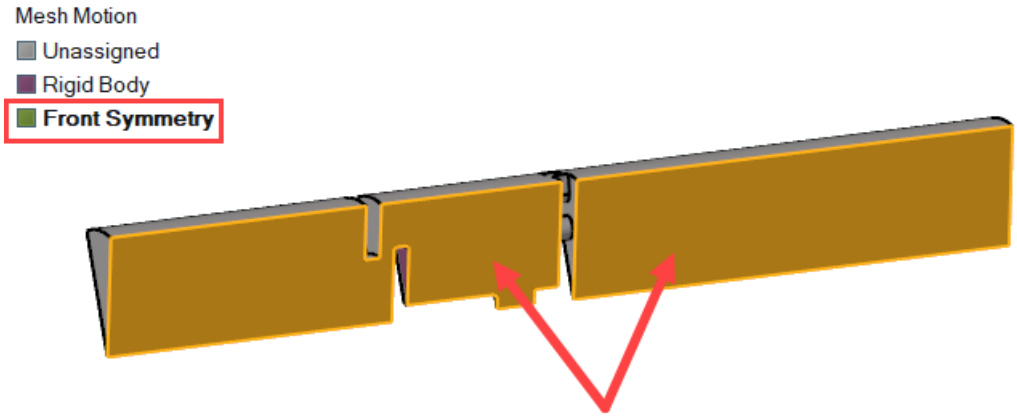


Figure 1102:

4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Rotate the model and select the two back symmetry faces.
6. In the Mesh Motion legend, rename Planar Slip to `Back_symmetry`.
7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Save the model.

Generate the Mesh

In this step, you will define the mesh controls and then generate the mesh.

Define the Zone Mesh Controls

The zone mesh controls allow you to define a local volume mesh size for specific regions using standard shaped zones like cylindrical, spherical, and cuboidal zones.

1. From the Mesh ribbon, Zones tool group, click the **Cylinder** tool.

Figure 1103:

2. In the modeling window, hover the mouse around the point shown in the figure below. When the preview cylinder zone is parallel to the axis of the pipe, click on the model near the point shown below.

   This point will be the center of the front face of the cylinder. In the next few steps you will edit the co-ordinates of this point manually.

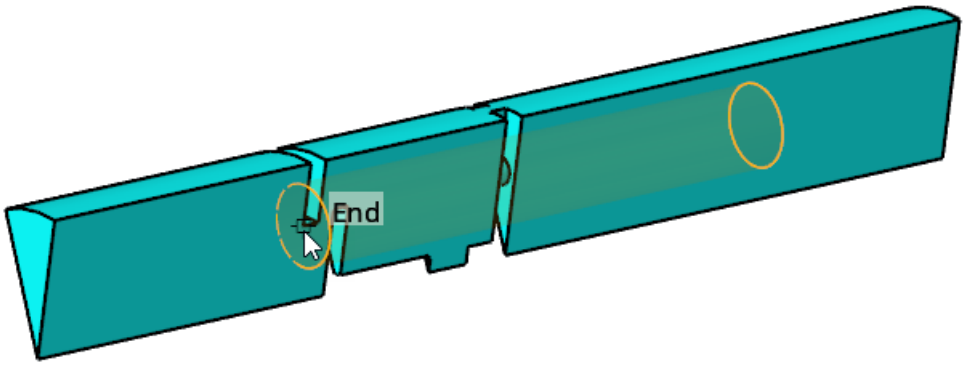


Figure 1104:

3. Move the mouse cursor away from the center and then click again.

A preview of the zone should be displayed on the screen along with the manipulator. The manipulator allows you to modify the location and orientation of the zone.

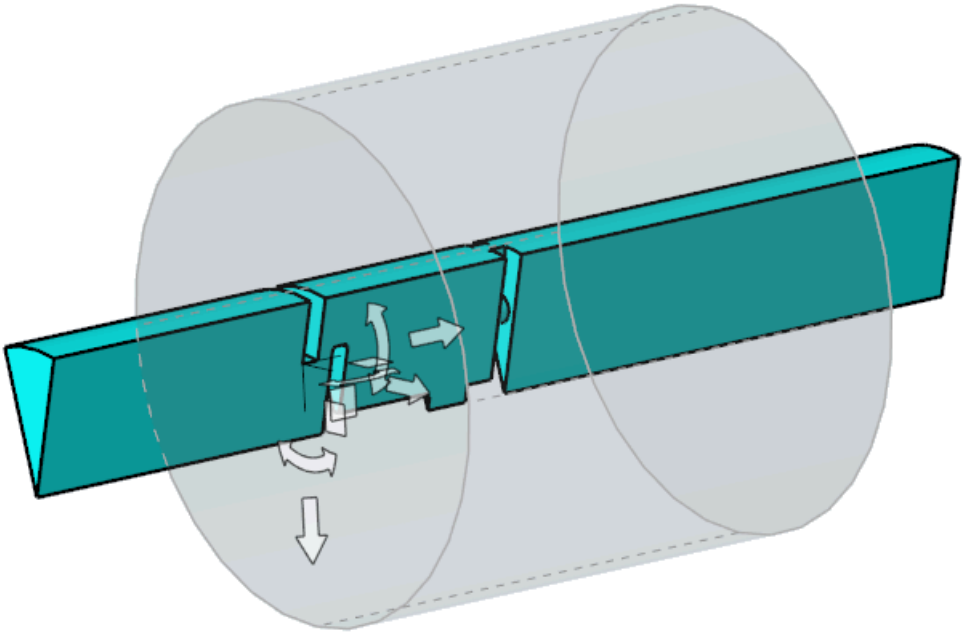


Figure 1105:

4. Click the center of the manipulator and enter the following coordinates for the center (`-0.06, 0.02, 0`)

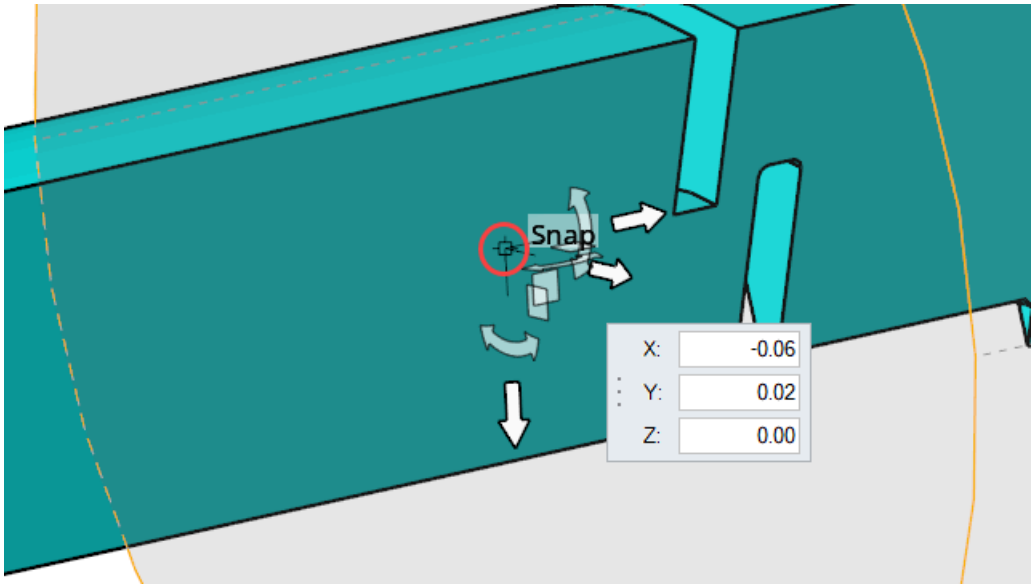


Figure 1106:

5. Double-click on the cylindrical surface of the zone and enter `0.022` m as the radius of the cylinder.

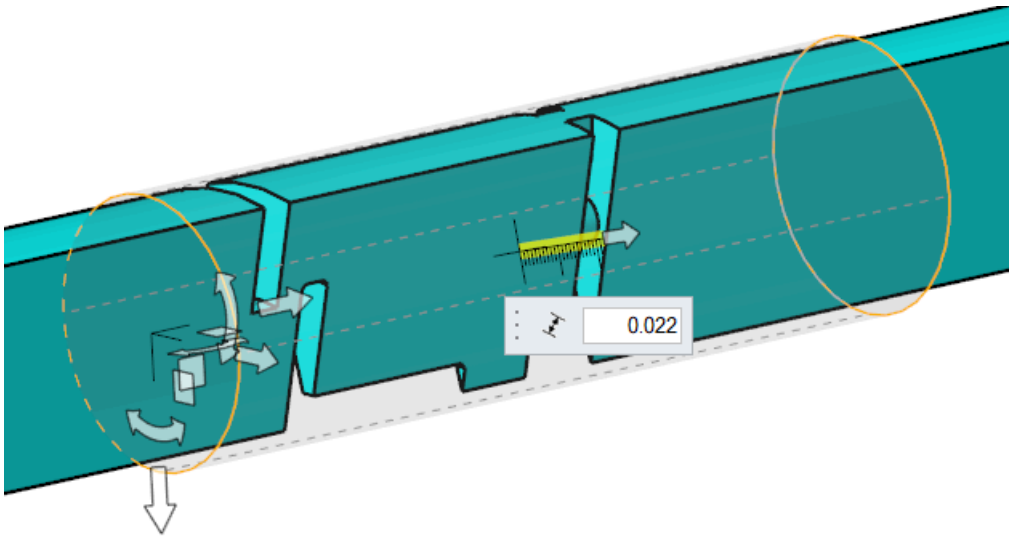


Figure 1107:

6. Double-click on the base of the cylinder and enter `0.1` m as the height of the cylinder.

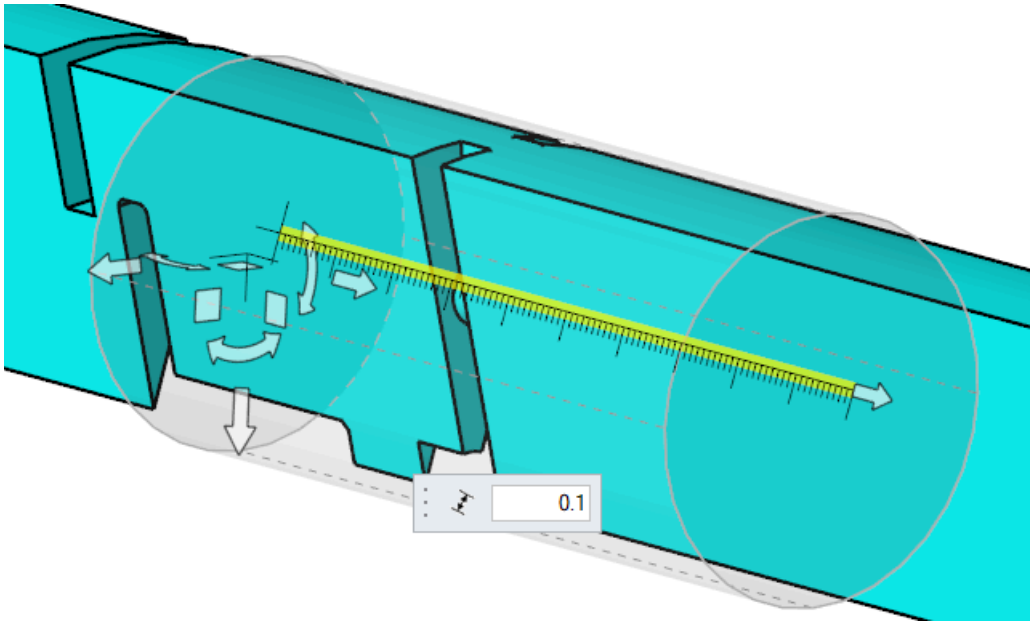


Figure 1108:

7. Click anywhere in the empty space in the modeling window.
8. Enter a value of `0.0015` m for the mesh size in the microdialog.
9. In the Refinement zones legend, rename the zone to `Valve region mesh refinement`.

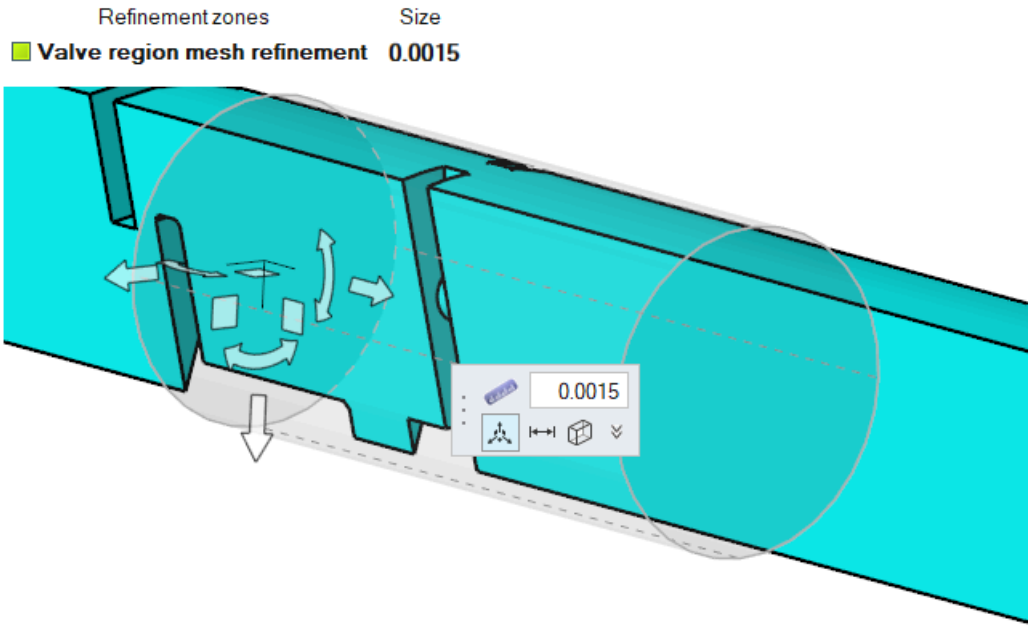


Figure 1109:

10. Save the model.

Define the Boundary Layer Controls

1. From the Mesh ribbon, click the **Boundary Layer** tool.

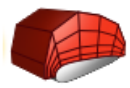


Figure 1110:

2. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Boundaries** > **Default Wall**.
   All the wall surfaces should be highlighted and a dialog for BL specification appears.
3. Enter the following values in the dialog:
   a) First layer thickness: `0.00035`
   b) Total number of layers: `3`
   c) Growth method: **Constant**
   d) Initial growth rate: `1.2`
   e) Termination policy: **Truncate**
   f) Activate **Enable surface mesh modification**.

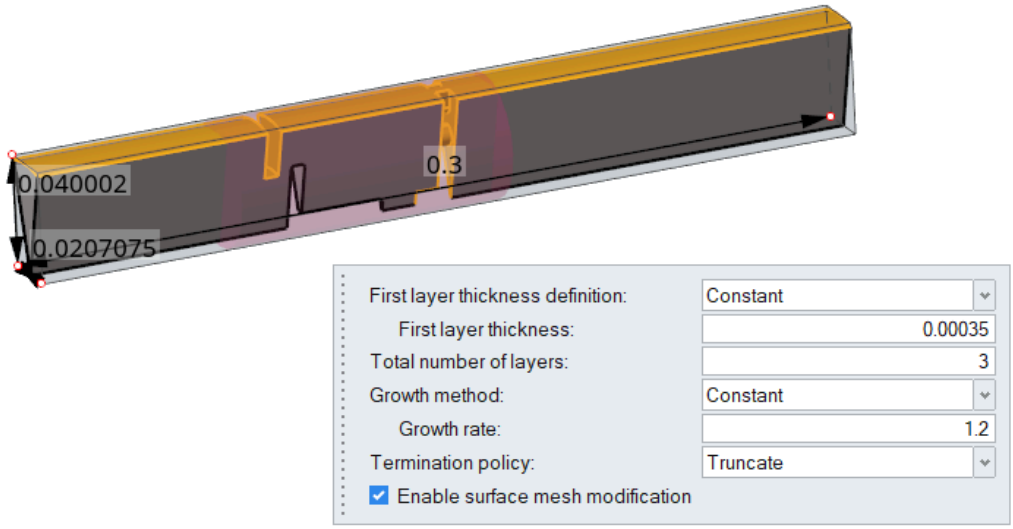


Figure 1111:

4. In the BL Controls legend, rename the BL control to `Pipe walls BL`.
5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. In the next few steps, you will define the boundary layer mesh control for the valve walls. For convenience, hide the front and back symmetry faces.
7. Select the valve wall surfaces using the box selection method.
8. In the dialog that appears, enter the following values for the BL specification:
   a) First layer thickness: `0.00015`

b) Total number of layers: `3`
c) Growth method: **Constant**
d) Initial growth rate: `1.2`
e) Termination policy: **Truncate**
f) Activate **Enable surface mesh modification**.

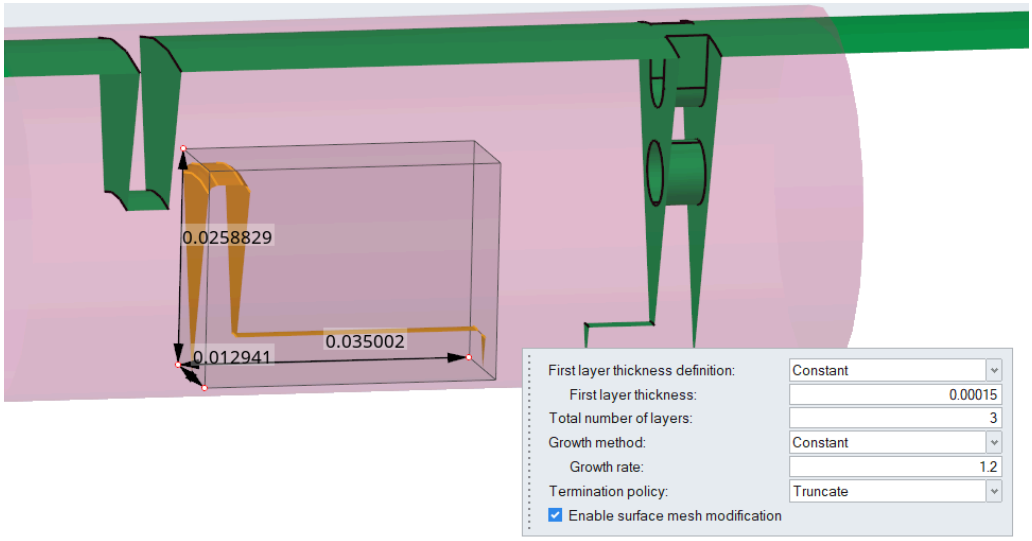


Figure 1112:

9. In the BL Controls legend, rename the BL control to `Valve walls BL`.

10. On the guide bar, click ✓ to execute the command and exit the tool.

11. Click the drop-down next to the **Mesh Controls** tool set and select **Advanced**.

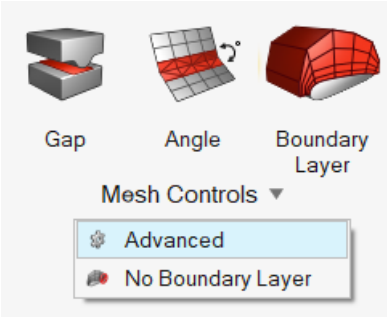


Figure 1113:

12. In the **Advanced mesh settings** dialog, change the Boundary layer element type to **Tetra** then close the dialog.

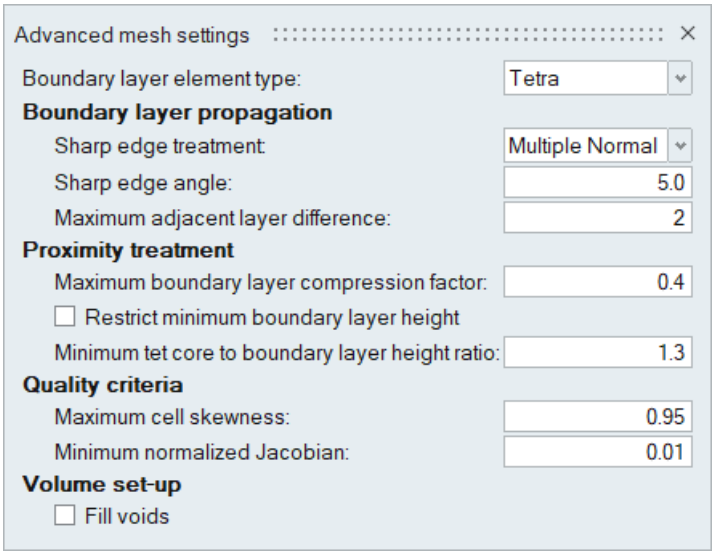


Figure 1114:

13. Turn on the display of all the surfaces and save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

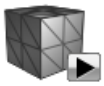


Figure 1115:

2. In the **Meshing Operations** dialog, set the Mesh size option to **Maximum size** and set the Maximum element size to `0.005`.

3. Deactivate **Curvature based surface refinement**.

4. Click **Mesh** to generate the mesh.

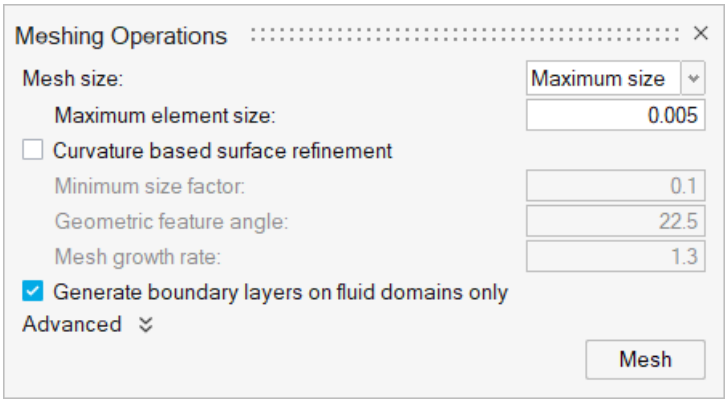


Figure 1116:

Once the meshing process has started, the **Run Status** dialog appears and the application moves to the Solution ribbon.

5. Once the meshing is completed, close the **Run Status** dialog.

Compute the Solution

Define Nodal Outputs

1. From the Solution ribbon, click the **Field** tool.

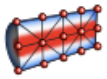


Figure 1117:

2. In the **Field Outputs** dialog, set the Time step interval to 3 for the Solution variables.

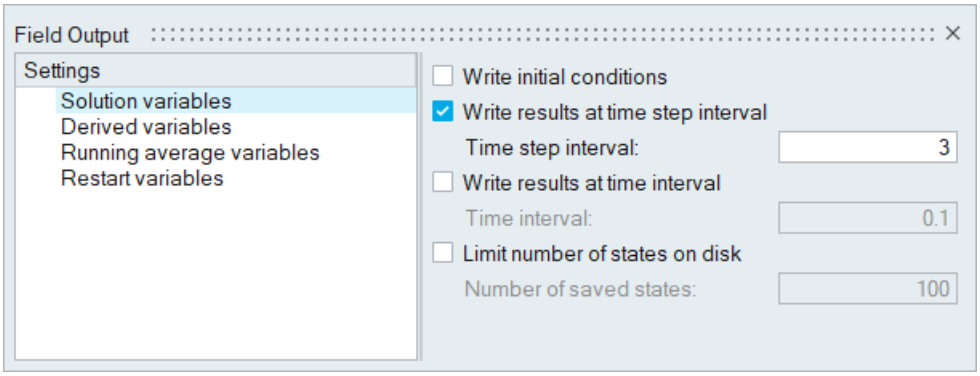


Figure 1118:

Launch AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1119:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

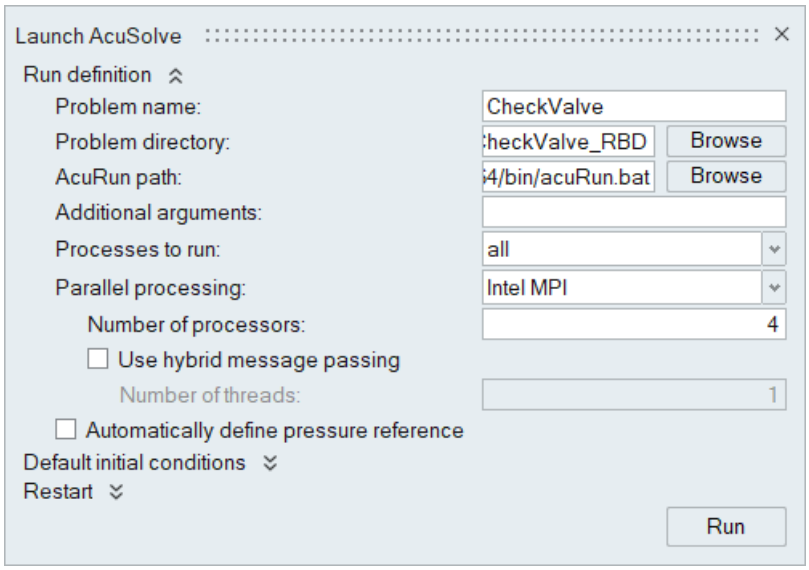


Figure 1120:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

Monitor Results

1. From the Solution ribbon, click the **Plot** tool.

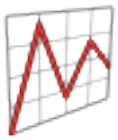


Figure 1121:

2. Click + next to the Library tab.
3. Keep the X-axis set to **Step**. For the Y-axis, click the tree browser and select **mesh_displacement** under Surface Output.
   mesh_x_displacement is selected by default for the coordinate direction.
4. Check the box for **Rigid Body - surface output** under Selector.
5. Check the box for **Twin Y-Axis**.
6. Click the tree browser for the second Y-axis and select **pressure** under Surface Output.
7. Check the box for **Inlet - Output** under Selector.

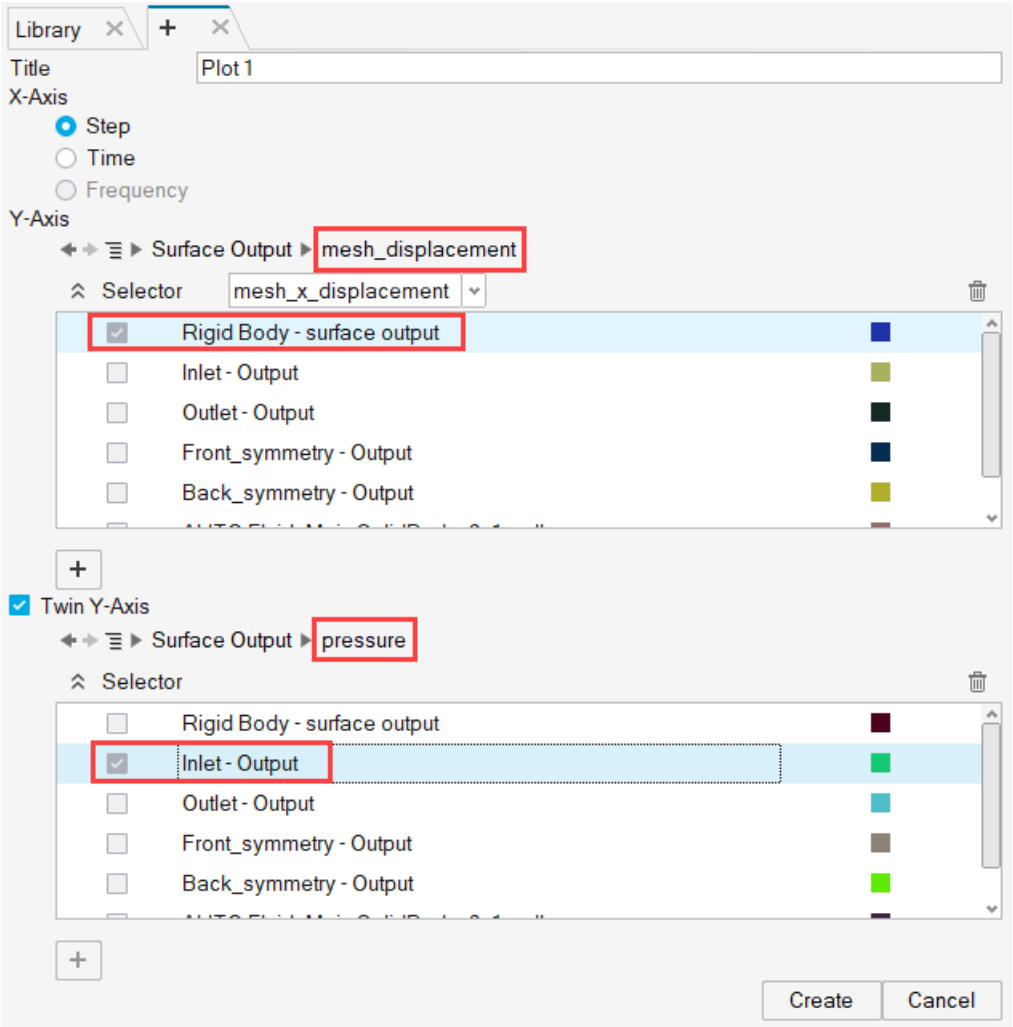


Figure 1122:

8. Click **Create**.

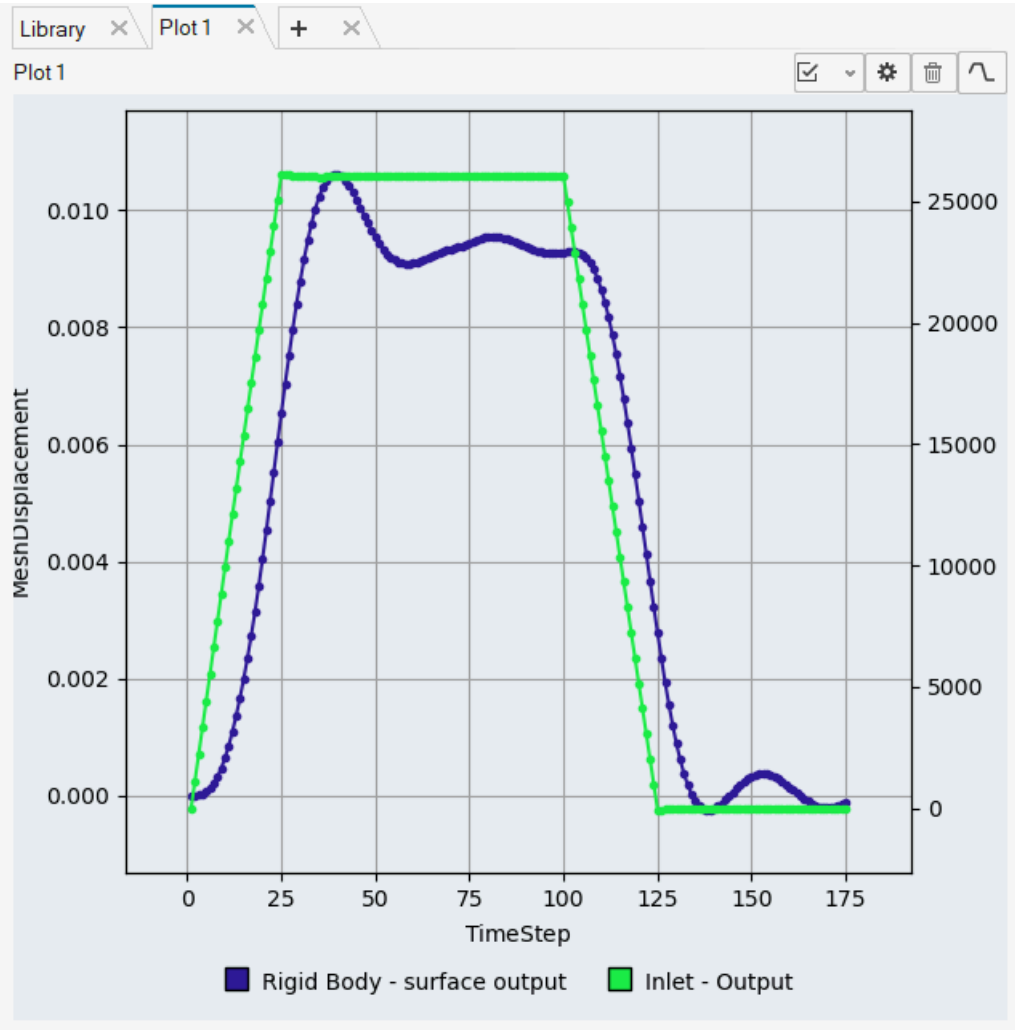


Figure 1123:

Post-Process the Results with HM-CFD Post

Create an Animation of Velocity Magnitude

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Click the **Top** face on the View Cube to align the model.

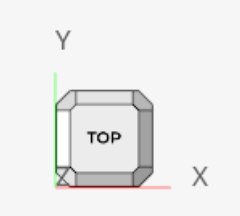


Figure 1124:

5. In the Post Browser, right-click on **Front_symmetry** and select **Isolate**.
6. Right-click on **Front_symmetry** once more and select **Edit**.
7. In the display properties microdialog, set the display to **velocity**.
8. Activate the **Legend** toggle and set the max velocity to `6.0`.
9. Click ≡ and set the Colormap Name to **Rainbow Uniform**.

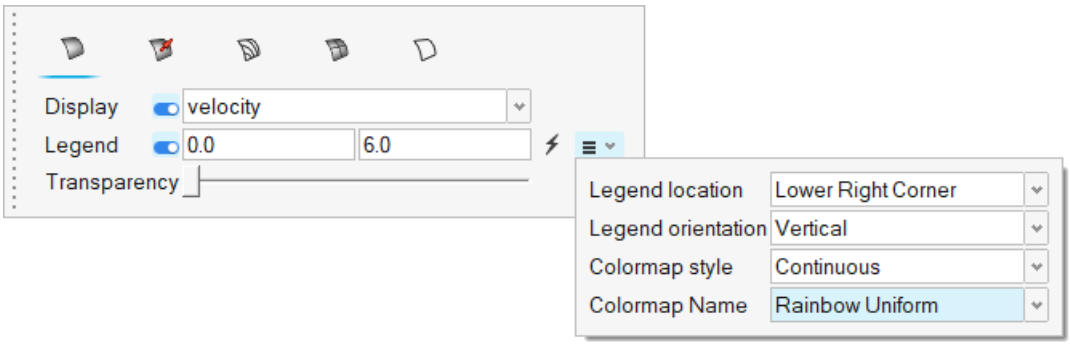


Figure 1125:

10. Click ✓ on the guide bar.
11. Click ▶ at the bottom of the modeling window to view a live animation of the flow.

Figure 1126:

12. Save the animation.
    a) Go to **File** > **Screen Capture** > **Advanced Capture**.
    b) Click ≡ on the toolbar.
    c) Uncheck **Include mouse cursor**.
    d) Set the frame rate to `50`.
    e) Click ⌖ on the toolbar then drag over the area you want to record.
    f) Click ◉ to start recording and the same button to stop recording.
    g) Name the file and save it.

Display Pressure Contours and Velocity Vectors on a Section Cut

1. Right-click in the modeling window and select **Show All**.
2. Click the **Slice Planes** tool.

Figure 1127:

3. Select the plane perpendicular to the z-axis.
4. Make sure the View Cube is aligned with the Top face.

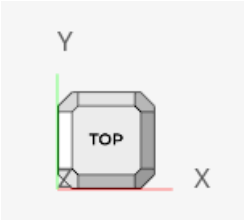


Figure 1128:

5. Click in the slice plane microdialog.
6. Turn off the **Display** in the Contour tab of the display properties microdialog.

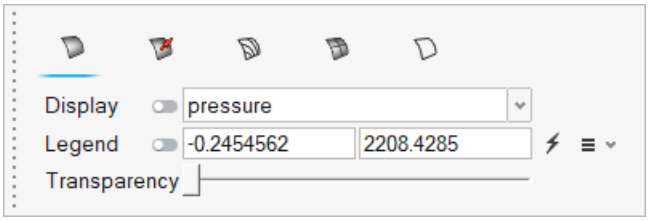


Figure 1129:

7. Click the **Vector Display** tab.
8. Turn the **Display** on, set the variable to **velocity**, and the value to `0.001`.
9. Set the Coloring to **pressure**.
10. Activate the **Legend** toggle and click to reset the range.
11. Click , set the Legend location to **Upper Center**, the Legend orientation to **Horizontal**, and the Colormap Name to **Rainbow Uniform**.

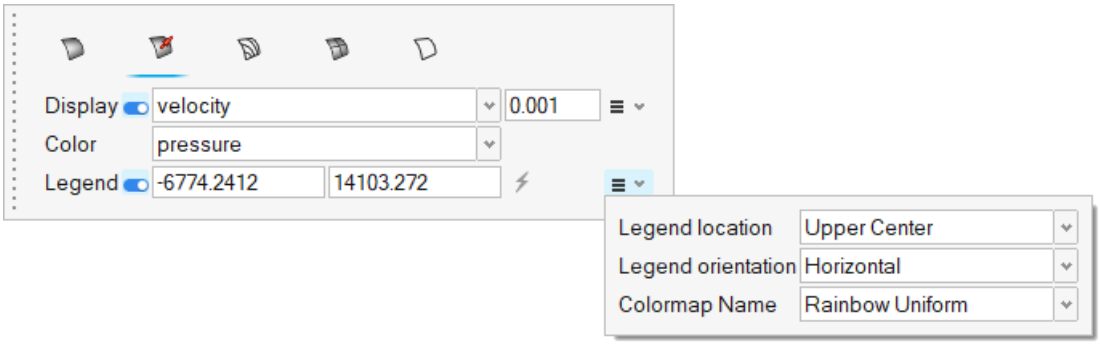


Figure 1130:

12. Click ✓ on the guide bar.
13. In the Post Browser, right-click on **Slice Plane 1** and select **Isolate**.

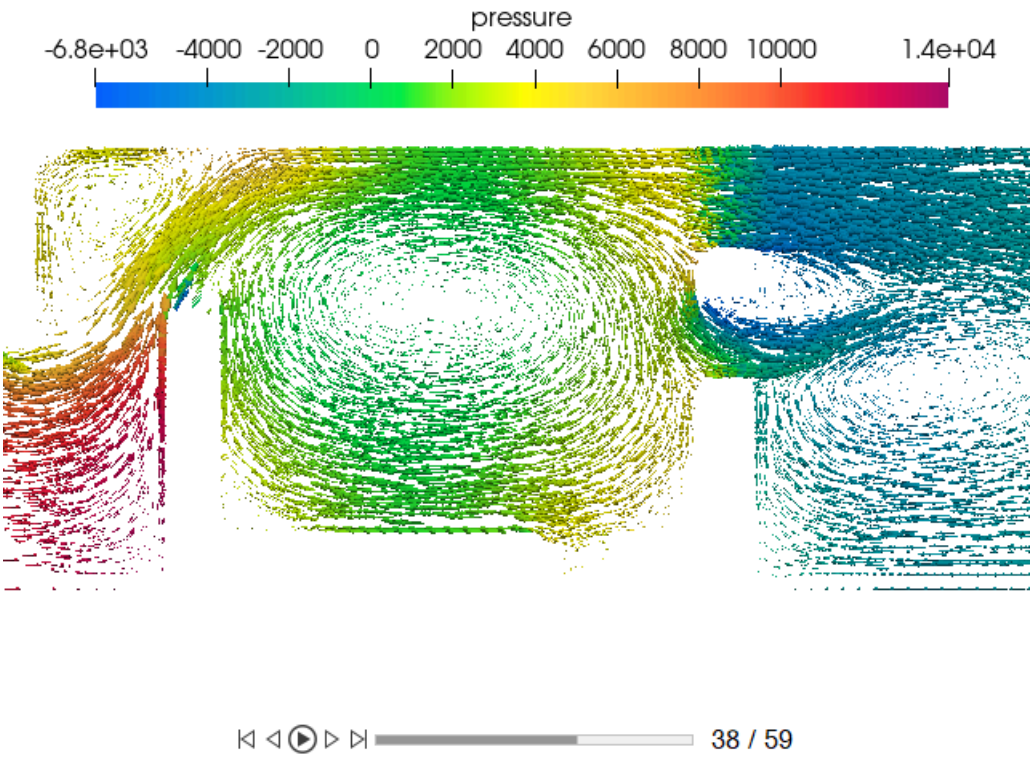


Figure 1131:

Summary

In this tutorial, you learned how to set up and solve a CFD simulation involving rigid body mesh motion. You started by importing geometry, then defined the flow set up. Next, you defined the mesh motion set up and then generated the mesh. Once the solution was computed, you post-processed the results using the Plot tool and HyperMesh CFD Post.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/ow3aoiksgz

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

Prerequisites

This tutorial introduces you to the workflow for setting up an AcuSolve-MotionSolve co-simulation using HyperMesh CFD. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD, MotionSolve, and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5201_pressureCheckValve.x_t`

`Valve_model.xml`

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1132. It consists of a cylindrical pipe containing water that flows past a check valve with a shutter attached to a virtual spring (not included in the geometry). The inlet pressure varies over time and the movement of the shutter will be determined as a function of the balance of the fluid forces against the reactive force of the spring. The problem is rotationally periodic at 30° increments about the longitudinal axis, and it is assumed that the resulting flow is also rotationally periodic, allowing for modeling with the use of a wedge-shaped section. For this tutorial, a 30° section of the geometry is modeled, as shown in the figure. Modeling a portion of an rotationally periodic geometry leads to reduced computation time while still providing an accurate solution.

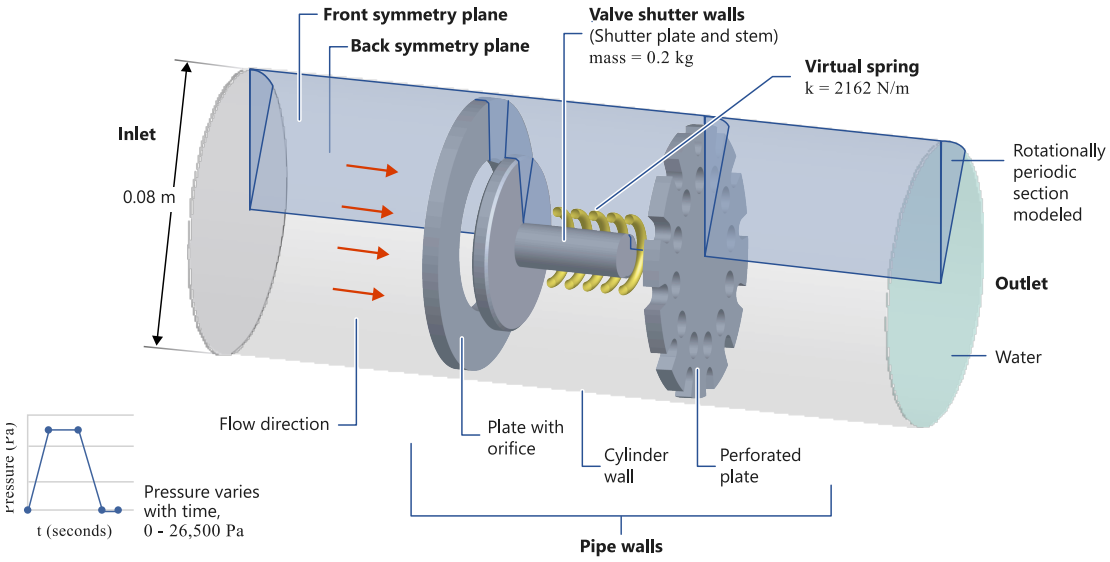


Figure 1132: Schematic of Check Valve with Spring-Loaded Shutter

The pipe has an inlet diameter of 0.08 m and is 0.3 m long. The check-valve assembly is 0.085 m downstream of the inlet. It consists of a plate 0.005 m thick with a centered orifice 0.044 m in diameter and a shutter with an initial position 0.005 m from the opening, simulating a nearly closed condition. The shutter plate is 0.05 m in diameter and 0.005 m thick. The shutter plate is attached to a stem 0.03 m long and 0.01 m in diameter. The mass of the shutter and stem is 0.2 kg and its motion is affected by a virtual spring with a stiffness of 2162 N/m. The motion of the valve shutter is limited by a stop mounted on a perforated plate downstream of the shutter.

Note that AcuSolve's internal rigid-body-dynamics solver is not able to simulate contact. Therefore, this problem is formulated to avoid contact between the valve and the stop.

Modeling the geometry as a 30° section requires that the fluid model is set up to be consistent with the rigid-body model. Since only $^{1}/_{12}$ of the rigid body is modeled, the forces computed by AcuSolve that act on the valve shutter represent $^{1}/_{12}$ of the actual force on the device. Therefore, it is also necessary to account for this in the simulation. There are two methods that can be used to accomplish this:

1. Scale up the fluid forces calculated by AcuSolve by a factor of 12 to represent the full load on the device when the displacement of the body is computed.

   Using this approach, the full stiffness of the valve spring is used in the rigid-body solution, and the full mass of the valve is used.
2. Scale down the mass of the valve and the stiffness of the spring to by a factor of 12 to match the fraction of the valve geometry to be modeled.

   Using this approach, the loading passed to the rigid-body solver is not scaled.

This second approach is used in this tutorial; the scaled mass of 0.0167 kg and the scaled stiffness of 180.1667 N/m will be used

The fluid in this problem is water, which has a density (ρ) of 1000 kg/m^3 and a molecular viscosity (μ) of 1 X 10^{-3} kg/m-sec.

At the start of the simulation, the flow field is stationary. Flow is driven by the pressure at the inlet, which varies over time as a piecewise linear function shown in Figure 1133. As the pressure at the inlet rises, the flow will accelerate as the valve opens. The turbulence viscosity ratio is assumed to be 10.

The initial inlet pressure is 0 Pa. At 0.002 s the pressure begins to ramp up and reaches 29,000 Pa at 0.05 s. The pressure is held at 29,000 Pa and begins to ramp back down starting at 0.2 s, reaching the initial pressure at 0.25 s where it remains for the rest of the simulation.

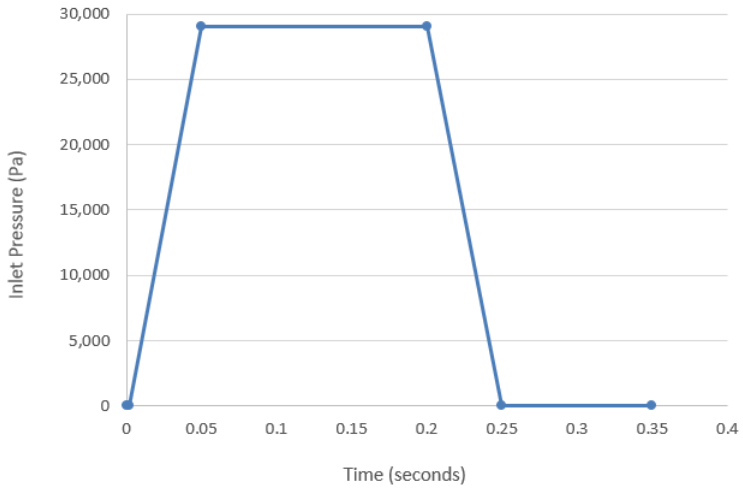


Figure 1133: Transient pressure at the inlet

Prior simulations of this geometry indicate that the average velocity at the inlet reaches a maximum of 0.98 m/s. At this velocity, the Reynolds number for the flow is 78,400. When the Reynolds number is above 4,000 it is generally accepted that flow should be modeled as turbulent.

Note that the initial conditions of the flow are actually laminar; however, the increase in flow velocity and flow around the valve shutter is expected to cause a rapid transition to turbulent conditions. Therefore, the simulation will be set up to model transient, turbulent flow. When performing a transient analysis, convergence is achieved at every time step based on the defined stagger criteria. Mesh motion will be modeled using arbitrary mesh movement (arbitrary Lagrangian-Eulerian mesh motion).

For this case, the transient behavior of interest occurs in the time it takes for the pressure to ramp up and ramp back down, which is given by the transient pressure profile. To allow time for the spring to recover additional time will be simulated. For this tutorial 0.1 s is added after the pressure drops back to initial conditions for a total duration of 0.35 s.

Another critical decision in a transient simulation is choosing the time increment. The time increment is the change in time during a given time step of the simulation. It is important to choose a time increment that is short enough to capture the changes in flow properties of interest, but does not require unnecessary computation time. The change in inlet pressure from initial conditions to maximum

occurs over 0.048 s. A time increment of 0.002 s would allow for excellent resolution of the transient changes without requiring excessive computational time.

Start HyperMesh CFD and Create the HyperMesh Model Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.

   When HyperMesh CFD is loaded, the Geometry ribbon is open by default.
2. Create a new `.hm` database in one of the following ways:
   - From the menu bar, click **File** > **Save**.
   - From the Home tools, Files tool group, click the **Save As** tool.

Figure 1134:

3. In the **Save File As** dialog, navigate to the directory where you would like to save the database.
4. Enter `CheckValve_Coupled` as the name for the database then click **Save**.

   This will be your problem directory and all the files related to the simulation will be stored in this location.
5. Move the `Valve_model.xml` file to the problem directory.

Import and Validate the Geometry

Import the Geometry

1. From the menu bar, click **File** > **Import** > **Geometry Model**.
2. In the **Import File** dialog, browse to your working directory then select **ACU-T5201_pressureCheckValve.x_t** and click **Open**.
3. In the **Geometry Import Options** dialog, leave all the default options unchanged then click **Import**.

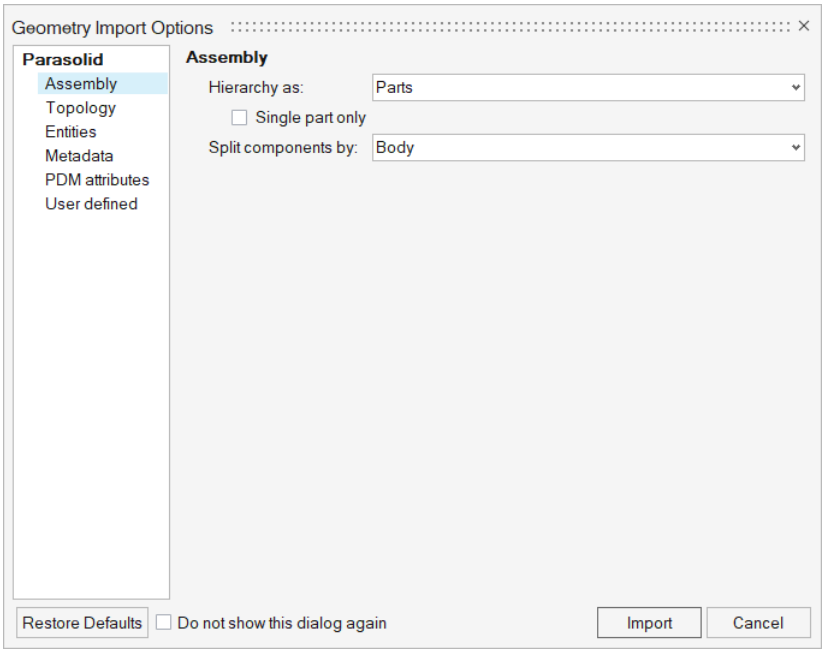


Figure 1135:

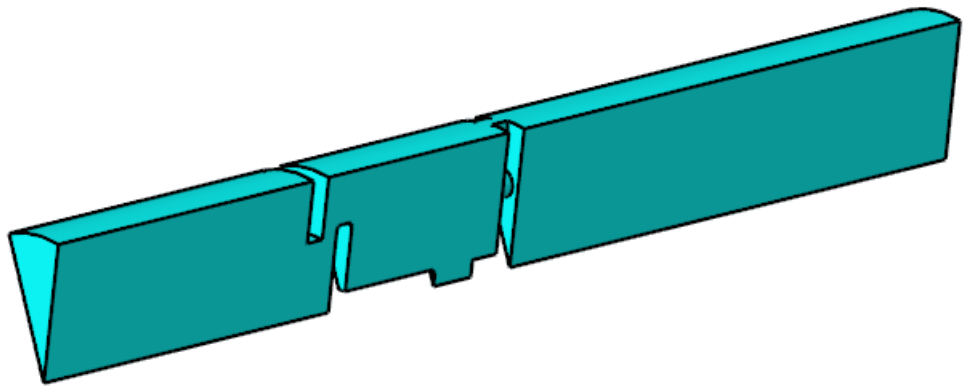


Figure 1136:

Validate the Geometry

1. From the Geometry ribbon, click the **Validate** tool.

Figure 1137:

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

The current model does not have any of the issues mentioned above. Alternatively, if any issues are found, they are indicated by the number in the brackets adjacent to the tool name.

Observe that a blue check mark appears on the top-left corner of the Validate icon. This indicates that the tool found no issues with the geometry model.

Figure 1138:

2. Press **Esc** or right-click in the modeling window and swipe the cursor over the green check mark from right to left.
3. Save the database.

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

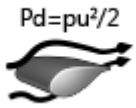


Figure 1139:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Transient**.
   b) Set the Time step size to `0.002` and the Final time to `0.35`.
   c) Select **Spalart-Allmaras** as the Turbulence model.

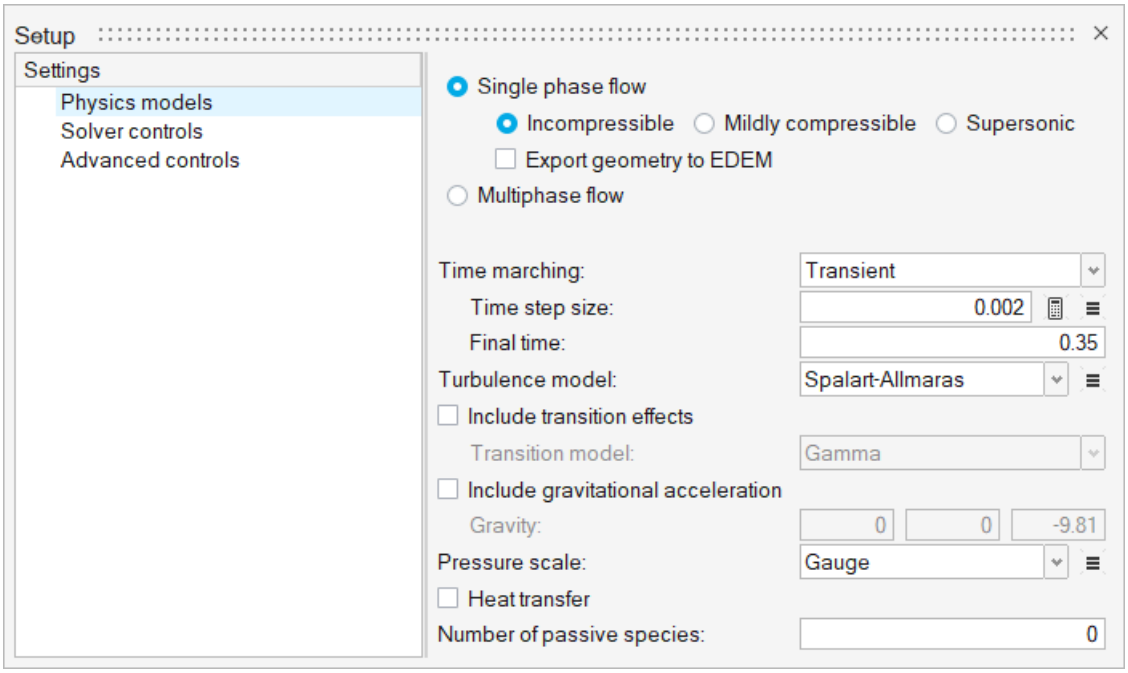


Figure 1140:

3. Click the **Solver controls** setting and set the Maximum stagger iterations to `3`.

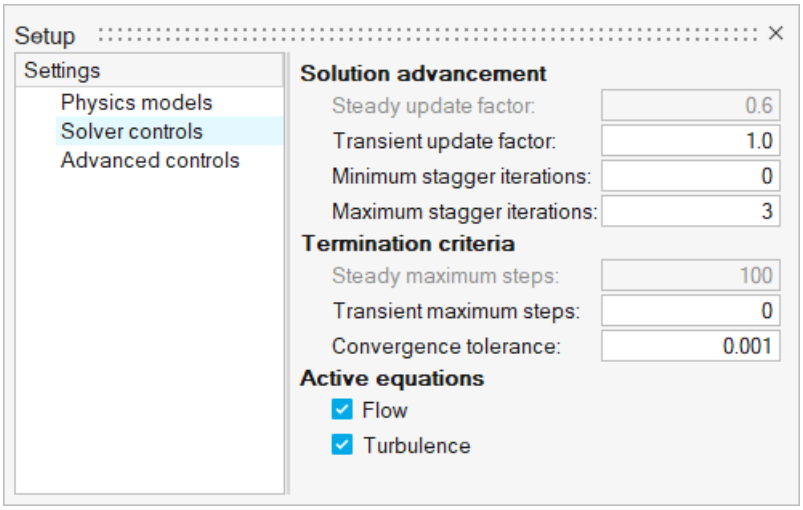


Figure 1141:

4. Close the dialog and save the model.

Create a Multiplier Function for the Inlet Pressure

1. From the Flow ribbon, click the arrow next to the Setup tool set, then select **Multipliers**.

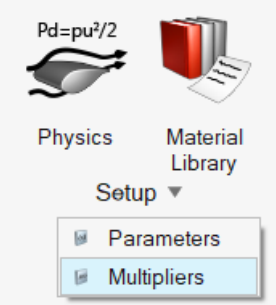


Figure 1142:

2. Click + in the **Multiplier Library** dialog.
3. In the multiplier creation dialog, change the name of the multiplier function to `Inlet Pressure` by clicking on the top-left corner.
4. Set the Type to **Piecewise Linear**.
5. Verify that the Variable is set to **Time** and Evaluation is set to **Time Step**.
6. Click four times to add four rows to the bottom of the table.
7. Enter the table values according to the image below.

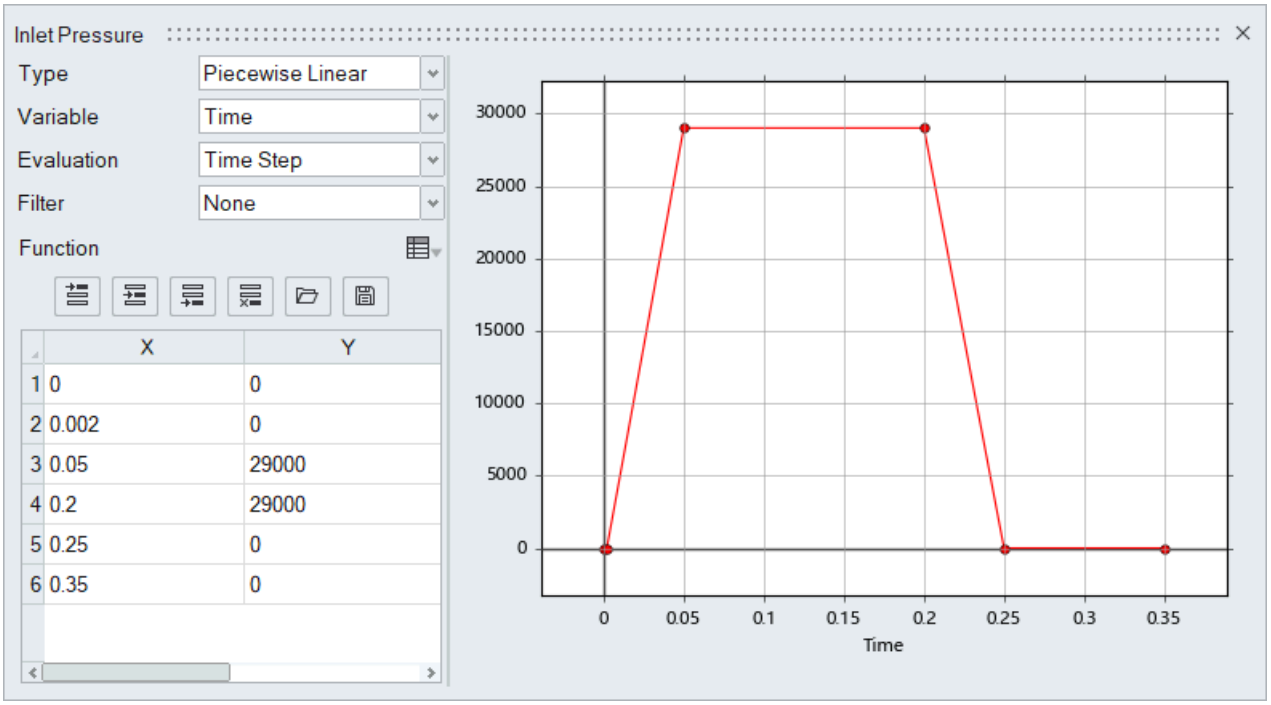


Figure 1143:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1144:

2. Click anywhere on the pipe geometry.
   The entire geometry is highlighted.

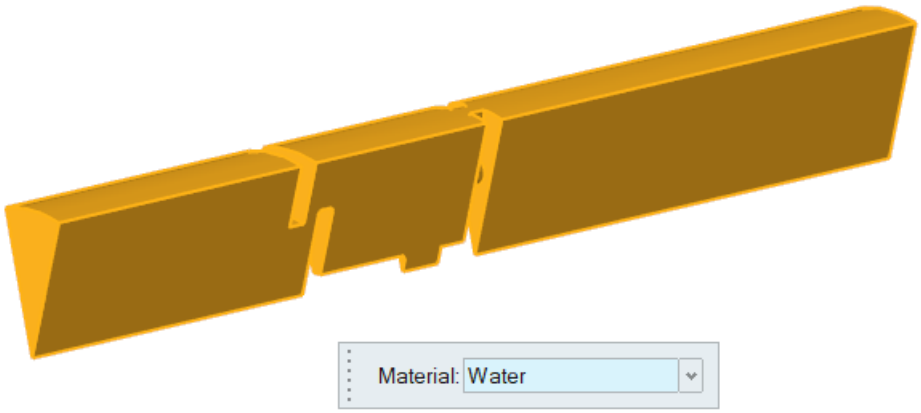


Figure 1145:

3. In the microdialog, select **Water** from the Material drop-down menu.
4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Save the model.

Define Flow Boundary Conditions

In this step, you will assign the boundary conditions for the inlet, outlet and the symmetry faces.

1. From the Flow ribbon, Pressure tool group, click the **Stagnation Pressure** tool.

Figure 1146:

2. In the modeling window, click on the inlet surface, as shown in Figure 1147.
3. In the microdialog, enter a value of `1` N/m^2 for the Stagnation pressure.
4. Click the Multiplier function drop-down and select the function **Inlet Pressure** from the list.

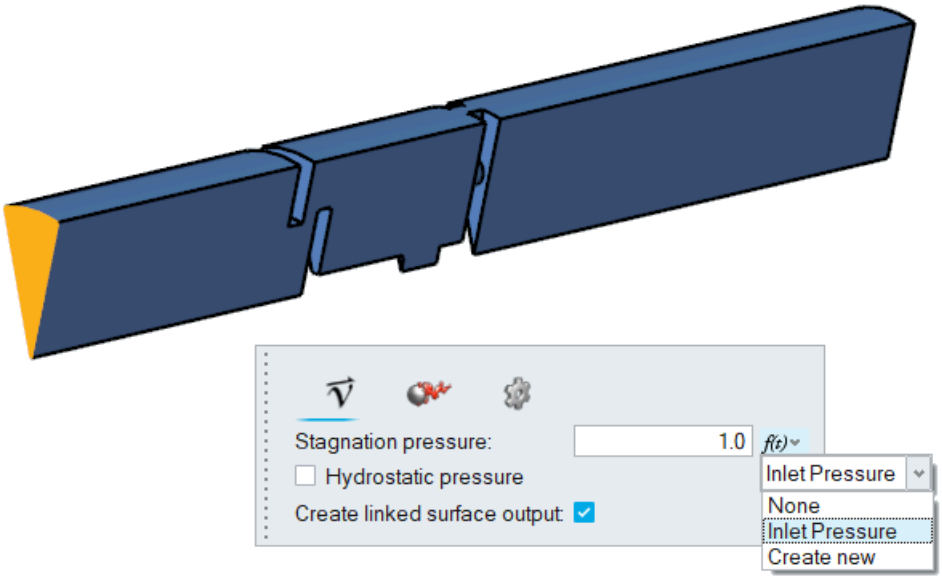


Figure 1147:

5. Click the Turbulence icon the microdialog, set the Turbulence input type to **Viscosity Ratio**, and the enter a value of `10` for the Turbulence viscosity ratio.

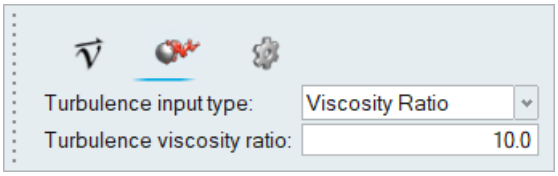


Figure 1148:

6. On the guide bar, click ▸ to execute the command and remain in the tool.
7. In the Boundaries legend, right-click on **Stagnation pressure** and rename it to `Inlet` then press **Enter**.
8. Click the **Outlet** tool.

Figure 1149:

9. Click the outlet surface shown in the figure below.

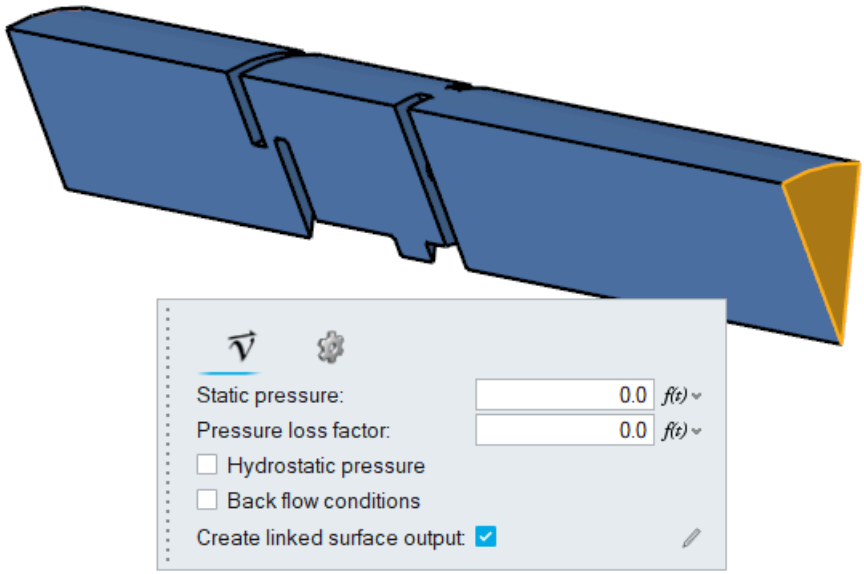


Figure 1150:

10. On the guide bar, click ✓ to execute the command and exit the tool.

11. Click the **Symmetry** tool.

Figure 1151:

12. Select the two surfaces shown in the figure below.

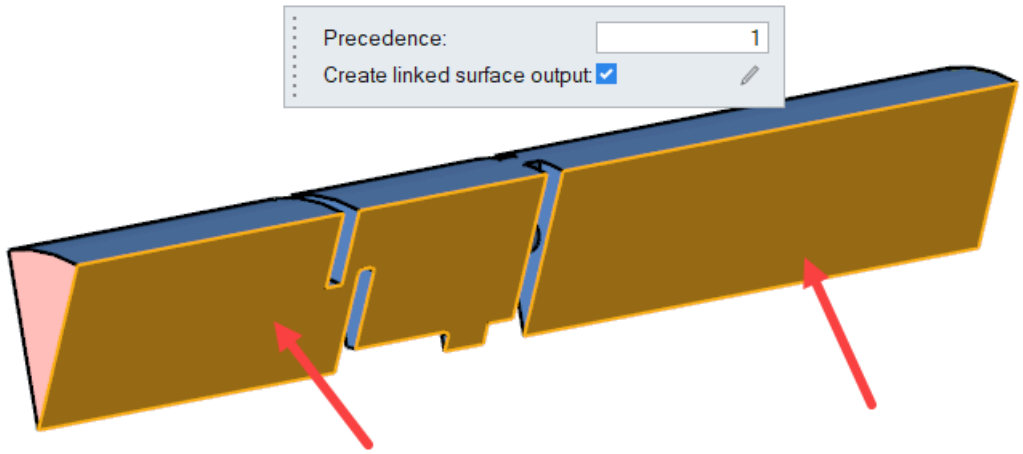


Figure 1152:

13. On the guide bar, click ▸ to execute the command and remain in the tool.
14. In the Boundaries legend, rename Symmetry to `Front_symmetry`.
15. Rotate the model and select the other two symmetry faces.

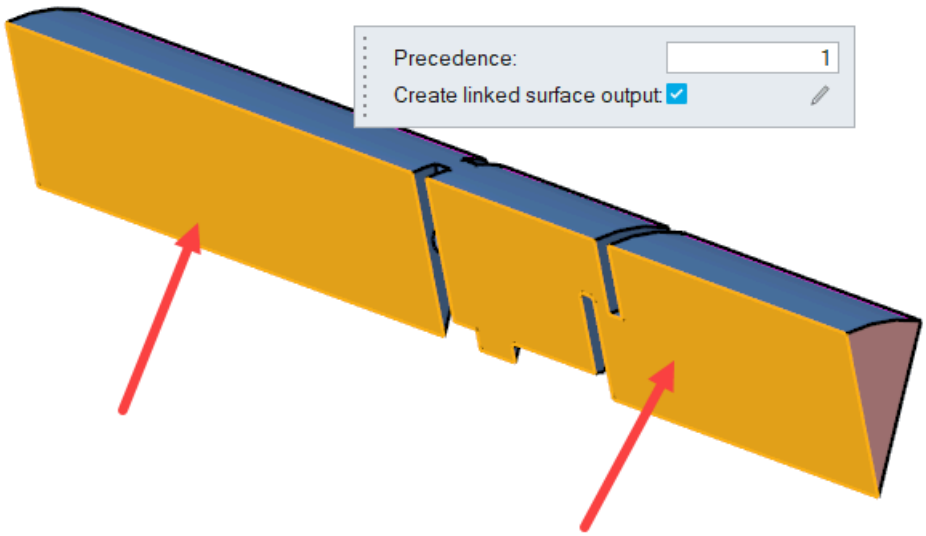


Figure 1153:

16. On the guide bar, click ▸ to execute the command and remain in the tool.
17. In the Boundaries legend, rename Symmetry to `Back_symmetry`.
18. Save the model.

Set Up Motion

In this step, you will activate the mesh motion and define the rigid body motion for the valve wall. Then, you will define the mesh displacement boundary conditions for the symmetry surfaces.

Define the Mesh Motion Type

1. From the Motion ribbon, click the **Settings** tool.

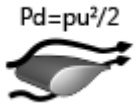


Figure 1154:

2. In the **Setup** dialog, change the Mesh motion to **Arbitrary**.

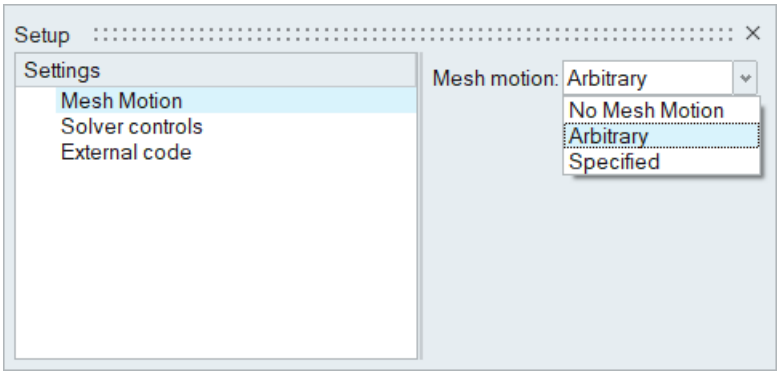


Figure 1155:

3. Click the **External code** setting and activate the **External code** option.
4. Set the External Solver to **MotionSolve** and the Communication type to **Socket**.
5. Deactivate the **External socket** option and set the Socket host to **localhost**.
6. Set the Socket port to `48000`.

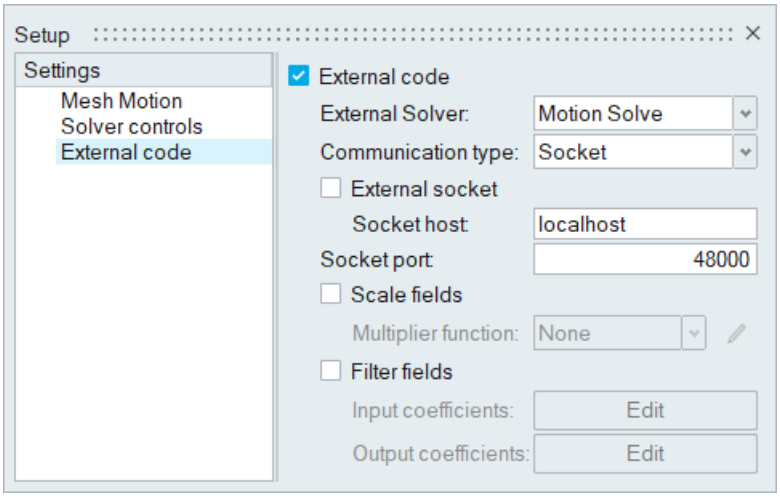


Figure 1156:

7. Close the dialog.

Define the Mesh Displacement Boundary Conditions

1. From the Motion ribbon, click the **Planar Slip** tool.

Figure 1157:

2. Select the two front symmetry surfaces shown Figure 1158.
3. In the Mesh Motion legend, rename Planar Slip to `Front_symmetry` by double-clicking on the name.

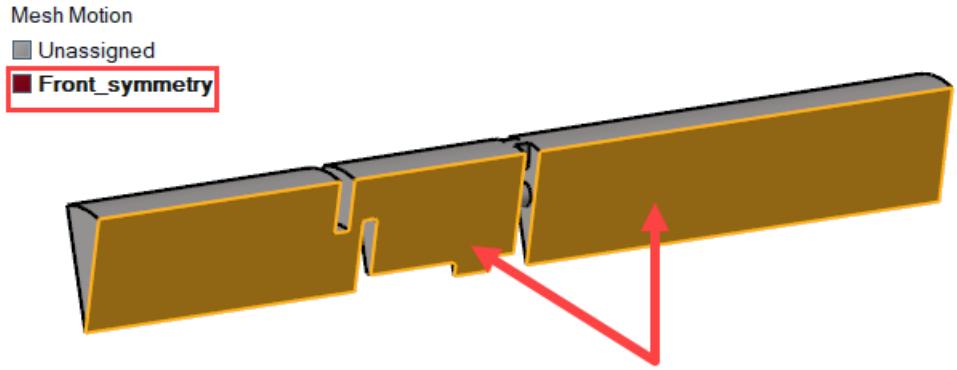


Figure 1158:

4. On the guide bar, click ▸ to execute the command and remain in the tool.
5. Rotate the model and select the two back symmetry faces.
6. In the Mesh Motion legend, rename Planar Slip to `Back_symmetry`.
7. On the guide bar, click ✓ to execute the command and exit the tool.
8. Click the **External Surface** tool.

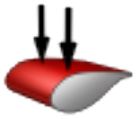


Figure 1159:

9. On the guide bar, make sure the **Surfaces** selector is active.
10. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Boundaries** > **Default Wall**.
11. Now that all the wall surfaces are highlighted, isolate the selected surfaces either by pressing the **I** key, or by right-clicking in the modeling window and selecting **Isolate**.
12. Zoom in on the valve region and select all the valve wall surfaces using the window selection method.

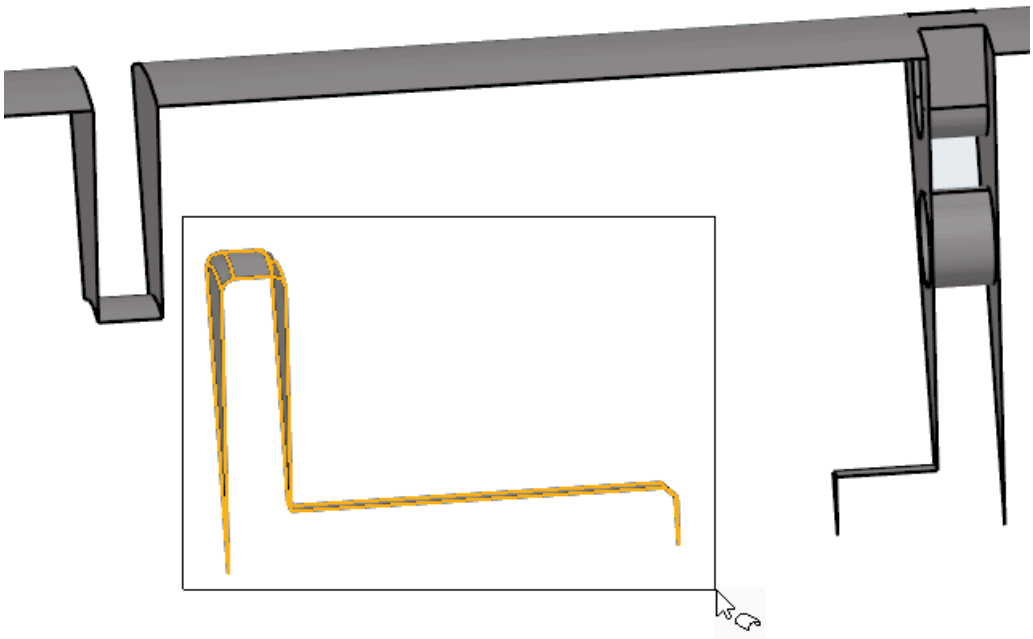


Figure 1160:

13. In the microdialog, set the Coupling direction to **Both** and enter `Model-Shutter Body` as the name of the Rigid body.

> **Note:** The name of the rigid body specified here should match the name of the rigid body in the MotionSolve model.

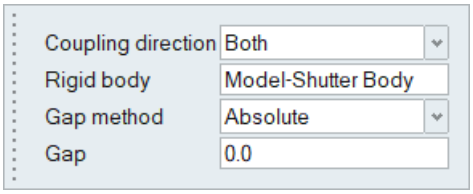


Figure 1161:

14. In the External Code Surfaces legend, rename the External Surface to `Valve wall`.
15. Right-click **Valve wall** and select isolate.
16. On the guide bar, click ✓ to execute the command and exit the tool.
17. Click the **No Constraints** tool.

Figure 1162:

18. Select the external surfaces of **Valve wall**.
19. On the guide bar, click ✓ to execute the command and exit the tool.
20. Turn on the display of all surfaces and save the model.

Generate the Mesh

In this step, you will define the mesh controls and then generate the mesh.

Define the Zone Mesh Controls

The zone mesh controls allow you to define a local volume mesh size for specific regions using standard shaped zones like cylindrical, spherical, and cuboidal zones.

1. From the Mesh ribbon, Zones tool group, click the **Cylinder** tool.

Figure 1163:

2. In the modeling window, hover the mouse around the point shown in the figure below. When the preview cylinder zone is parallel to the axis of the pipe, click on the model near the point shown below.

   This point will be the center of the front face of the cylinder. In the next few steps you will edit the co-ordinates of this point manually.

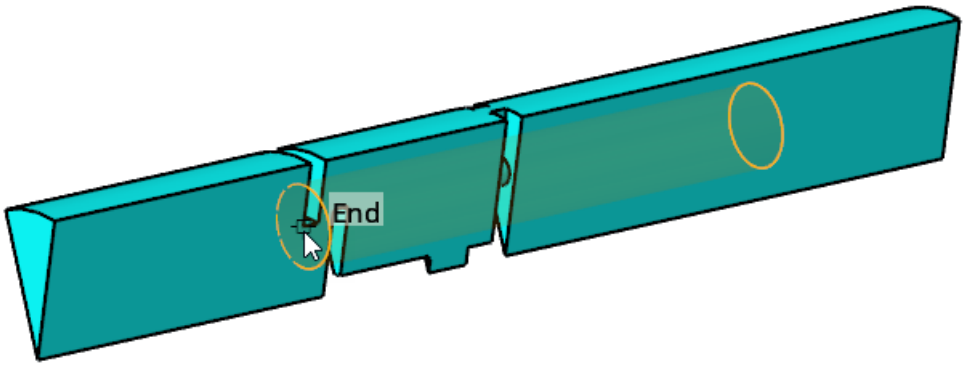


Figure 1164:

3. Move the mouse cursor away from the center and then click again.

   A preview of the zone should be displayed on the screen along with the manipulator. The manipulator allows you to modify the location and orientation of the zone.

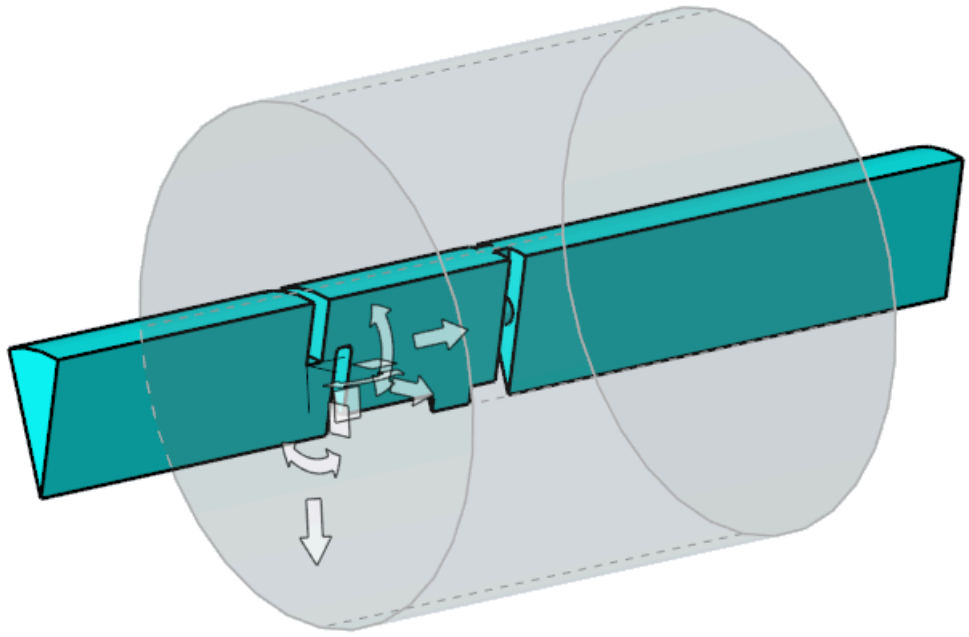


Figure 1165:

4. Click the center of the manipulator and enter the following coordinates for the center (`-0.06, 0.02, 0`)

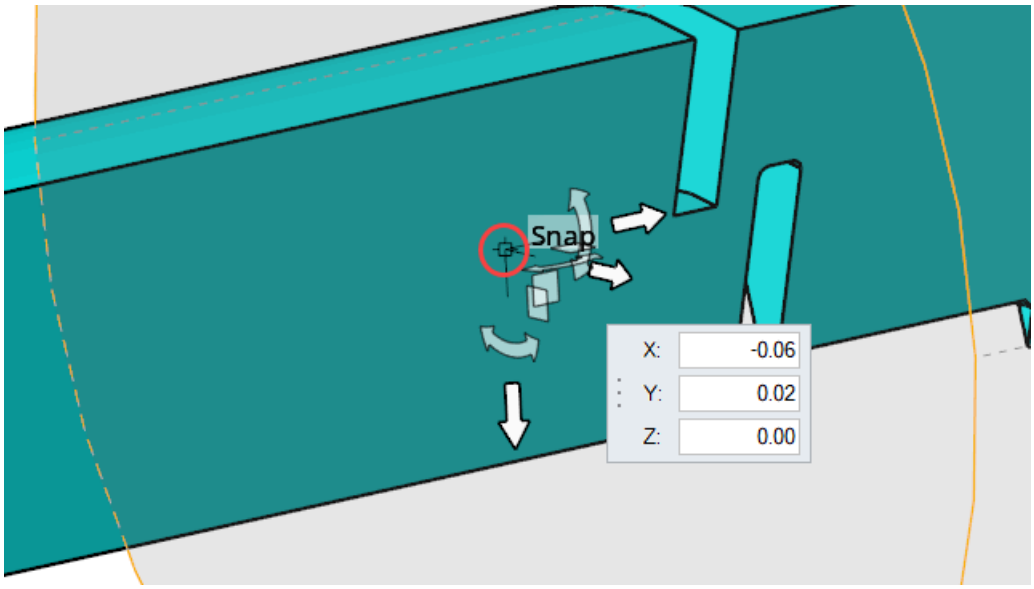


Figure 1166:

5. Double-click on the cylindrical surface of the zone and enter `0.022` m as the radius of the cylinder.

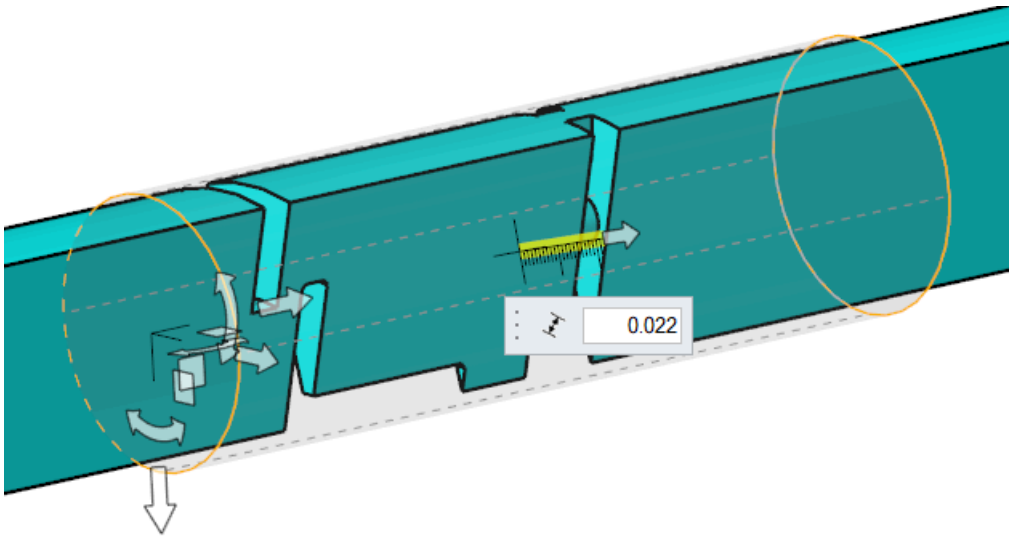


Figure 1167:

6. Double-click on the base of the cylinder and enter `0.1` m as the height of the cylinder.

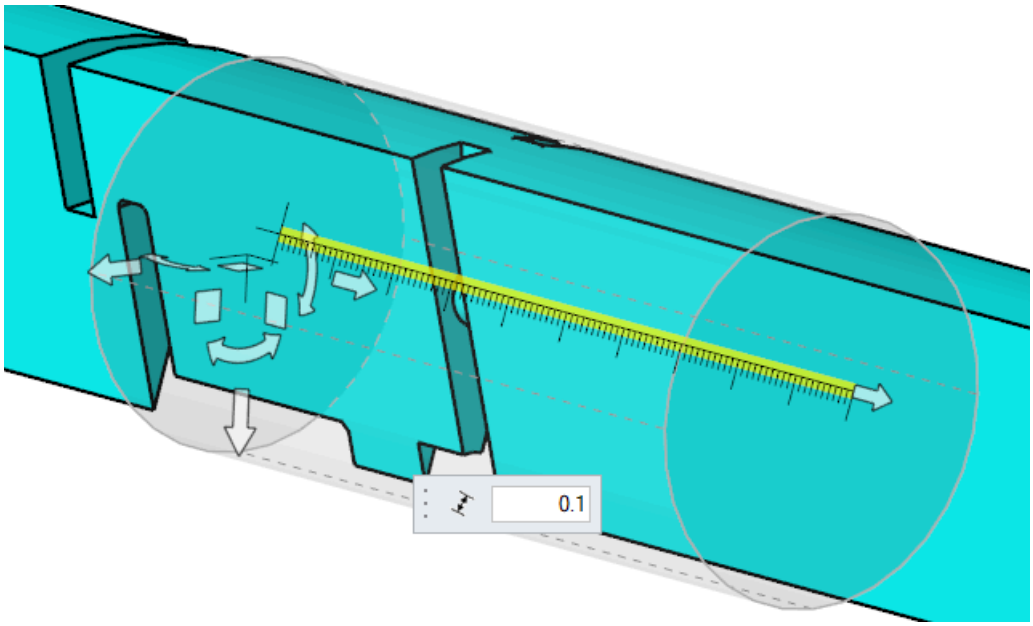


Figure 1168:

7. Click anywhere in the empty space in the modeling window.
8. Enter a value of `0.0015` m for the mesh size in the microdialog.
9. In the Refinement zones legend, rename the zone to `Valve region mesh refinement`.

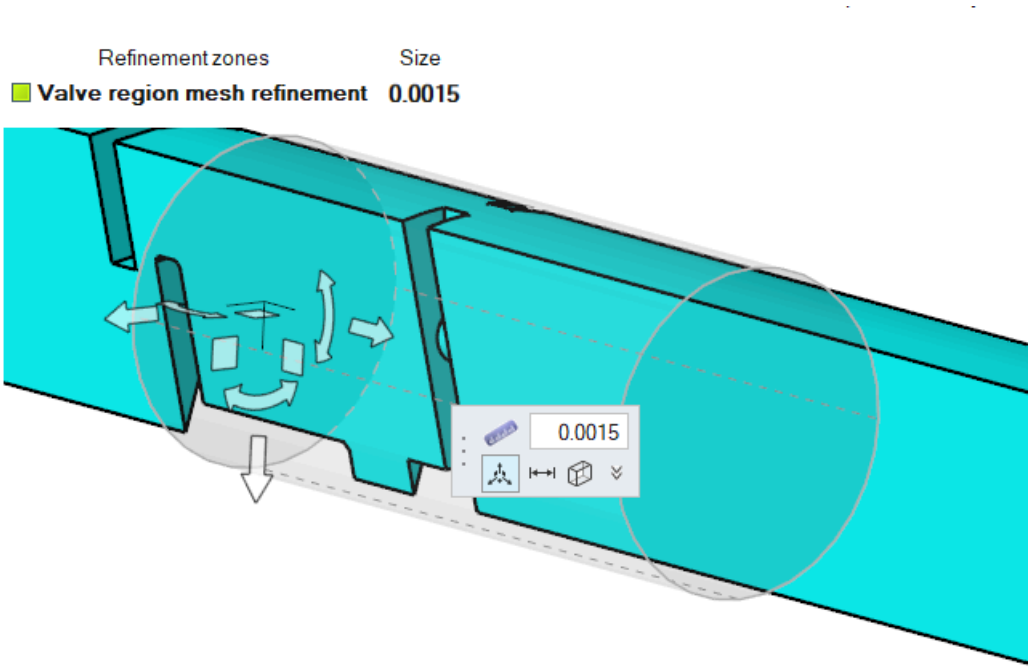


Figure 1169:

10. Save the model.

Define the Boundary Layer Controls

1. From the Mesh ribbon, click the **Boundary Layer** tool.

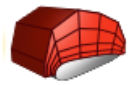


Figure 1170:

2. Right-click in the modeling window and go to **Select** > **Advanced Select** > **By Boundaries** > **Default Wall**.
All the wall surfaces should be highlighted and a dialog for BL specification appears.

3. Enter the following values in the dialog:
   a) First layer thickness: `0.00035`
   b) Total number of layers: `3`
   c) Growth method: **Constant**
   d) Initial growth rate: `1.2`
   e) Termination policy: **Truncate**
   f) Activate **Enable surface mesh modification**.

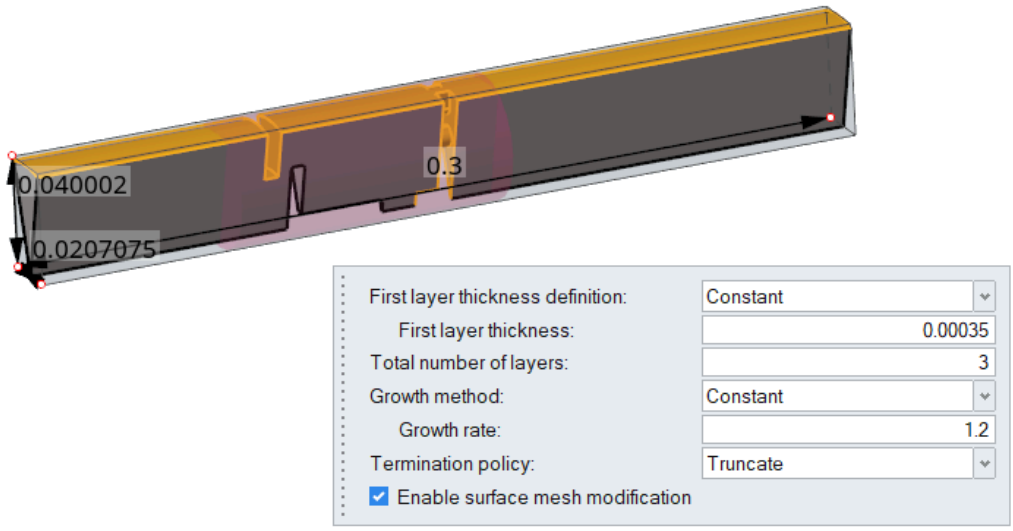


Figure 1171:

4. In the BL Controls legend, rename the BL control to `Pipe walls BL`.
5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. In the next few steps, you will define the boundary layer mesh control for the valve walls. For convenience, hide the front and back symmetry faces.
7. Select the valve wall surfaces using the box selection method.
8. In the dialog that appears, enter the following values for the BL specification:
   a) First layer thickness: `0.00015`
   b) Total number of layers: `3`
   c) Growth method: **Constant**
   d) Initial growth rate: `1.2`
   e) Termination policy: **Truncate**
   f) Activate **Enable surface mesh modification**.

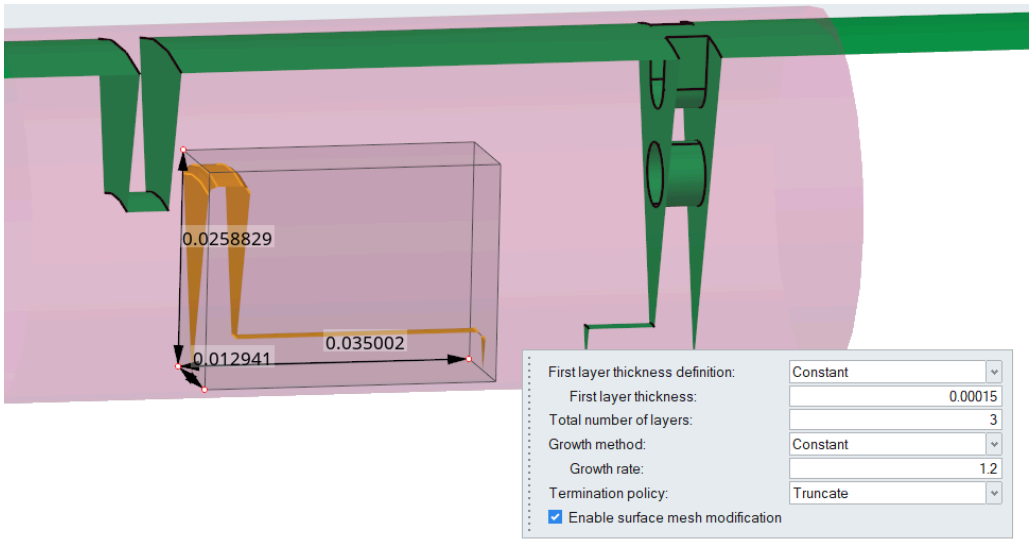


Figure 1172:

9. In the BL Controls legend, rename the BL control to `Valve walls BL`.

10. On the guide bar, click ✓ to execute the command and exit the tool.

11. Click the drop-down next to the **Mesh Controls** tool set and select **Advanced**.

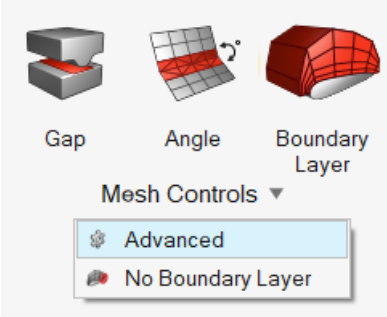


Figure 1173:

12. In the **Advanced mesh settings** dialog, change the Boundary layer element type to **Tetra** then close the dialog.

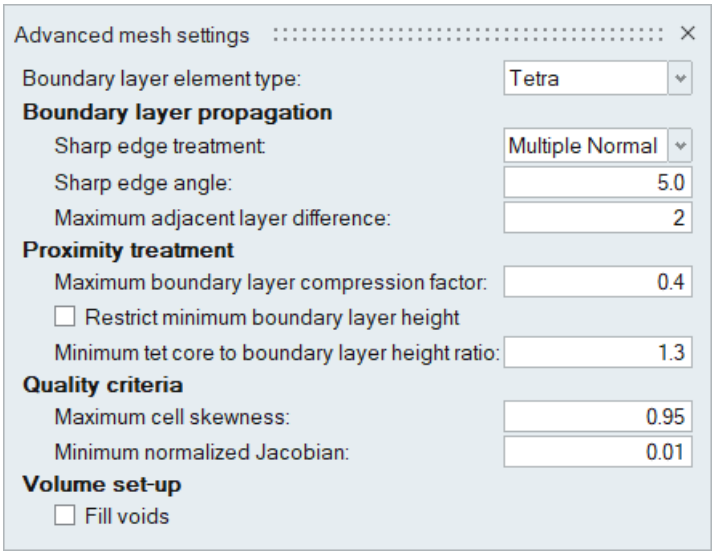


Figure 1174:

13. Turn on the display of all the surfaces and save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

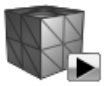


Figure 1175:

2. In the **Meshing Operations** dialog, set the Mesh size option to **Maximum size** and set the Maximum element size to `0.005`.

3. Deactivate **Curvature based surface refinement**.

4. Click **Mesh** to generate the mesh.

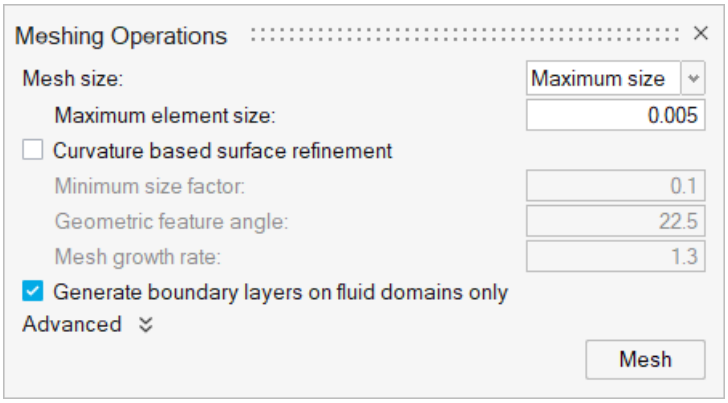


Figure 1176:

Once the meshing process has started, the **Run Status** dialog appears and the application moves to the Solution ribbon.

5. Once the meshing is completed, close the **Run Status** dialog.

Compute the Solution

Define Nodal Outputs

1. From the Solution ribbon, click the **Field** tool.

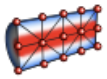


Figure 1177:

2. In the **Field Outputs** dialog, set the Time step interval to `1` for the Solution variables.

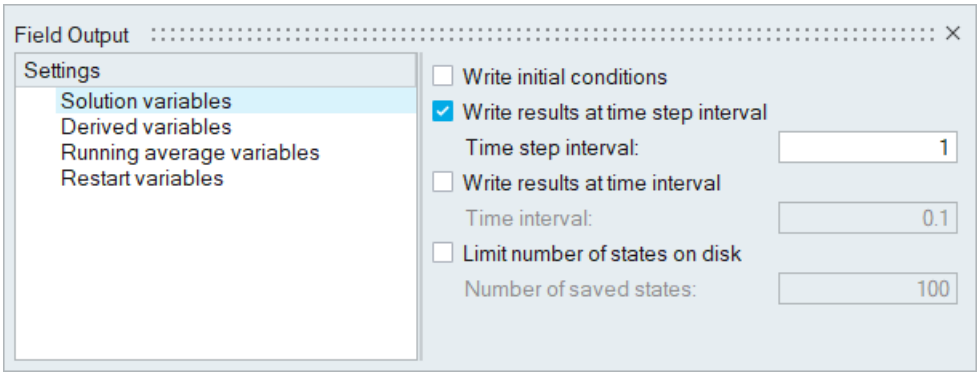


Figure 1178:

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1179:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Leave the remaining options as default and click **Run** to launch AcuSolve.

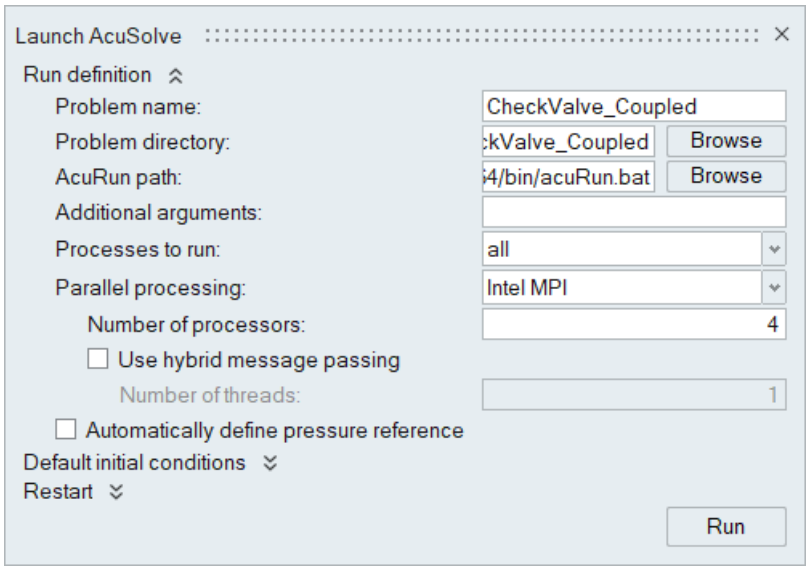


Figure 1180:

The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

Run MotionSolve

1. Run **Start** > **All Programs** > **Altair <version>** > **Compute Console**.
2. Select **Solver** > **MotionSolve** from the menu bar.

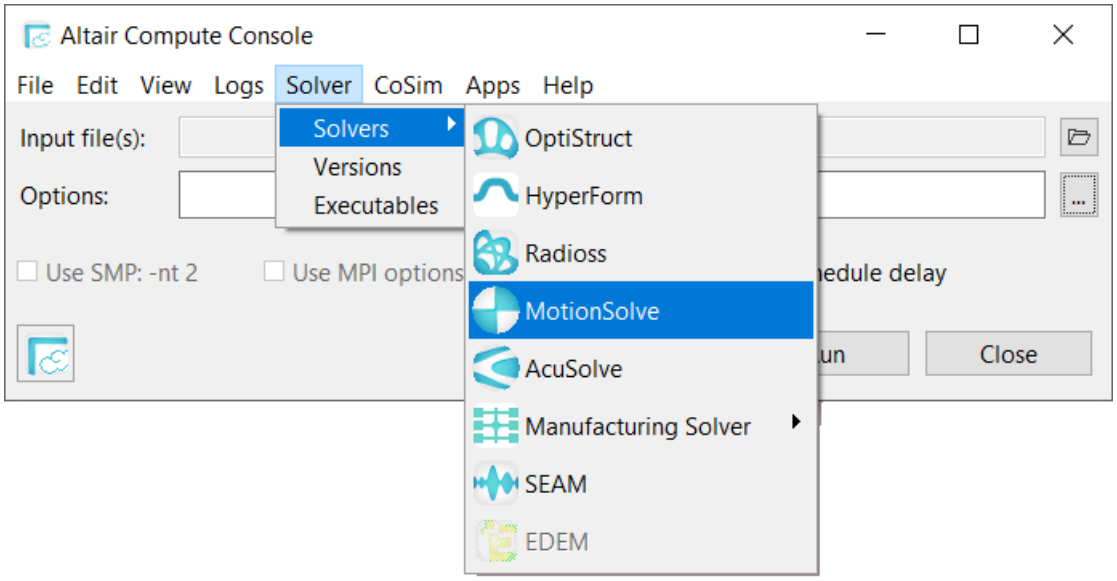


Figure 1181:

3. Click 🗁 besides the Input file(s) field, browse to the location where you saved `Valve_model.xml`, and open it
4. For Options, type `-as_cosim` to indicate coupling between MotionSolve and AcuSolve.

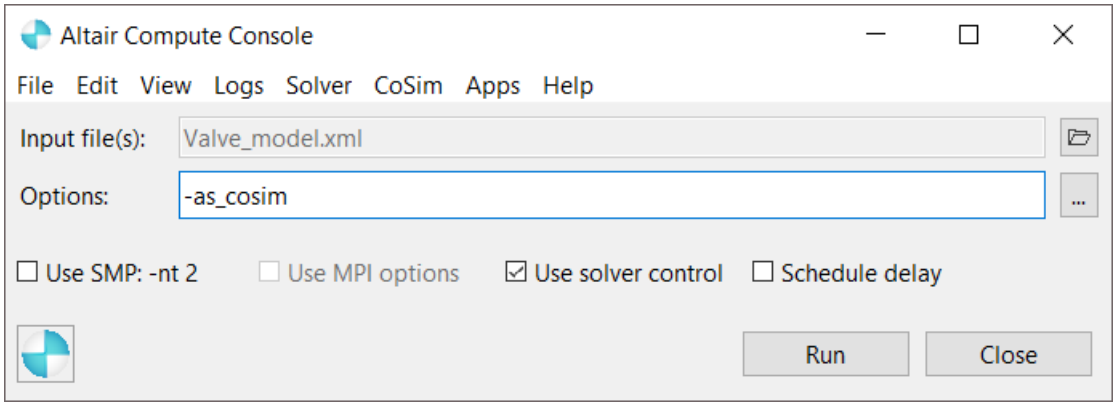


Figure 1182:

5. Click **Run** to start MotionSolve.

Monitor the Results

1. From the Solution ribbon, click the **Plot** tool.

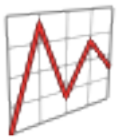


Figure 1183:

2. Click + next to the Library tab.
3. Keep the X-axis set to **Step**. For the Y-axis, click the tree browser and select **mesh_displacement** under Surface Output.
   mesh_x_displacement is selected by default for the coordinate direction.
4. Check the box for **Valve wall - Output** under Selector.
5. Check the box for **Twin Y-Axis**.
6. Click the tree browser for the second Y-axis and select **pressure** under Surface Output.
7. Check the box for **Inlet - Output** under Selector.

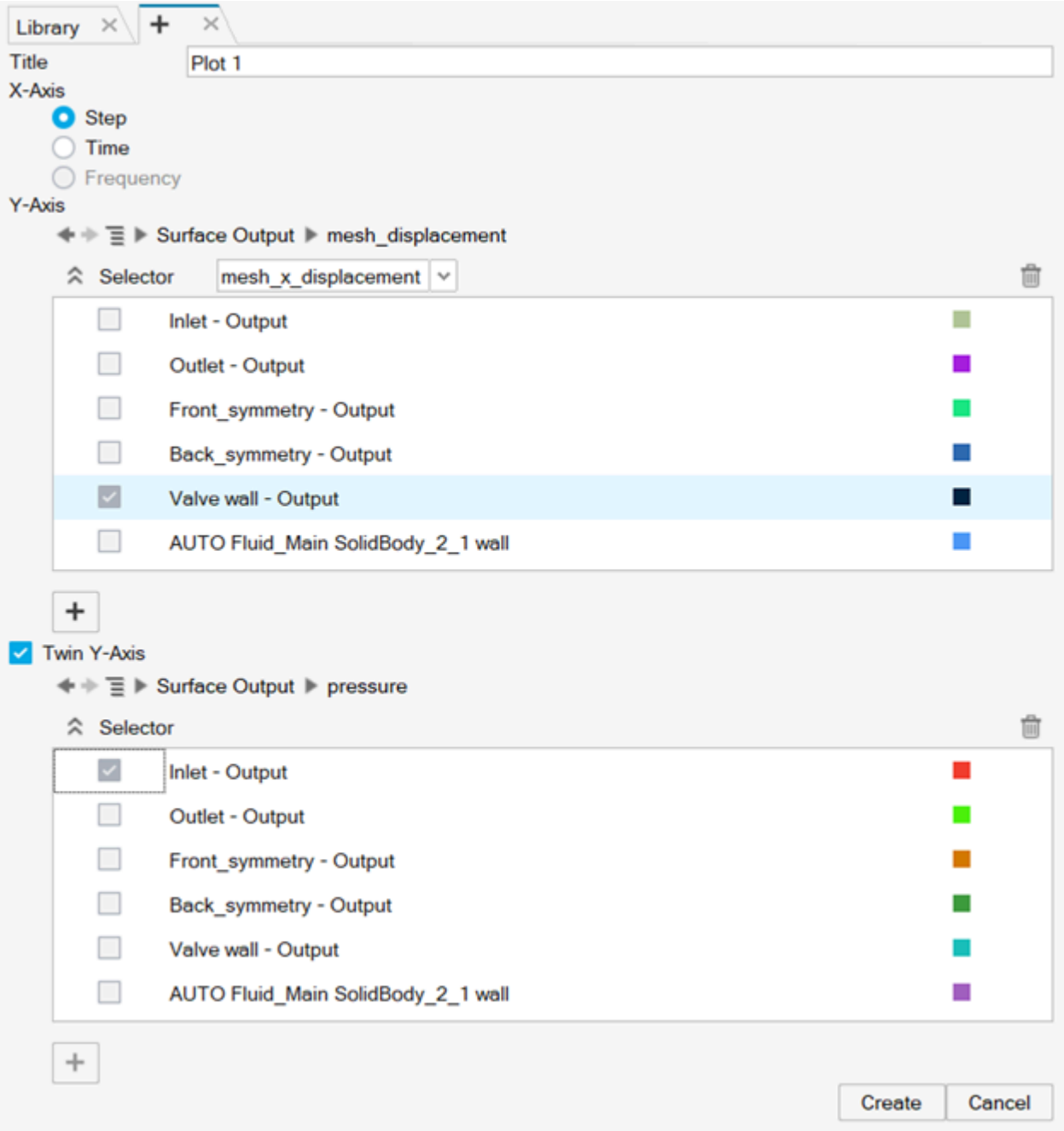


Figure 1184:

8. Click **Create**.

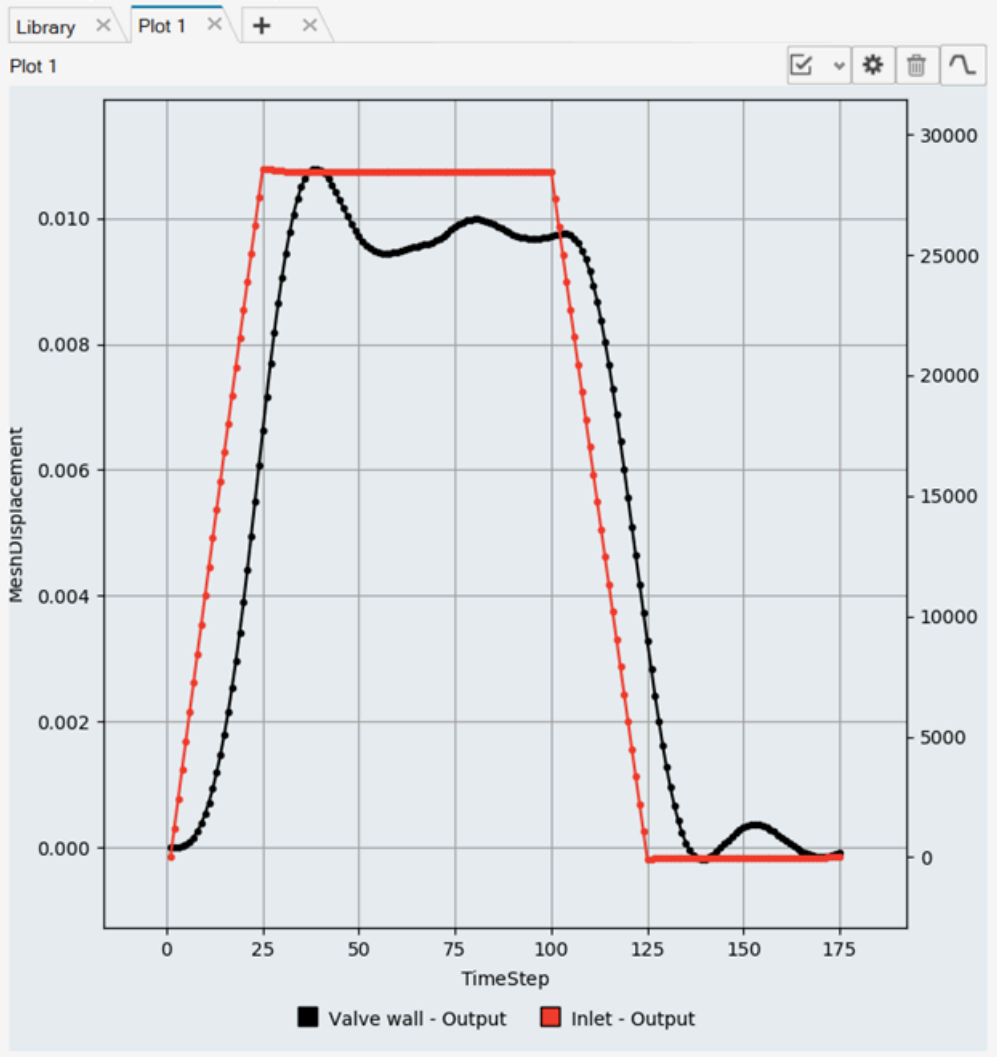


Figure 1185:

Note: The valve motion lags behind the pressure changes at the inlet. Note also the spike around the 37th time step. This represents the collision of the valve stem with the stop and the subsequent rebound of the valve.

Post-Process the Results with HM-CFD Post

Create an Animation of Velocity Magnitude

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve log file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

4. Click the **Top** face on the View Cube to align the model.

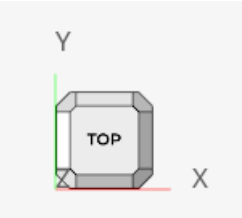


Figure 1186:

5. In the Post Browser, right-click on **Front_symmetry** and select **Isolate**.
6. Right-click on **Front_symmetry** once more and select **Edit**.
7. In the display properties microdialog, set the display to **velocity**.
8. Activate the **Legend** toggle and set the max velocity to `6.0`.
9. Click ≡ and set the Colormap Name to **Rainbow Uniform**.

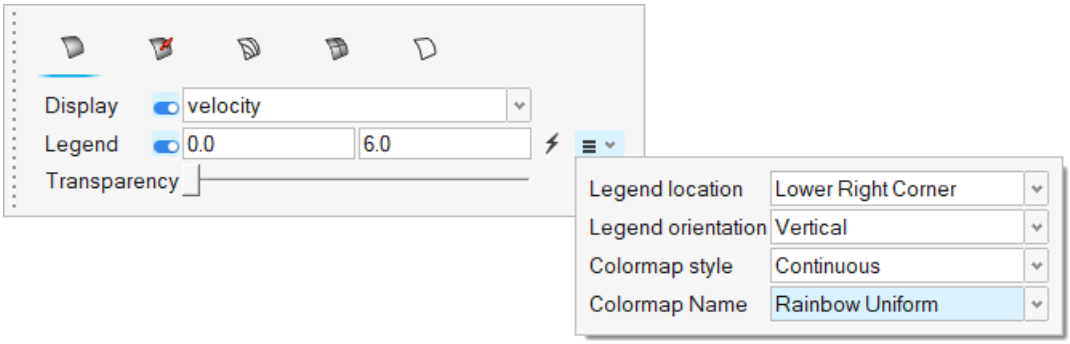


Figure 1187:

10. Click ✓ on the guide bar.
11. Click ▶ at the bottom of the modeling window to view a live animation of the flow.

Figure 1188:

12. Save the animation.
    a) Go to **File** > **Screen Capture** > **Advanced Capture**.
    b) Click ≡ on the toolbar.
    c) Uncheck **Include mouse cursor**.
    d) Set the frame rate to `50`.
    e) Click ⌖ on the toolbar then drag over the area you want to record.
    f) Click ◉ to start recording and the same button to stop recording.
    g) Name the file and save it.

Display Pressure Contours and Velocity Vectors on a Section Cut

1. Right-click in the modeling window and select **Show All**.
2. Click the **Slice Planes** tool.

Figure 1189:

3. Select the plane perpendicular to the z-axis.
4. Make sure the View Cube is aligned with the Top face.

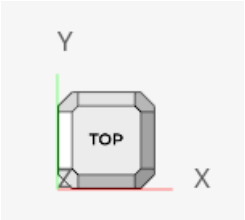


Figure 1190:

5. Click in the slice plane microdialog.
6. Turn off the **Display** in the Contour tab of the display properties microdialog.

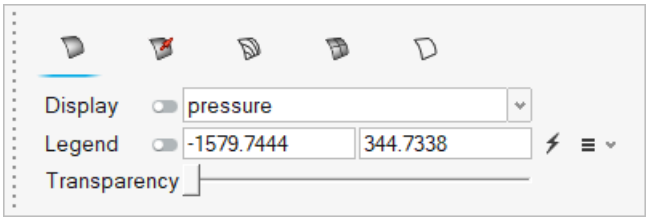


Figure 1191:

7. Click the **Vector Display** tab.
8. Turn the **Display** on, set the variable to **velocity**, and the value to `0.001`.
9. Set the Coloring to **pressure**.
10. Activate the **Legend** toggle and click to reset the range.
11. Click , set the Legend location to **Upper Center**, the Legend orientation to **Horizontal**, and the Colormap Name to **Rainbow Uniform**.

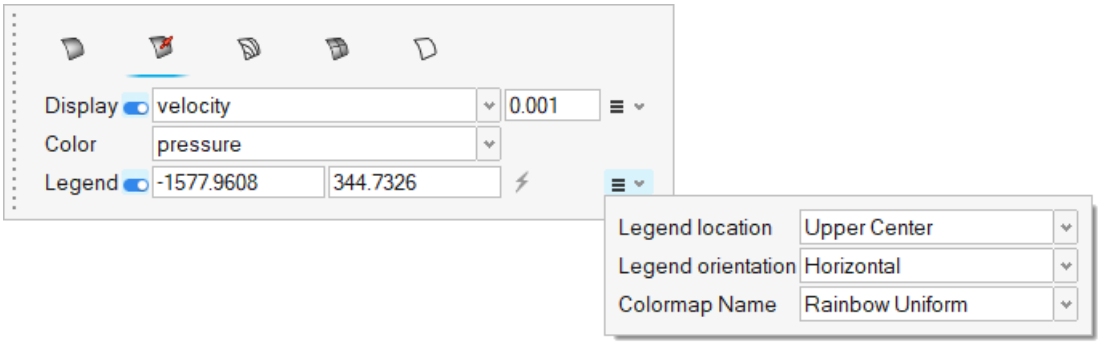


Figure 1192:

12. Click ✓ on the guide bar.

13. In the Post Browser, right-click on **Slice Plane 1** and select **Isolate**.

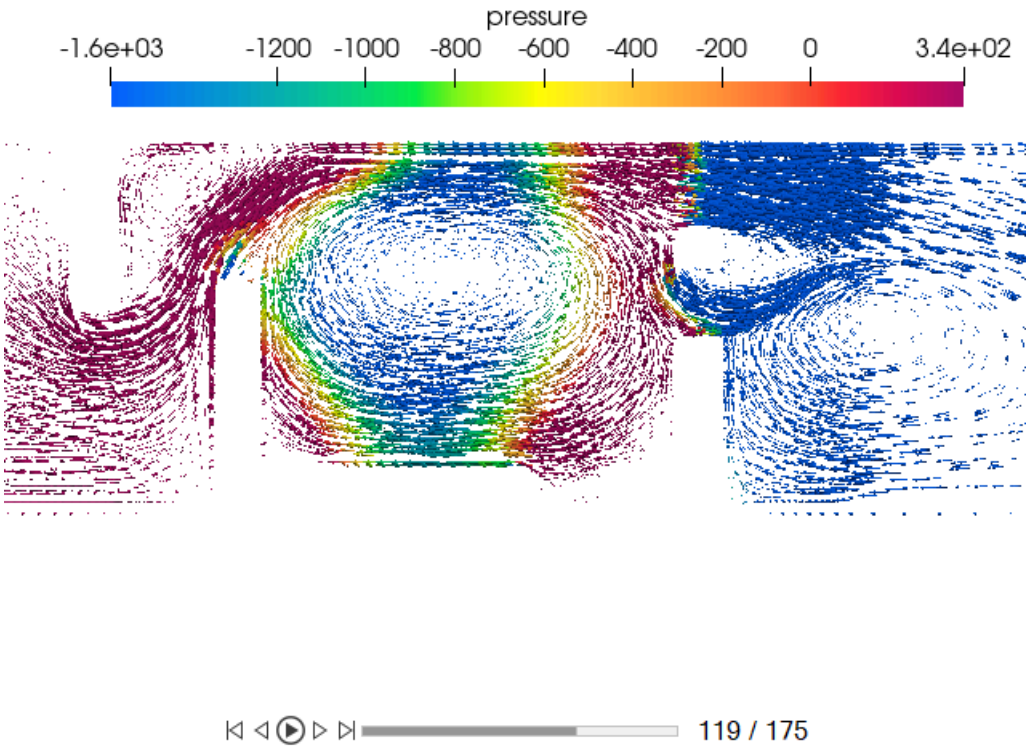


Figure 1193:

Summary

In this tutorial, you learned the basic workflow to set up a co-simulation using AcuSolve and MotionSolve. The tutorial introduced you to the steps involved in setting up external code communication between AcuSolve and MotionSolve using HyperMesh CFD, and then running the

simulation and post-process the results using HyperMesh CFD Post. You also learned how to create a vector plot on an existing contour plot on a cut plane.

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/95gy79h8l8

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5202 Flow Closing Valve

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/nj852bqe5u

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5400 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (P-FSI)

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/vibiujb2ox

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 5403 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Advanced

This tutorial provides the instructions for setting up, solving and viewing results for a simulation of a piezoelectric fluid harvester. In this simulation, a piezoelectric flow harvester is placed in a fluid flow channel. The harvester is attached to a cylinder mount which also acts as a bluff body causing vortices in the fluid flow. The interaction between the pressure fields generated by the vortices and the flow harvester structure is simulated in this tutorial. AcuSolve is used in conjunction with a structural solver to compute the structural displacement of the harvester using a direct-coupled fluid structure interaction (DC-FSI) approach. Arbitrary Lagrangian Eulerian (ALE) approach is used to compute the mesh deformation in the fluid domain as it interacts with the deforming structure.

The basic steps in any CFD simulation are shown in ACU-T: 2000 Turbulent Flow in a Mixing Elbow. The PFSI approach for a fluid-structure multiphysics problem has been discussed in the Piezoelectric Flow Energy Harvester PFSI tutorial. The following additional capabilities of AcuSolve are introduced in this tutorial:

- Set up a Direct Coupled FSI simulation (DC-FSI)
- AcuSolve/OptiStruct multiphysics coupling
- Setting up external code parameters to support coupling
- Assigning external code parameters to surfaces

In this tutorial you will do the following:

- Analyze the problem
- Start HyperMesh CFD and create a simulation database
- Import the mesh for the simulation.
- Set general problem parameters
- Set the appropriate boundary conditions for the coupling surfaces
- Run OptiStruct
- Run AcuSolve
- Monitor the solution with HyperMesh CFD
- Post-process the nodal output with HyperMesh CFD
- Create an animation for the transient simulation with HyperMesh CFD Post

Prerequisites

You should have already run through the introductory tutorial, ACU-T: 2000 Turbulent Flow in a Mixing Elbow. It is assumed that you have some familiarity with HyperMesh CFD and AcuSolve. You will also need access to a licensed version of AcuSolve.

The coupled structural solver used for this tutorial is another product, OptiStruct. Thus, to follow this tutorial you will also need access to a licensed version of OptiStruct. The corresponding OptiStruct setup for this tutorial is available in the OptiStruct tutorial manual.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T5403_Harvester_DCFSI.hm`

`slab_dcfsi.fem`

The file `ACU-T5403_Harvester_DCFSI.hm` stores the geometry information for the fluid portion of the model for this problem, and the file `slab_dcfsi.fem` is the OptiStruct input deck for this problem.

The color of objects shown in the modeling window in this tutorial and those displayed on your screen may differ. The default color scheme in HyperMesh CFD is "random," in which colors are randomly assigned to groups as they are created. In addition, this tutorial was developed on Windows. If you are running this tutorial on a different operating system, you may notice a slight difference between the images displayed on your screen and the images shown in the tutorial.

Note: This tutorial does not cover the steps related to geometry cleanup and meshing.

Analyze the Problem

An important step in any CFD simulation is to examine the engineering problem at hand and determine the important parameters that need to be provided to AcuSolve. Parameters can be based on geometrical elements, such as inlets, outlets, or walls, and on flow conditions , such as fluid properties, velocity.

The schematics of the problem is shown in Figure 1194. The CFD model consists of a cylindrical body and a cantilever beam. This cylindrical body generates vortex shedding. This vortex shedding creates a zone of alternating asymmetric pressure distribution on either side of the beam. Such an alternating pressure distribution exerts an oscillating force on the beam, creating a sustainable oscillating vibration in the beam.

The fluid solver does not require the solid body to be modeled. However, the structural solver will solve for the structural deformation using the fluid flow and pressure field which will be acting on the surface of the structure. The information required by the structural solver for structure displacement calculation will be passed on to it by the fluid solver. The structural solver on the other hand, will pass on the displacement information to the fluid solver. The fluid solver will then use this displacement information to calculate the flow field at the next time step, which will then be used by the structural solver to update the displacement information. This back and forth exchange of information between the fluid and structural solver will take place at each time step, and continue until one of the solvers signals the end of simulation. Figure 1195 shows the arrangement of the beam with its various layers.

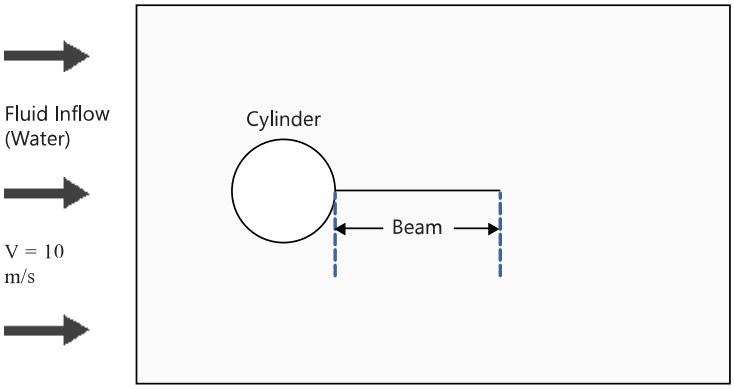


Figure 1194: Schematic of the Problem

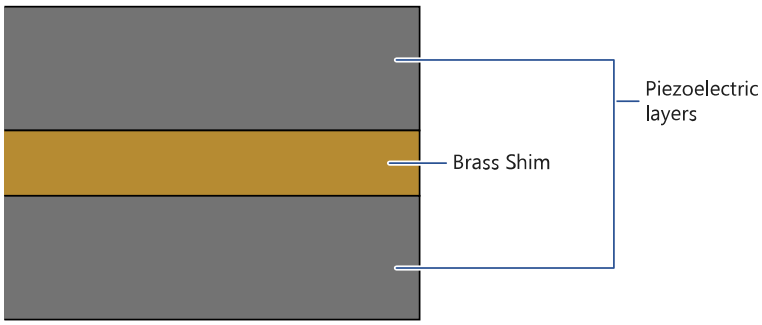


Figure 1195: The Beam with its Various Layers

Fluid Structure Interaction

Fluid Structure Interaction is the interaction between a fluid flow and a deformable solid structure in contact with this flow. An FSI problem can be an external or an internal flow problem. The fluid flow can be external with the solid body immersed in the flow, for example, a windmill blade in open atmosphere. The fluid flow can also be internal with the solid body enclosing the flow, for example, fluid flow inside a deformable pipe. In both cases, the principle behind solving the problem remains the same. When a fluid flow encounters a structure, fluid pressure exerts a stress on the solid body that can lead to deformations in the structure. The magnitude of the deformation depends on the stiffness of the structure material and the magnitude of pressure force exerted by the fluid. The deformation in the structure shape then leads to altering of the flow characteristics in vicinity of the structure.

An FSI problem is thus a multiphysics problem, with the coupled laws of structural and fluid mechanics affecting the solution. There are two widely used approaches with which this coupling is achieved:

1. Practical-FSI (P-FSI): The structure is reduced in the modal space and coupled to the fluid domain through interface nodes. The coupling between the solvers happens in a single pass itself. Structural behavior is limited to be linear in a P-FSI simulation.
2. Direct coupling (DC-FSI): The coupling is a co-simulation between the structural and the fluid solver, with each solver stepping through time simultaneously and iterating to equilibrium in each time step.

In case the deformations in the structure are large enough to alter the fluid flow significantly, the DC-FSI co-simulation approach should be used. With this approach, as the fluid flow and pressure fields

affect the structural deformations, and the structural deformations affect the flow and pressure, the information about these effects is exchanged between the solvers in real time.

Given the difference in coupling methodology, it is likely that slightly different results will be observed when a same problem is solved using P-FSI and DC-FSI approaches. The choice of the approach that should be used shall depend on the problem and the available resources. As mentioned above, the P-FSI approach should be limited to the cases when displacements in the structure are small, and the structural behavior can be approximated to be linear. For all other cases, DC-FSI should be preferred. However, DC-FSI simulations incur a higher computational resources cost. With this consideration, P-FSI simulation can also be used as a preliminary test simulation before a DC-FSI simulation is carried out.

FSI can be stable or oscillatory. In a stable FSI, the deformed shape of the structure will not change with time, unless the flow changes as well. In an oscillatory FSI, once the structure is deformed, it will try to return to its non-deformed state and then the whole deformation process repeats itself.

Mesh Motion Approaches in AcuSolve

Many simulations require deformation of the domain with time. AcuSolve provides two approaches for handling dynamic meshes:

1. Arbitrary Lagrangian Eulerian (ALE)
2. Interpolated Mesh Motion.

Arbitrary Lagrangian Eulerian (ALE)

ALE is an approach for mesh motion in which the computational nodes are moved arbitrarily with the aim of optimizing the element quality. An additional Partial Differential Equation (PDE) is solved to arrive at the appropriate mesh position. ALE is capable of handling complex arbitrary motions and is therefore the most general approach in simulating moving mesh problems. Generality comes with additional computational cost because of the extra PDE to be solved. For simpler motions like 1D or 2D motions faster approaches are available which include interpolated mesh motion, general specified motions, nodal boundary conditions based approaches.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1196:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5403_Harvester_DCFSI.hm` and click **Open**.
4. Click **File** > **Save As**.

5. Create a new directory named `Harvester` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Harvester` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1197:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

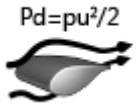
   

   Figure 1198:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Verify that the Incompressible option is selected under Single phase flow.
   b) Set the Time marching to **Transient**.
   c) Set the Time step size to `0.01` and the Final time to `0.0`.
   d) Select **Spalart-Allmaras** as the Turbulence model.

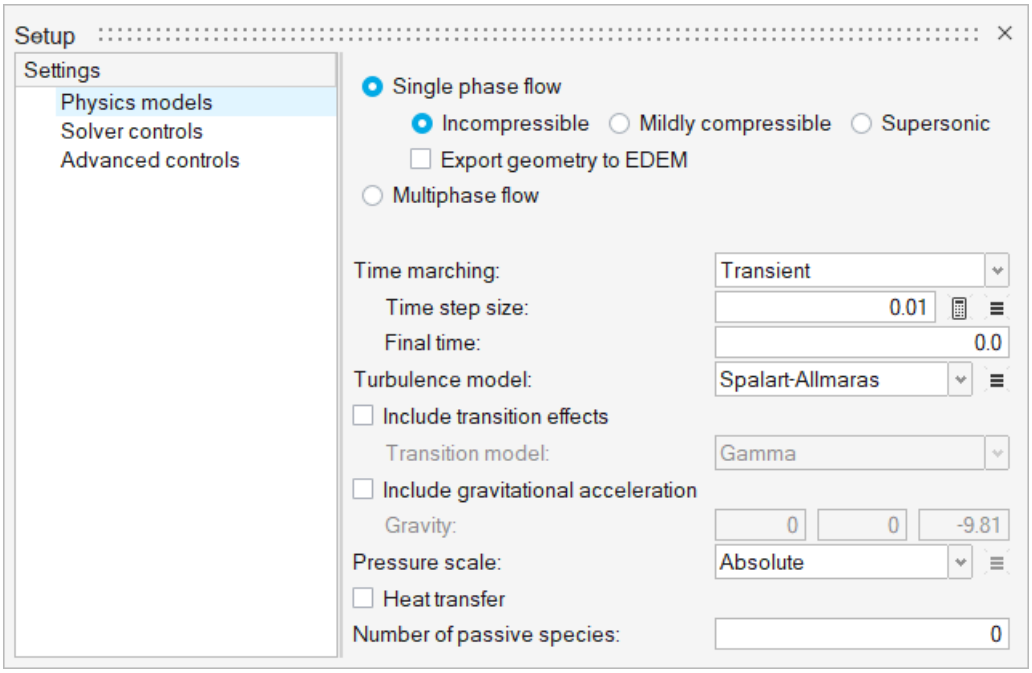


Figure 1199:

3. Click the **Solver controls** setting.

a) Set the Minimum stagger iterations to `3`.

b) Set the Maximum stagger iterations to `20`.

c) Set the Transient maximum steps to `300`.

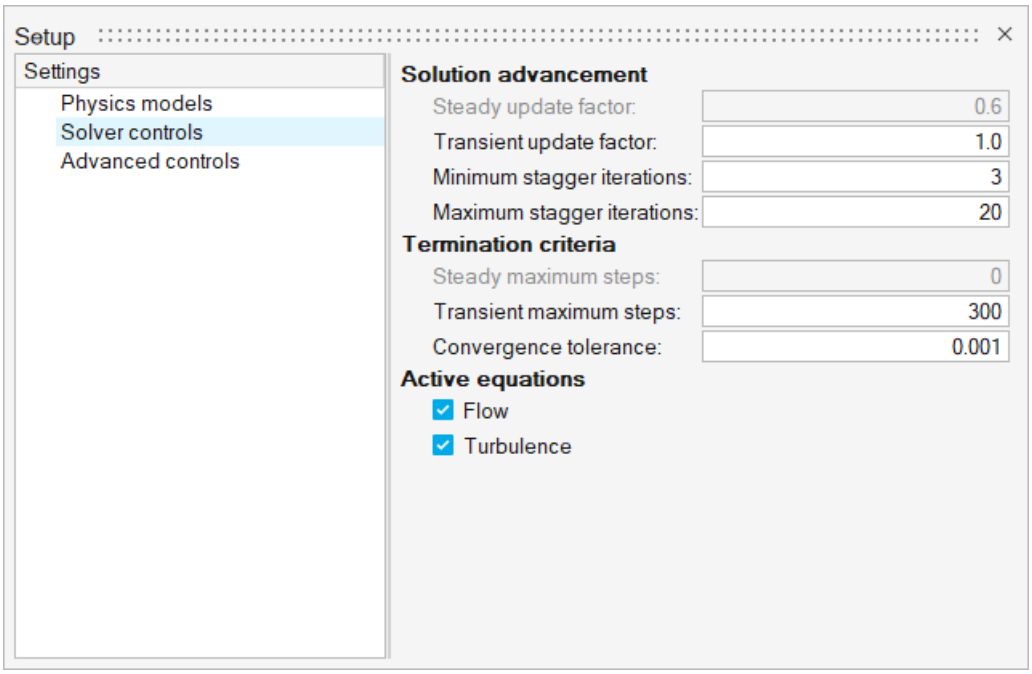


Figure 1200:

4. Close the dialog and save the model

Create a Multiplier Function

The force acting on the beam, due to the flow, will be ramped gradually over the first few time steps. After these first few time steps, the complete force from the fluid will be transferred to the beam without any ramping. This will be achieved using a Multiplier Function. In the next few steps you will create a linear multiplier function which will later be assigned as a force multiplier function for load acting on the beam.

1. From the Flow ribbon, click the arrow next to the Setup tool set, then select **Multipliers**.

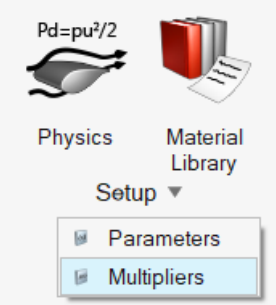


Figure 1201:

The **Multiplier Library** opens.

2. Click + to add a new multiplier.
3. In the multiplier creation dialog, change the name of the multiplier function to `ForceRamp` by clicking on the top-left corner.
4. Set the Type to **Piecewise Linear** and the Variable to **Time Step**.
5. Verify that the Evaluation is set to Time Step.
6. Enter the table values according to the figure below.

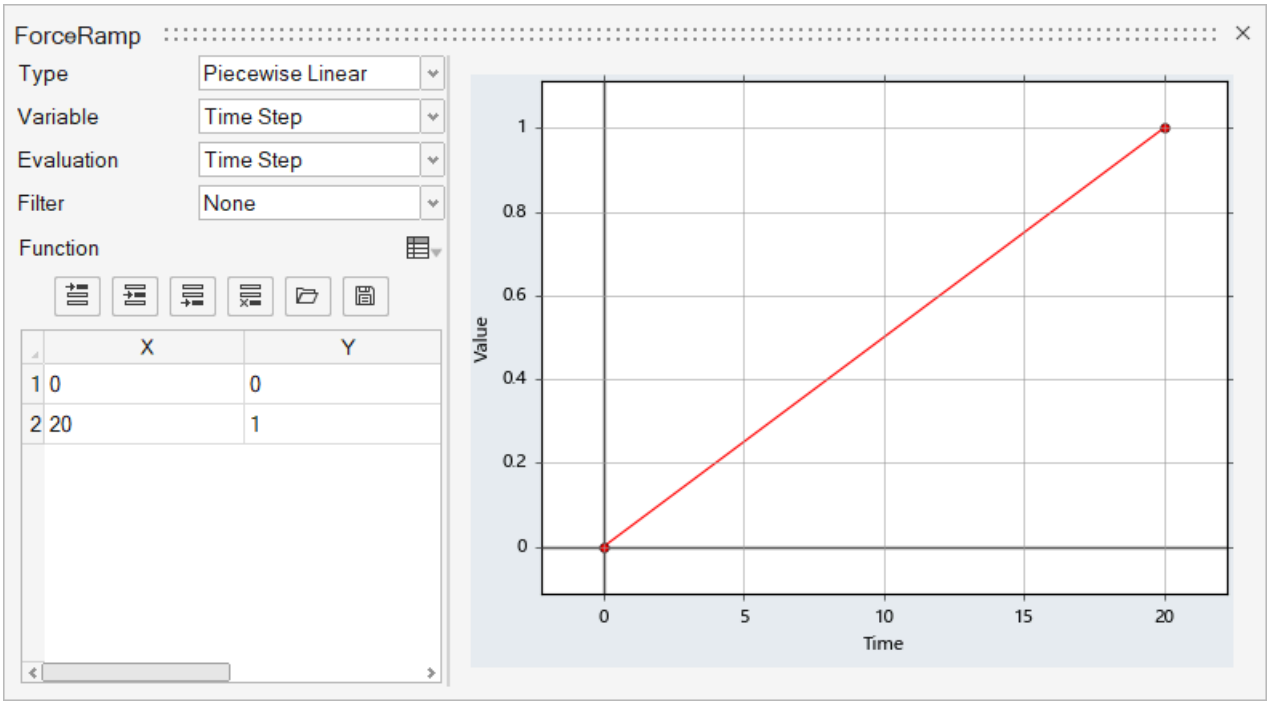


Figure 1202:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1203:

2. Select the model body.
   The entire solid is highlighted.
3. Select **Water** from the drop-down in the microdialog.

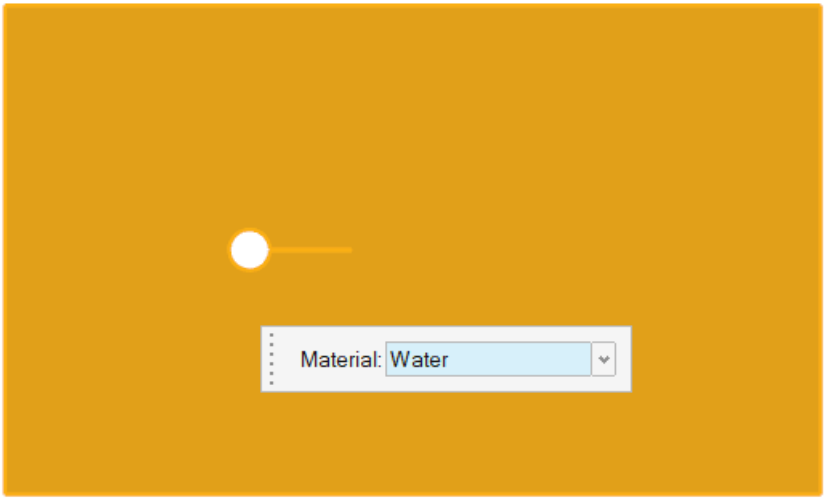


Figure 1204:

4. On the guide bar, click ✓ to execute the command and exit the tool.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 1205:

2. Select the left-most face on the negative x-axis, as shown in the figure below.
3. In the microdialog, choose **Cartesian** for the inflow velocity type and set X velocity to `10` m/s.

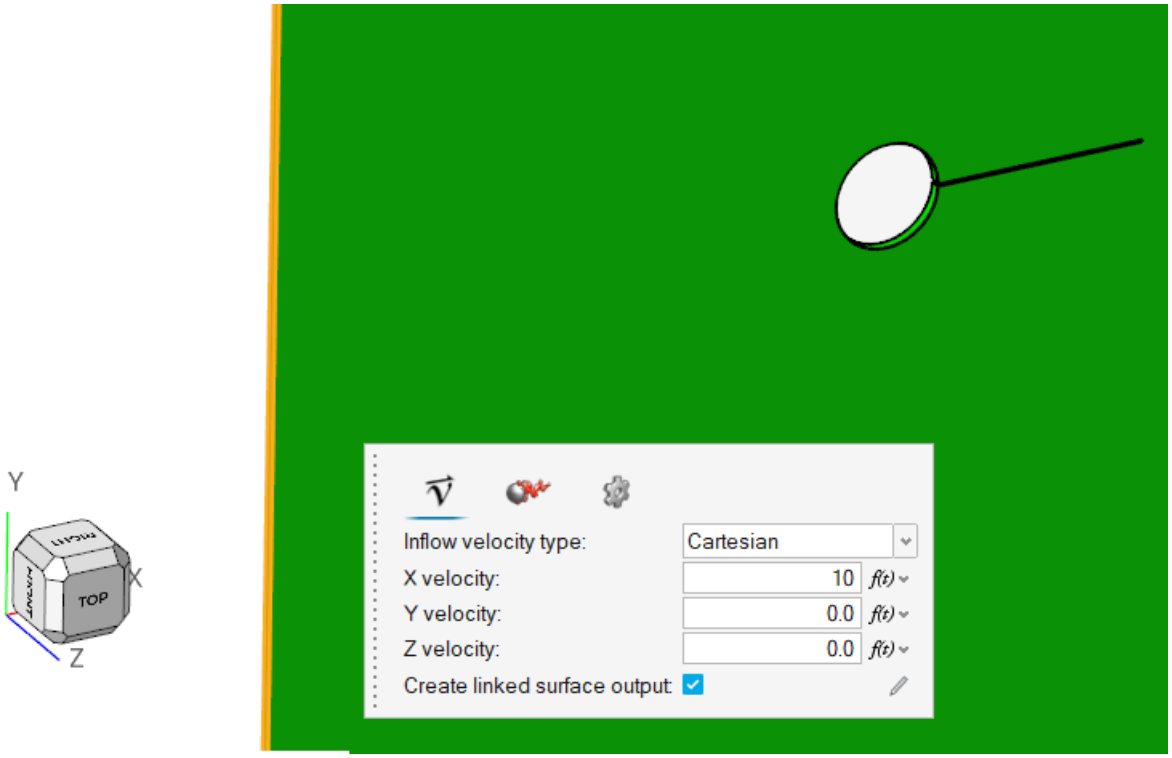


Figure 1206:

4. Click the **Turbulence** tab in the microdialog. Choose **Direct** for the Turbulence input type and set Eddy viscosity to `1e-5` m^2/sec.

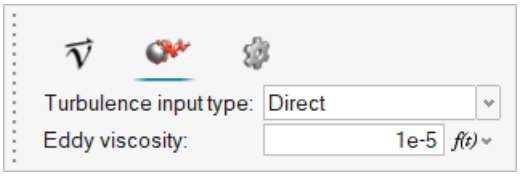


Figure 1207:

5. Click ✓ on the guide bar.
6. Click the **Outlet** tool.

Figure 1208:

7. Select the right-most face on the positive x-axis, as shown in the figure below
8. In the microdialog, verify that both the static pressure and the pressure loss factor are set to `0`.

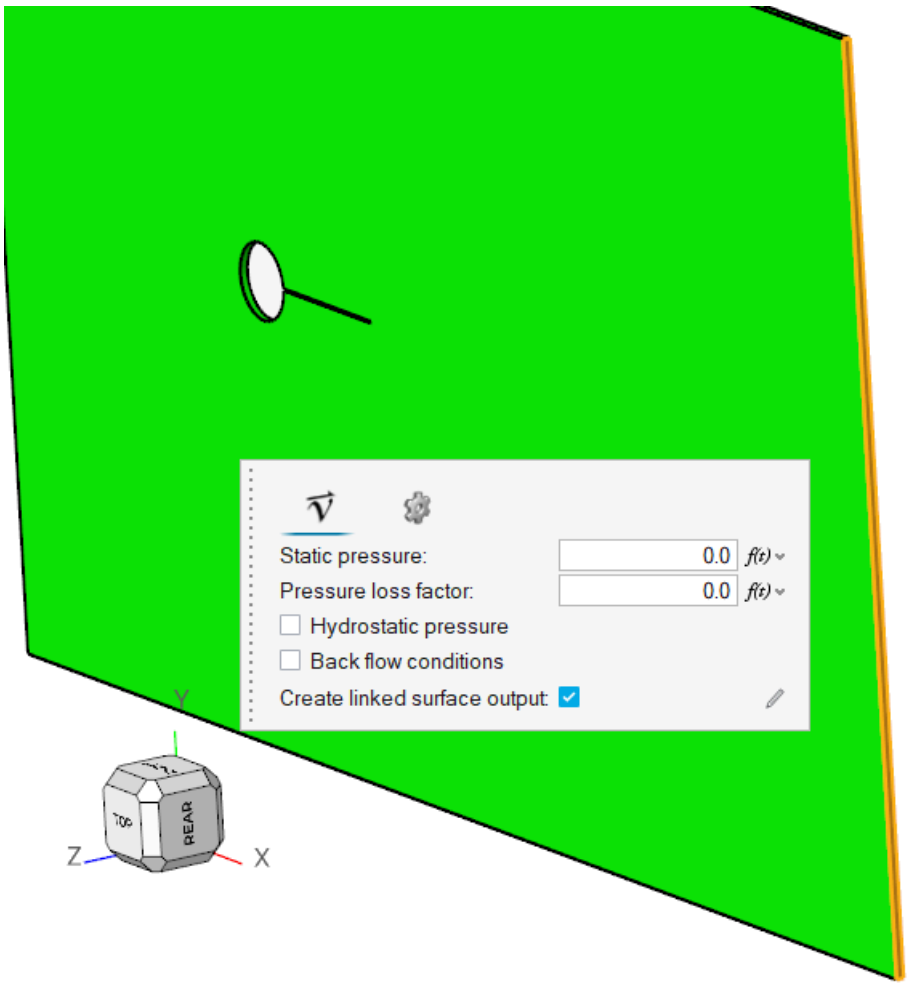


Figure 1209:

9. Click ✓ on the guide bar.

10. Click the **Slip** tool.

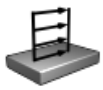


Figure 1210:

11. Select the right-most face on the negative z-axis, as shown in the figure below.

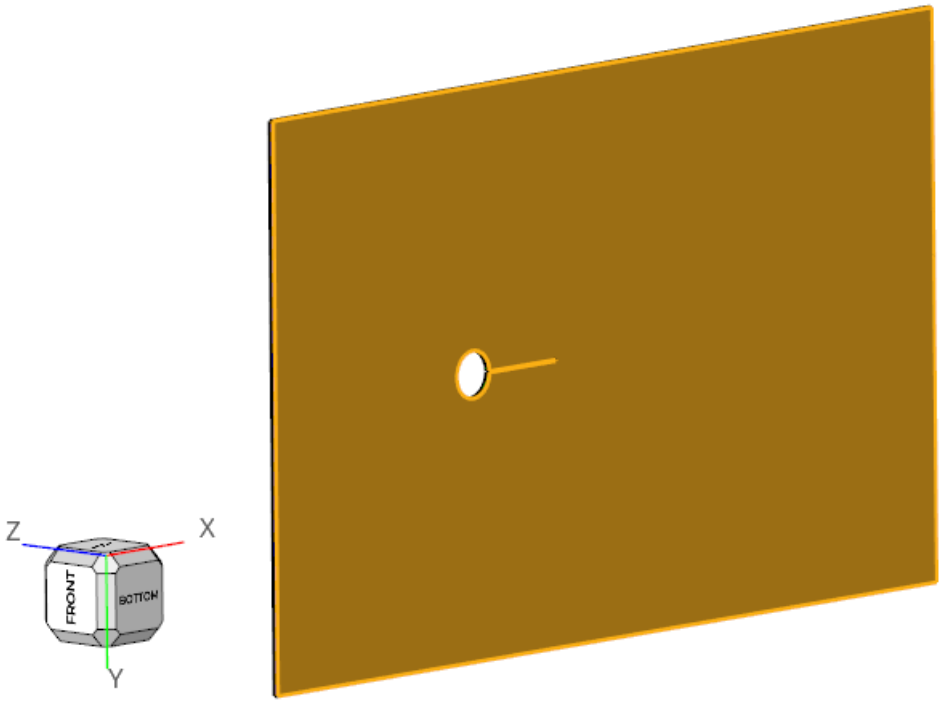


Figure 1211:

12. On the guide bar, click ▸ to execute the command and remain in the tool.

13. In the Boundaries legend, double-click **Slip**, rename it to `z_neg`, and press **Enter**.

14. Rotate the model and select the opposite face.

15. Click ▸ on the guide bar.

16. In the Boundaries legend, rename Slip to `z_pos`.

17. Select the right-most face on the positive y-axis, as shown in the figure below.

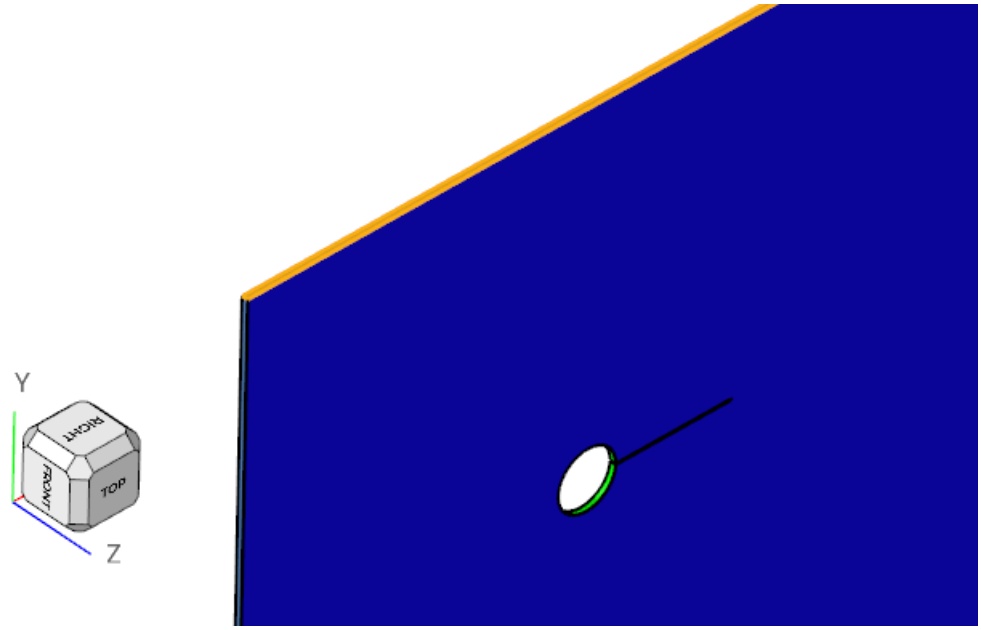


Figure 1212:

18. Click ▸ on the guide bar.

19. In the Boundaries legend, rename Slip to `y_pos`.

20. Rotate the model and select the opposite face.

21. Click ▸ on the guide bar.

22. In the Boundaries legend, rename Slip to `y_neg`.

23. In the Boundaries legend, right-click **Default Walls** and select **Isolate**.

24. Click the **No Slip** tool.

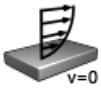


Figure 1213:

25. Select the cylinder face, as shown in the figure below.

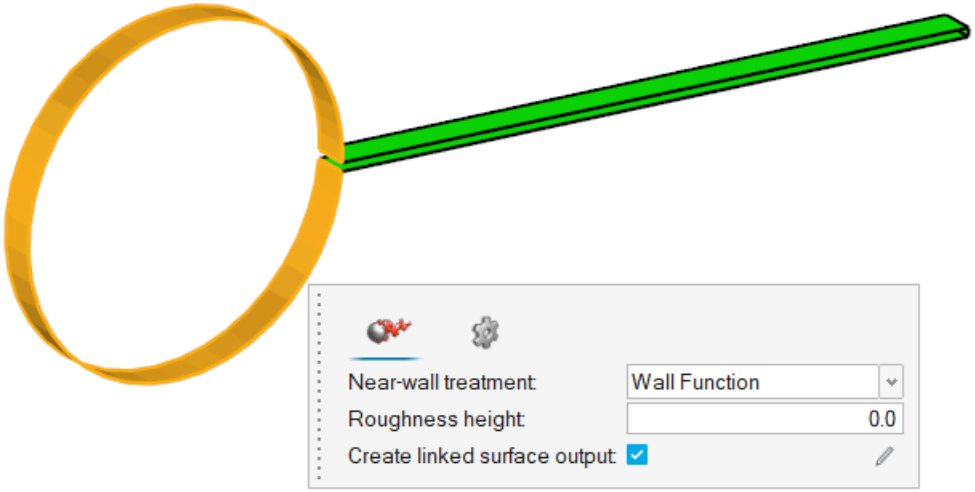


Figure 1214:

26. Click ▸ on the guide bar.

27. In the Boundaries legend, rename Wall to `cylinder`.

28. Right-click **cylinder** and select **Hide**.

Set Up Motion

Define the Mesh Motion Type

In the introduction it was discussed that the structural solver is responsible for calculating the displacement and passes on the calculated displacements to AcuSolve. In the following steps, you will set up the connection between AcuSolve and the external code, which in this case is OptiStruct.

1. From the Motion ribbon, click the **Settings** tool.

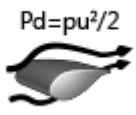


Figure 1215:

The **Setup** dialog opens.

2. Change the Mesh motion to **Arbitrary**.

3. Click the **Solver controls** setting and activate **Mesh deformation**.

4. Click the **External code** setting and activate the **External code** option.

5. Set the External Solver to **OptiStruct** and the Communication type to **Socket**.
6. Deactivate the **External socket** option and set the Socket host to **localhost**.
7. Set the Socket port to `48002`.

   The socket port depends on the structural solver. 48002 is the OptiStruct default and is thus used for this example. Refer to the manual of the structural solver you are using to determine the correct port number to be entered here.
8. Activate **Scale fields** and select **ForceRamp** for Multiplier function.

   This will gradually ramp up the forces transmitted from the external code over the first few steps as specified by the multiplier function (20 time steps in this case).

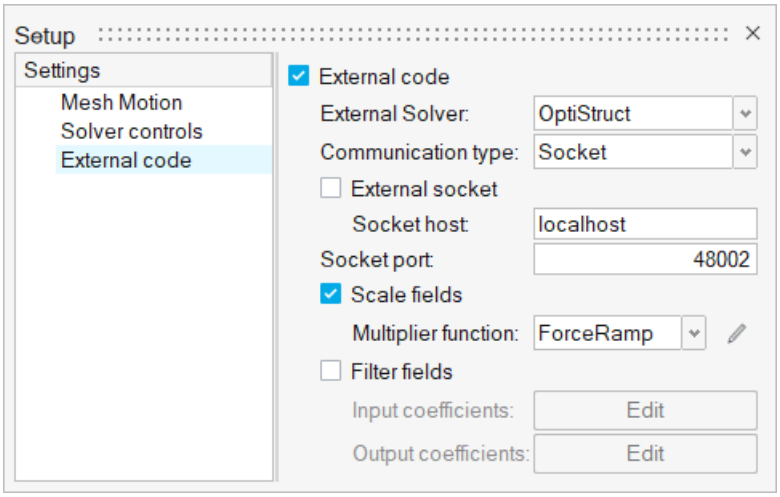


Figure 1216:

9. Close the dialog and save the model.

Define the Mesh Displacement Boundary Conditions

1. From the Motion ribbon, click the **External Surface** tool.

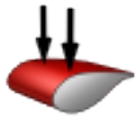


Figure 1217:

2. Select the beam faces, as shown in the figure below.
3. In the microdialog, select **Factor** for the Gap method and set Gap factor to `0`.

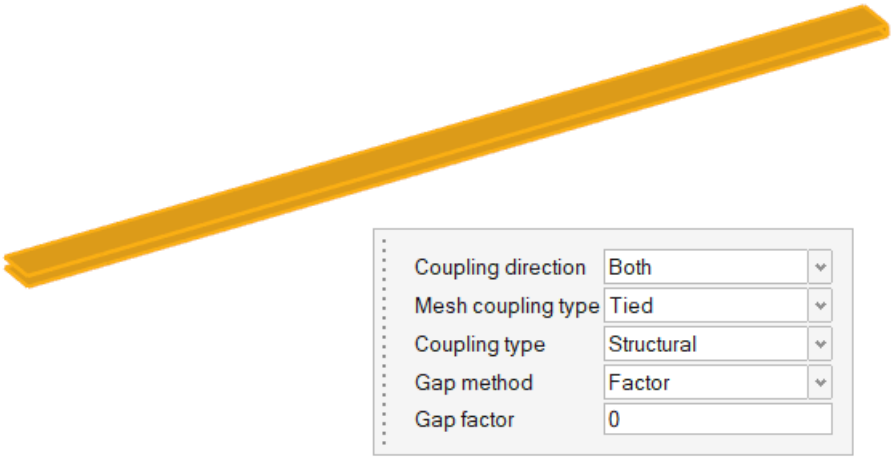


Figure 1218:

4. In the External Code Surface legend, rename the External surface to `beam`.
5. Click ✓ on the guide bar.
6. Click the **No Constraints** tool.

Figure 1219:

7. Select the external surfaces of **beam**.
8. Click ✓ on the guide bar.
9. Press **A** to show all surfaces.
10. Click the **Planar Slip** tool.

Figure 1220:

11. Select the right-most face on the negative z-axis, as shown in the figure below

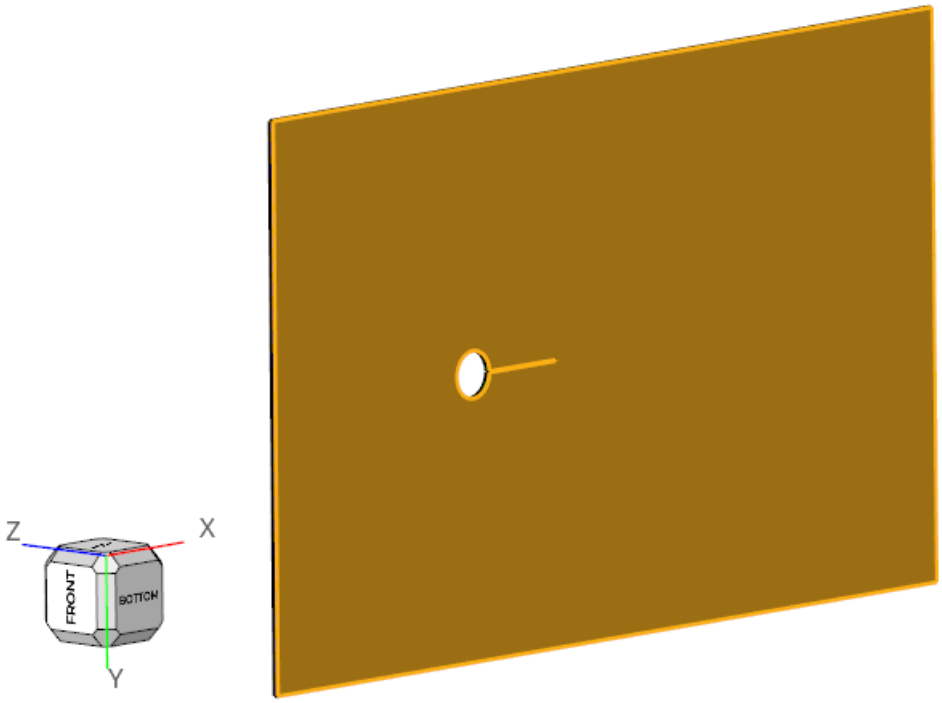


Figure 1221:

12. Click ▸ on the guide bar.
13. In the Mesh Motion legend, rename Planar Slip to `z_neg`.
14. Rotate the model and select the opposite face.
15. In the Mesh Motion legend, rename Planar Slip to `z_pos`.
16. Click ✓ on the guide bar.
17. Turn on the display of all surfaces and save the model.

Define Time History Output Points

Time History Output commands enable you to extract the nodal solution at any point within the domain. In this simulation, it would be interesting to observe the displacement at the tip of the cantilever beam.

1. Form the Solution ribbon, click the body of the **Probe** tool.

Figure 1222:

2. Select the top point of the beam tip end on the right-most face of the negative z-axis, as shown in the figure below.
3. In the microdialog, set the Point Radius to `0.01`.

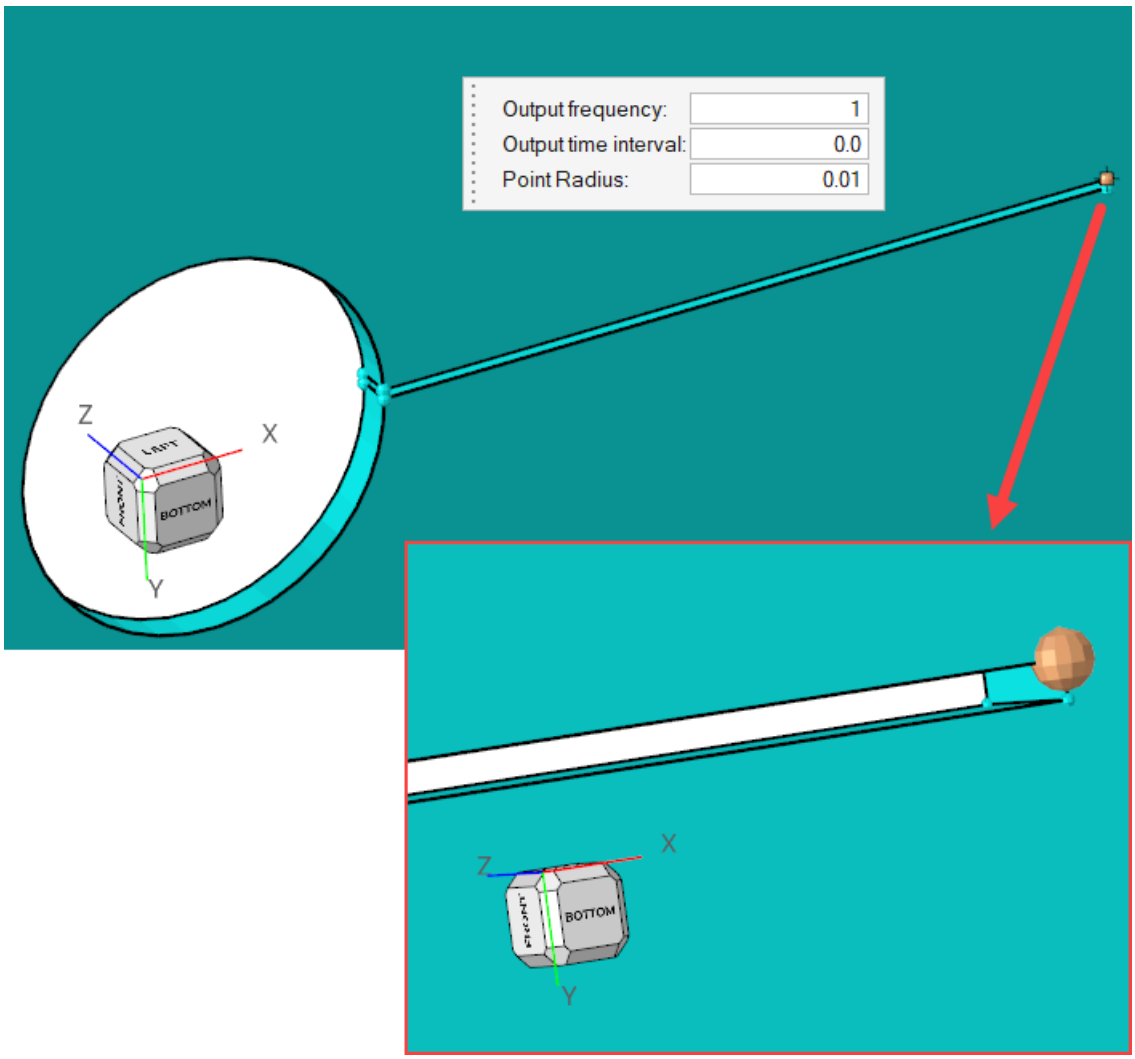


Figure 1223:

4. In the Point Probes legend, rename Point Probe to `Tip_Monitor_Point`.
5. Click ✓ on the guide bar.

Define Nodal Outputs

The nodal output command specifies the nodal output parameters; for instance, output frequency, number of saved states, and so on.

1. From the Solution ribbon, click the **Field** tool.

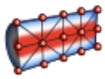


Figure 1224:

The **Field Output** dialog opens.

2. Under solution variables:
   a) Activate **Write initial conditions**.
   b) Activate **Write results at time step interval**.
   c) Set the Time step interval to `2`.

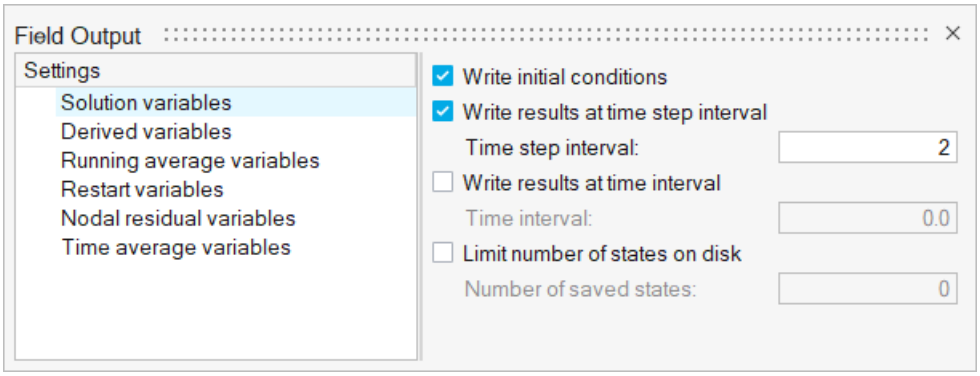


Figure 1225:

3. Close the dialog and save the model.

Run OptiStruct

1. Start OptiStruct from the Windows Start menu by clicking **Start** > **Altair <version>** > **Compute Console**.
2. Click 🗁 next to Input file(s).
3. Browse to the directory where you saved the OptiStruct model file.
4. Select the `slab_dcfsi.fem` file.
5. Activate **Use solver control**.

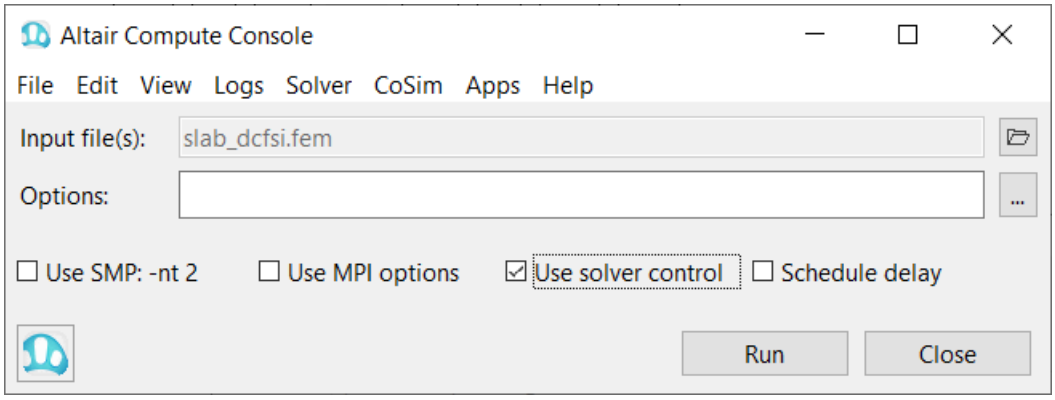


Figure 1226:

6. Click **Run** to launch OptiStruct.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1227:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Deactivate the **Automatically define pressure reference** option.
5. Expand **Default initial conditions** and deactivate **Pre-compute flow**.
6. Set the x velocity to `10` m/s.
7. Set the eddy viscosity to `1e-05` m^2/sec.
8. Leave the remaining options as default and click **Run** to launch AcuSolve.

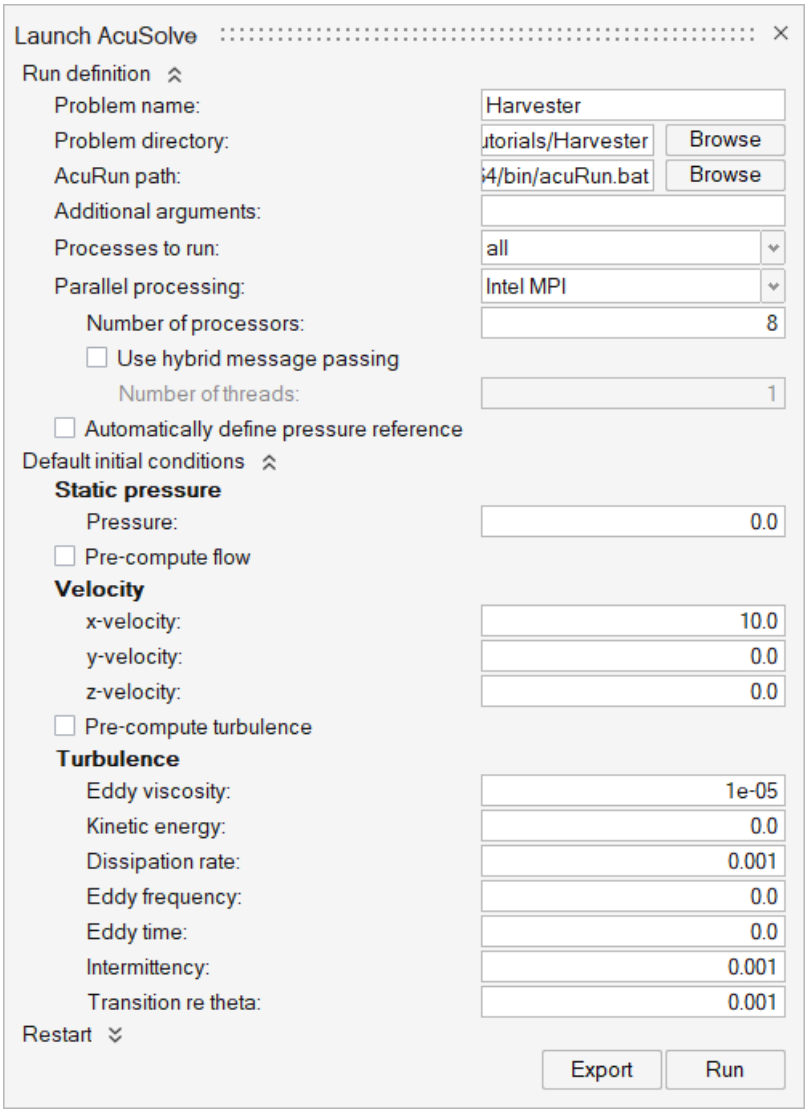


Figure 1228:

9. Right-click the solver job in the **Run Status** dialog and select **View log file**.
   As the solution progresses, progress is reported in this window.

   When AcuSolve is started, it will listen to the socket port specified while setting up the external code coupling for an initial communication from OptiStruct. You must make sure to start the OptiStruct run which will initialize the socket port for communication. Once the connection is established, the solvers will alternatively send and receive data through the simulation steps.

Plot Time History

1. Right-click the solver job in the **Run Status** dialog and select **Plot time history**.
2. In the **Plot Manager** dialog, click + to create a new plot.
3. Expand **Output Time History** > **Geometry** then select **mesh displacement**.

4. Set Components to **y**.
5. In the Parts field, choose **node 1** under Tip Monitor Point to displace the time history of the beam end.

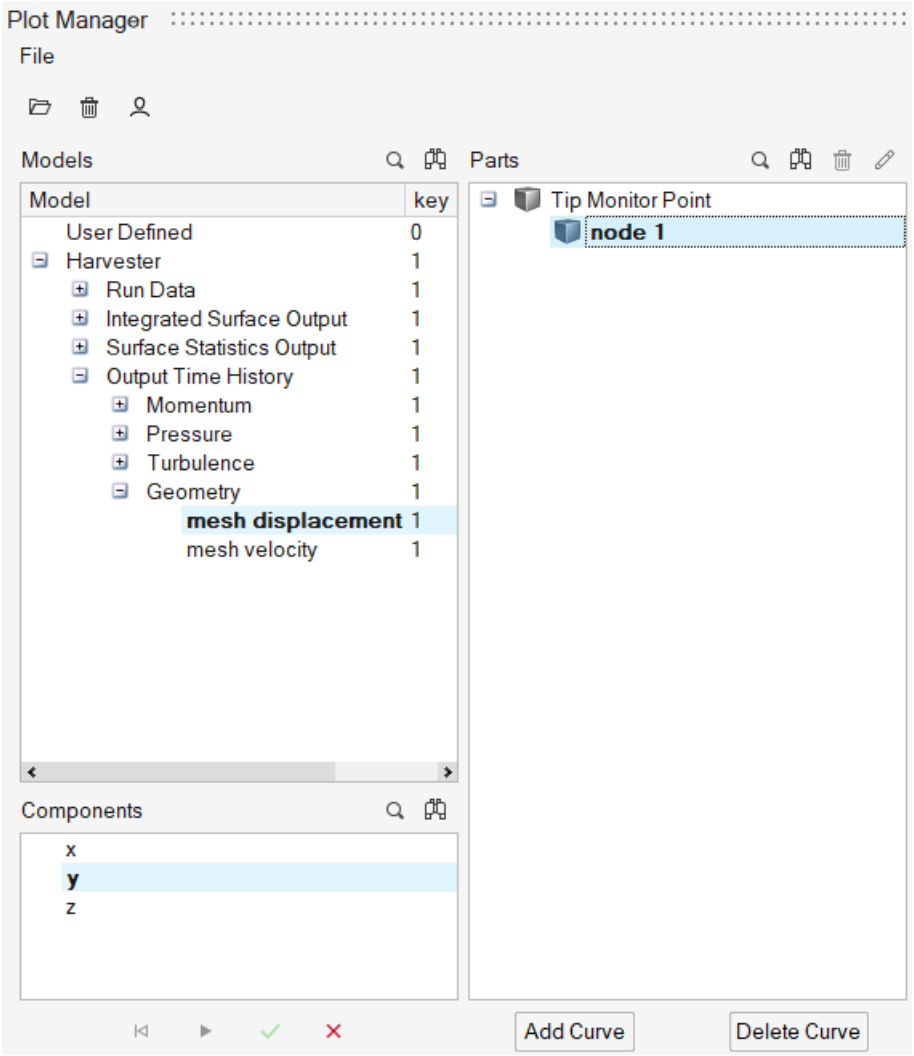


Figure 1229:

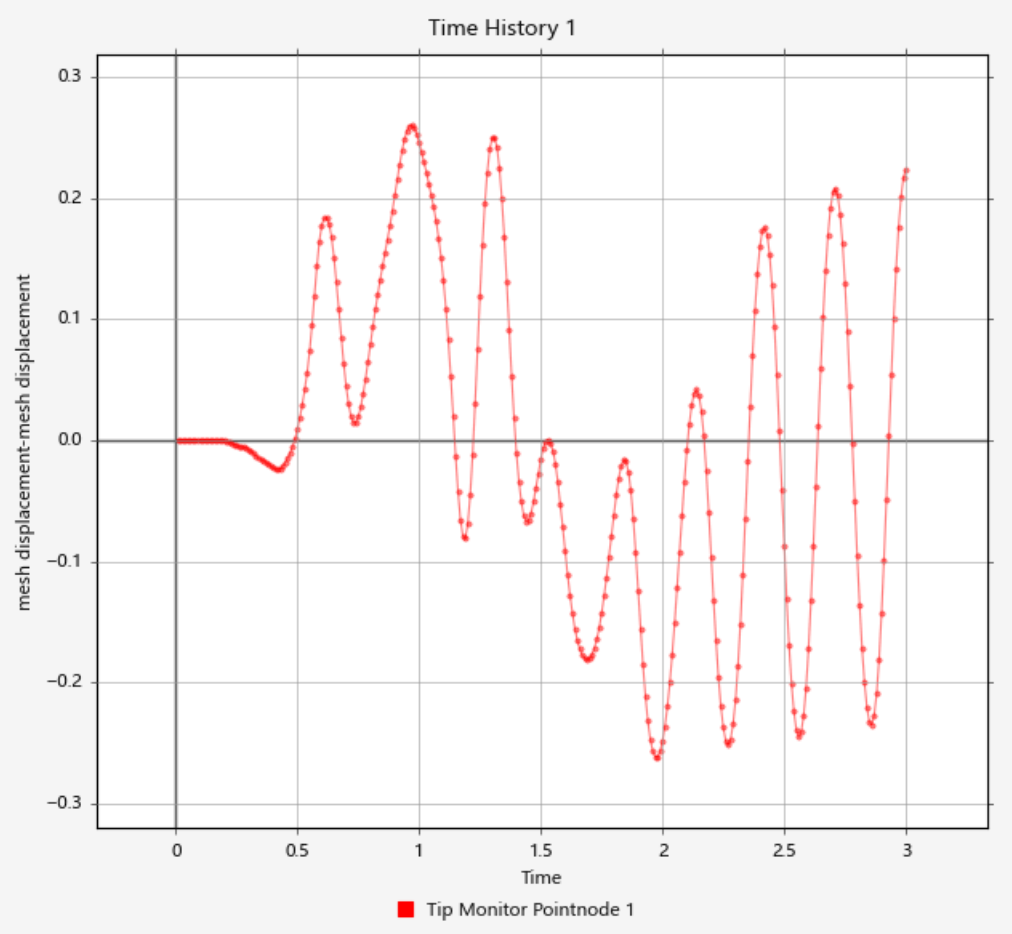


Figure 1230: Mesh_y_displacement plot at beam tip

The plot above shows the displacement of the tip of the beam due to the fluid forces as the beam interacts with the flow.

Post-Process the Results with HM-CFD Post

In this step, you will create an animation of pressure.

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.

   You can also import the results into HyperMesh CFD Post by right-clicking a solver job in the **Run Status** dialog and selecting **Visualize results**.
4. Click the **Bottom** face on the View Cube to align the model.

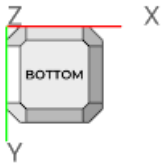


Figure 1231:

5. Right-click the **z_pos** boundary in the Post Browser and select **Isolate**.
6. Move the slide bar at the bottom of the modeling window to 74/151.

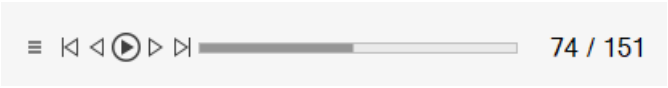


Figure 1232:

7. Right-click **z_pos** again and select **Edit**.
8. In the display properties microdialog, set the display to **pressure**.
9. Activate the **Legend** toggle and set the upper and lower bounds to `-65000` Pa and `55000` Pa, respectively.
10. Click ≡ and set the Colormap Name to **Rainbow Uniform**.

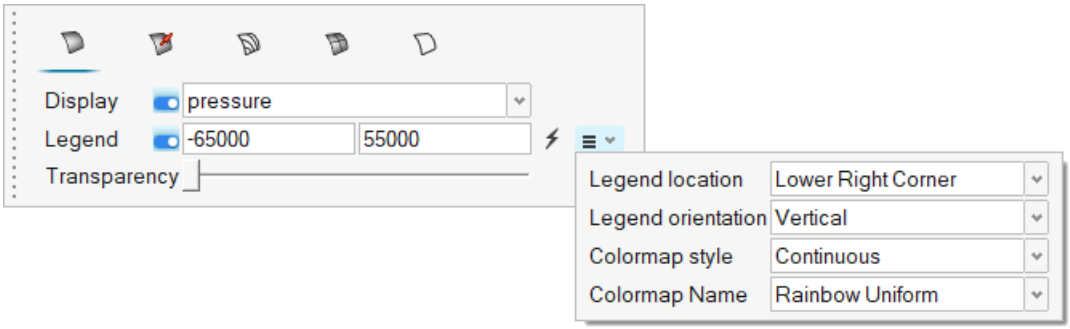


Figure 1233:

11. Click ✓ on the guide bar.
12. Click ▶ on the Animation toolbar to view a live animation of the flow.

Figure 1234:

13. Save the animation.
    a) Go to **File** > **Screen Capture** > **Advanced Screen Capture**.
    b) Click ≡ and uncheck **Include mouse cursor**.
    c) Set the frame rate to `24` and select **GIF** as the video setting.
    d) Click ⌖ on the toolbar then drag over the area you want to record.

e) Click ◉ to begin recording and the same icon to stop recording.

f) Name the file and save it.

Summary

In this AcuSolve tutorial, you successfully set up and solved an FSI problem using the Direct Coupling, or DC-FSI approach. DC-FSI approach is a co-simulation approach where both the structural and the fluid solvers run simultaneously, and exchange information at each time step of the simulation. The fluid solver passes the flow and pressure information to the structural solver, which are used to determine the displacements in the structure. The structural solver then passes the displacement information to the fluid solver, which then recalculates the flow and pressure field. Both the solvers thus update and exchange the results of the simulation to achieve the solution. You started the tutorial by creating a database in HyperMesh CFD and setting up the basic simulation parameters. Then, you defined the parameters for setting up the connection between AcuSolve and the structural solver, and generated a solution with co-simulation. Results were post-processed in HyperMesh CFD, where you generated an animation of the beam's displacement as it interacted with the fluid flow.

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/jygyblyrgw

Additional Resources

For more tutorials, visit the SimLab Learning Center.

Particle Tracing

6

This chapter covers the following:

- ACU-T: 6000 Static Mixer Simulation – AcuTrace (p. 713)
- ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM (p. 743)
- ACU-T: 6101 Particle Separation in a Windshifter using AcuSolve - EDEM Unidirectional Coupling (p. 776)
- ACU-T: 6102 Particle Separation in a Windshifter using AcuSolve - EDEM Bidirectional Coupling (p. 818)
- ACU-T: 6103 AcuSolve - EDEM Bidirectional Coupling with Heat Transfer (p. 863)
- ACU-T: 6104 AcuSolve - EDEM Bidirectional Coupling with Non-Spherical Particles (p. 901)
- ACU-T: 6105 Single Particle Sedimentation – Effect of Lift and Torque (p. 937)
- ACU-T: 6106 AcuSolve - EDEM Bidirectional Coupling with Mass Transfer (p. 962)
- ACU-T: 6107 AcuSolve - EDEM Bidirectional 3-Phase Coupling (p. 990)
- ACU-T: 6108 AcuSolve - EDEM Bidirectional Coupling with User-Defined Drag Model (p. 991)

ACU-T: 6000 Static Mixer Simulation – AcuTrace

Perform this tutorial using the following user interface(s):

HyperMesh CFD

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up, solving and viewing results for a simulation of a static mixer in combination with the post-processing moduleAcuTrace. In this simulation, AcuSolve is used to compute the species mixing within a simple mixer and AcuTrace is used to compute the particle motion of finite mass particles within the mixer. This tutorial is designed to introduce you to concepts necessary to visualize particulate flow with AcuTrace.

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. The following additional capabilities of AcuSolve are introduced in this tutorial:

- Generation of finite mass particle with AcuTrace.
- Conversion of the nodal output data with AcuTransTrace for reading into HyperMesh CFD Post.
- Post-processing the nodal output with HyperMesh CFD Post to visualize particles movement.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6000_Static_Mixer.hm`

`seeds1.csv`

`seeds2.csv`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1235. It consists of a mixing tube that contains several swept walls to instigate mixing within the tube. The diameter of the inlet is 0.1 m and the length of the mixing tube is 0.525 m. The fins have a mean diameter of 0.1 m. The maximum thickness of the fins is 0.003 m.

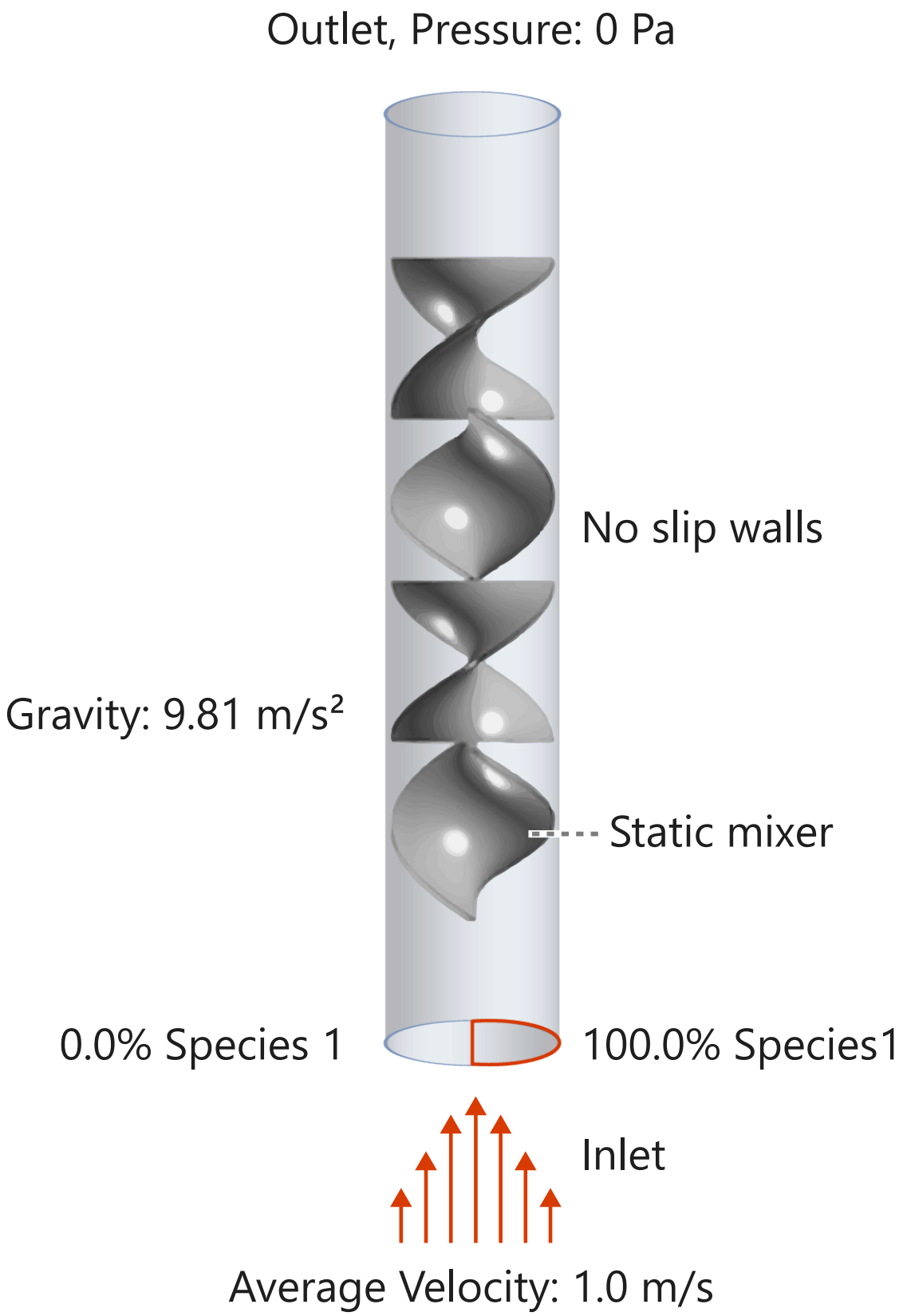

Outlet, Pressure: 0 Pa
No slip walls
Gravity: 9.81 m/s²
Static mixer
0.0% Species 1
100.0% Species1
Inlet
Average Velocity: 1.0 m/s

Figure 1235:

The boundary condition at the inlet is defined to produce a fully developed inlet profile with velocity of 1.0 m/s. One portion of the inlet is defined to contain 100 percent of species_1, while the other inlet is defined to contain 0.0 percent of species_1.

The fluid in this problem is an epoxy resin, which has a density of 1264.0 kg/m^3 and a viscosity of 1.49 kg/m-sec.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1236:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6000_Static_Mixer.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `Static_Mixer` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Static_Mixer` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1237:

Set Up the Problem

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

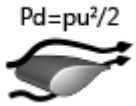


Figure 1238:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that the Time marching is set to **Steady**.
   b) Select **Spalart-Allmaras** as the Turbulence model.
   c) Set the Number of passive species to `1`.

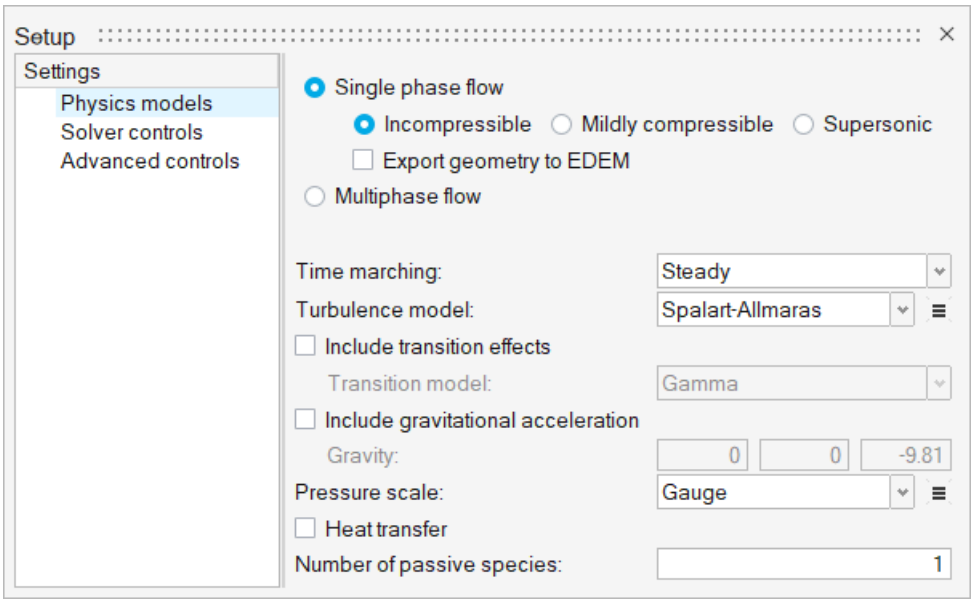


Figure 1239:

3. Click the **Solver controls** setting and verify that the parameters are set as shown in the figure below.

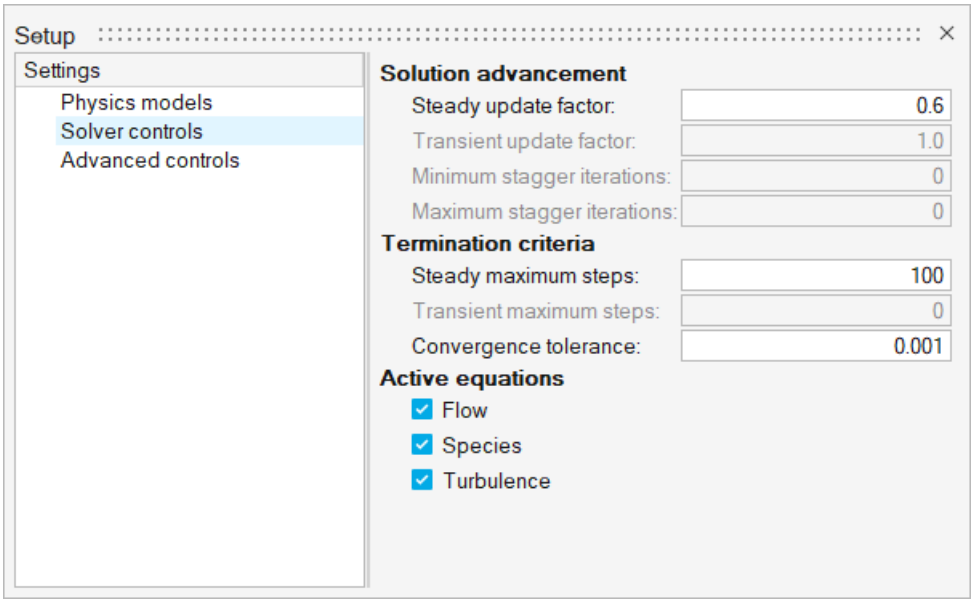


Figure 1240:

4. Close the dialog and save the model.

Define the Material Model

1. From the Flow ribbon, click the **Material Library** tool.

Figure 1241:

The **Material Library** dialog opens.

2. Click the **My Materials** tab.
3. Click **+** to add a new fluid material model.
4. In the **material creation** dialog, click the name in the top-left corner and rename the material to `Epoxy Resin`.
5. In the Density tab:
   a) Set the Type to **Constant**.
   b) Set the Density value to `1264` kg/m^3.
   c) Set the Isothermal compressibility value to `0`

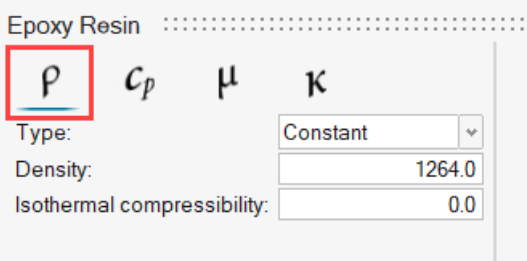


Figure 1242:

6. Click the **Specific Heat** tab and set the Specific heat value to `2386` J/kg-K.

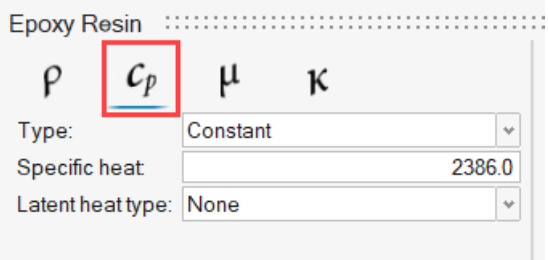


Figure 1243:

7. Click the **Viscosity** tab and set the Viscosity value to `1.49` kg/m-s.

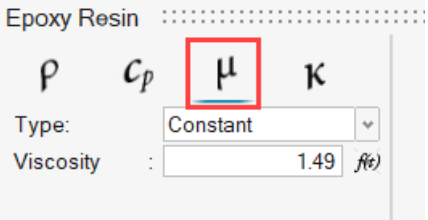


Figure 1244:

8. Click the **Conductivity** tab and set the Conductivity value to `0.286` W/m-K and Turbulent Prandtl number to `0.91`.

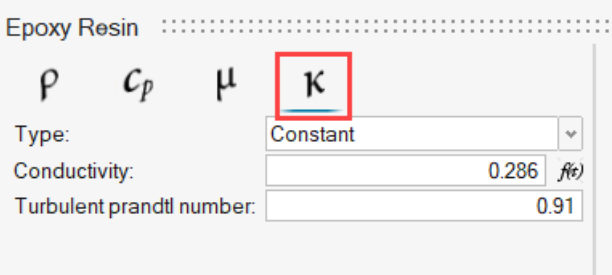


Figure 1245:

9. Close all dialogs and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1246:

2. Select the solid model then select **Epoxy_Resin** as the fluid medium.

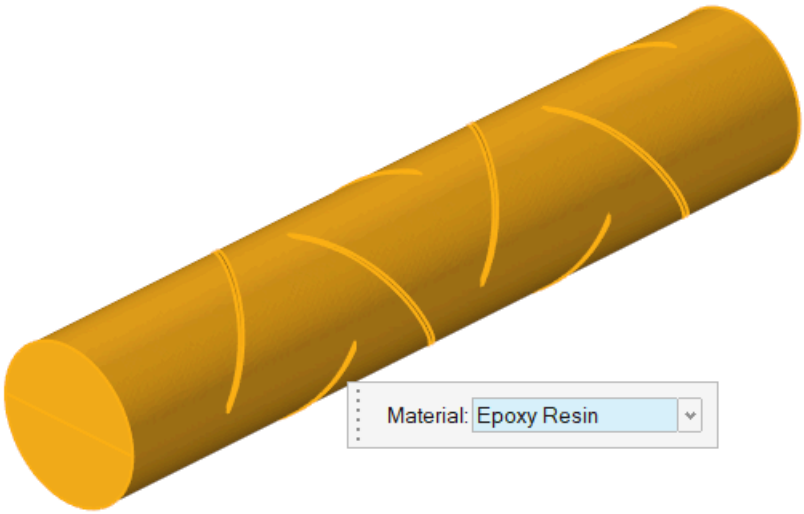


Figure 1247:

3. On the guide bar, click ✓ to execute the command and exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1248:

2. In the modeling window, click the inlet surface highlighted in the figure below.

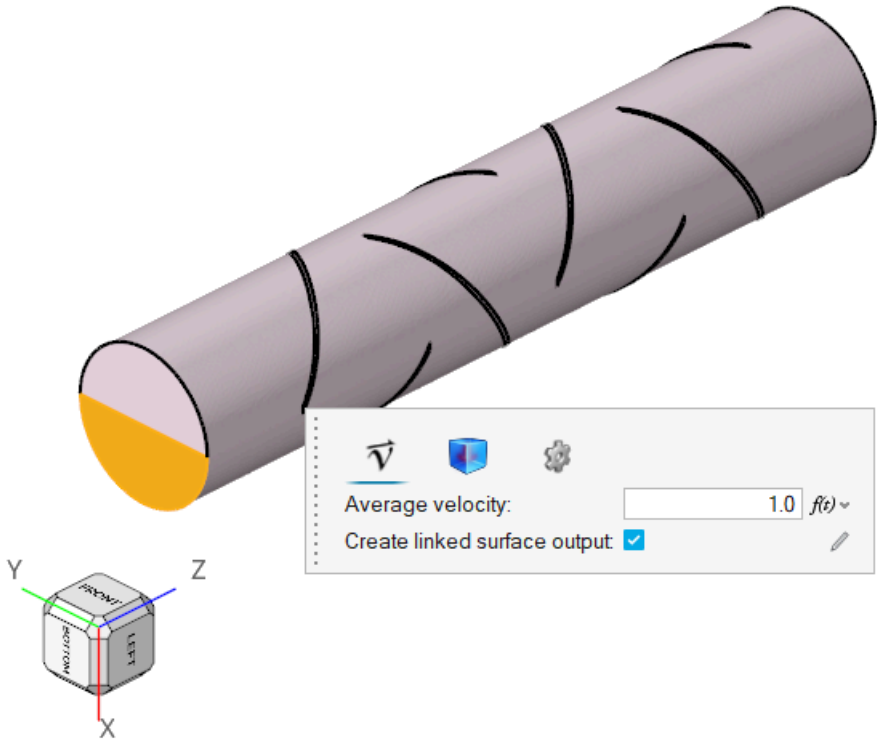


Figure 1249:

3. In the microdialog, set the Average velocity to `1.0` m/s.
4. Click the **Species** tab and enter a value of `1.0` for Species 1.

Figure 1250:

5. On the guide bar, click ▸ to execute the command and remain in the tool.
6. Right-click **Inlet** in the Boundaries legend and select **Rename**. Change the name to `Inlet S1`.
7. Select the other inlet surface highlighted in the figure below.

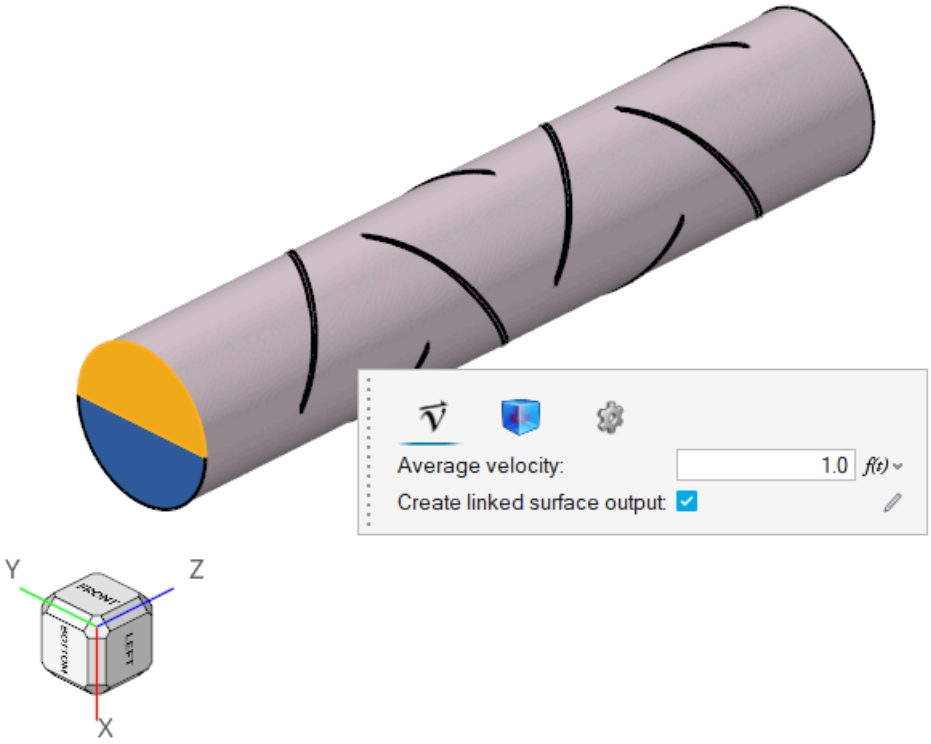


Figure 1251:

8. In the microdialog, set the Average velocity to `1.0` m/s.
9. Click the **Species** tab and enter a value of `0.0` for Species 1.
10. On the guide bar, click ▸ to execute the command and remain in the tool.
11. Right-click **Inlet** in the Boundaries legend and select **Rename**. Change the name to `Inlet S2`.
12. Click the **Outlet** tool.

Figure 1252:

13. Select the surface highlighted below and then click ▸ on the guide bar.

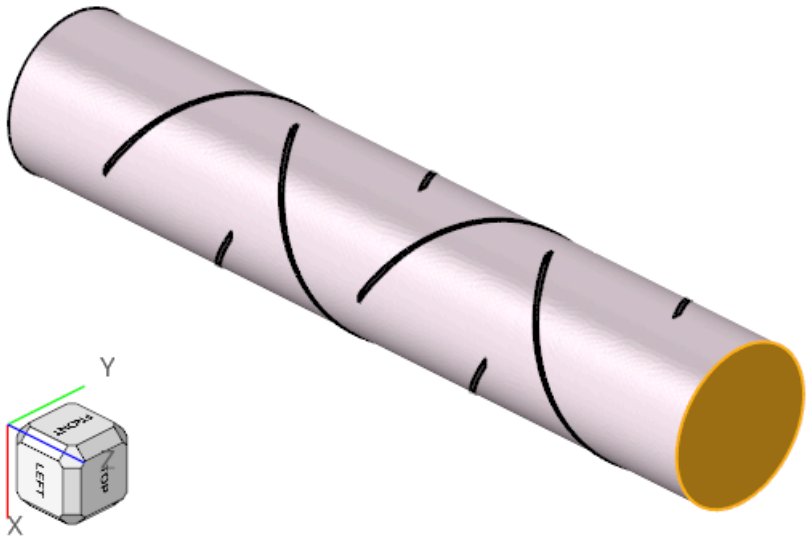


Figure 1253:

14. Click the **No Slip** tool.

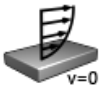


Figure 1254:

15. Select the wall of the tube highlighted in the figure below and then click ▸ on the guide bar.

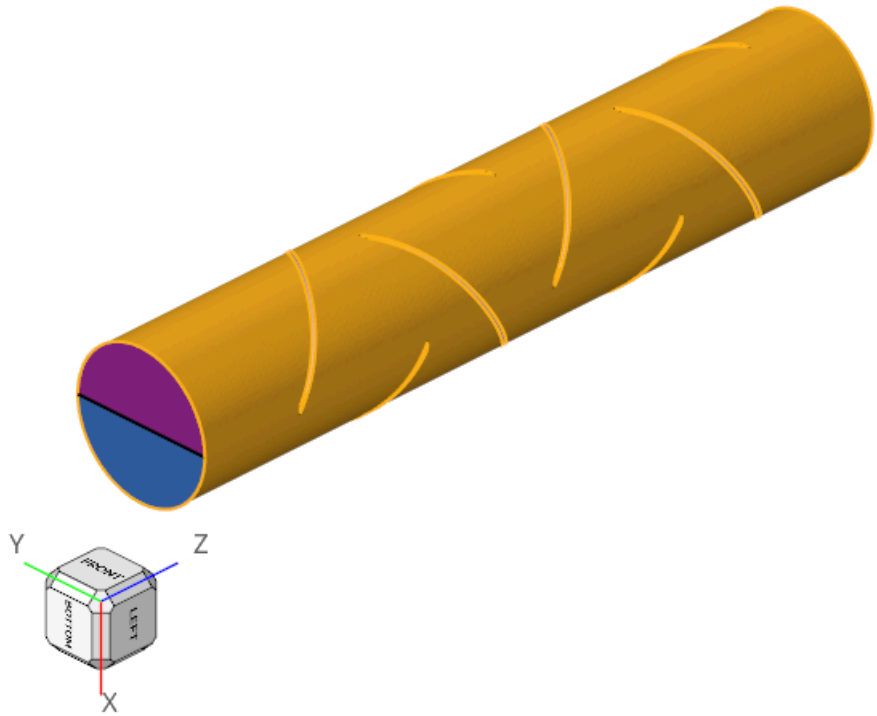


Figure 1255:

16. Rename the surface group from Wall to `Pipe Wall`.

17. Right-click **Default Wall** in the Boundaries legend and select **Isolate**.

18. Select all the surfaces highlighted in the figure below and then click ▸ on the guide bar.

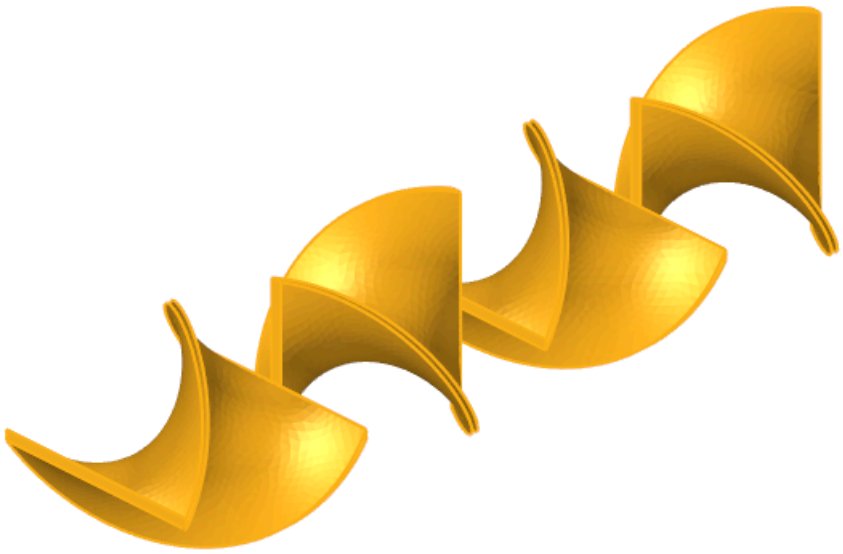


Figure 1256:

19. Rename the surface group from Wall to **Fin Walls**.
20. Click ✓ on the guide bar.
21. Turn on the visibility of all the surfaces by right-clicking in the modeling window and selecting **Show All** or by simply pressing the **A** key.
22. Save the model.

Define Surface Monitors and Run AcuSolve

1. From the Solution ribbon, click the **Volumes** tool.

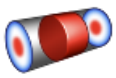


Figure 1257:

2. Select the model and then click ▸ on the guide bar.

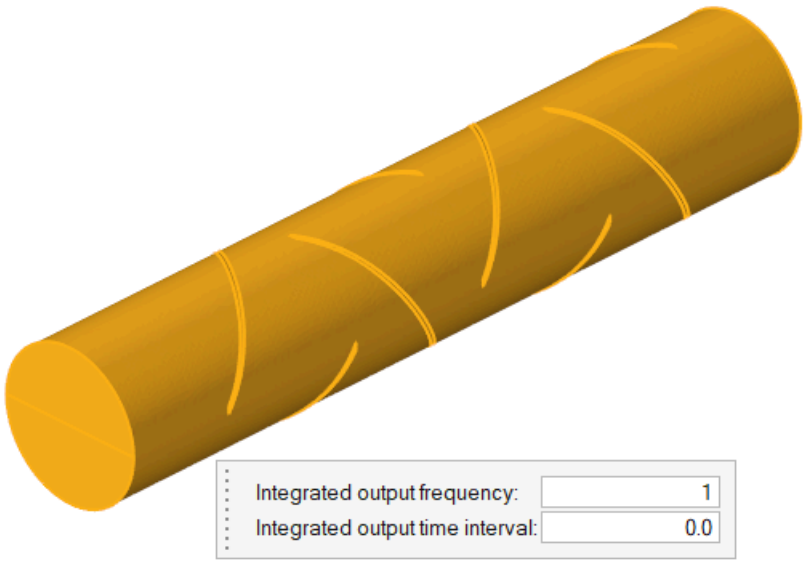


Figure 1258:

3. Click ✓ on the guide bar.
4. From the Solution ribbon, click the **Run** tool.

Figure 1259:

The **Launch AcuSolve** dialog opens.

5. Make sure the problem directory is set to the correct location. If not, use the Browse option to select the correct directory.
6. Set the Parallel processing option to **Intel MPI**.
7. Optional: Set the number of processors to **4** or **8** based on availability.
8. Leave the remaining default options and then click **Run** to launch AcuSolve.

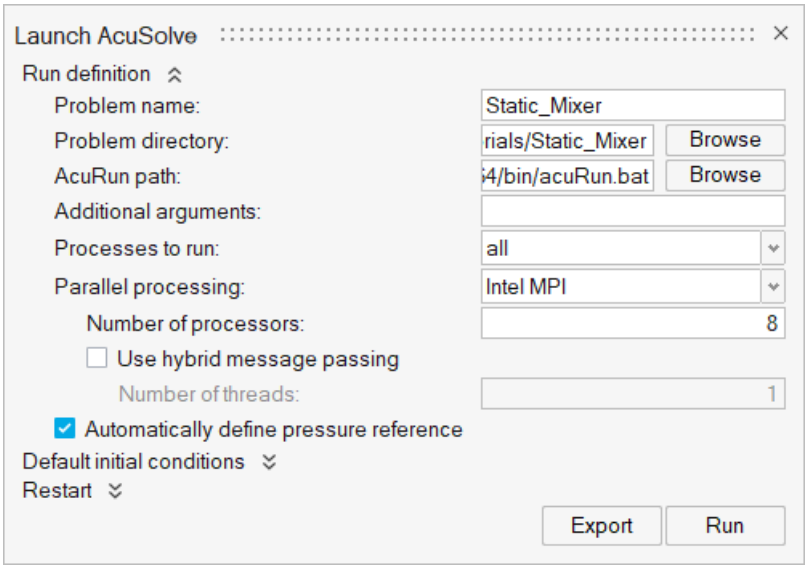


Figure 1260:

Post-Process With the Plot Tool

1. From the Solution ribbon, click the **Plot** tool.

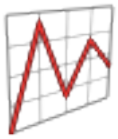


Figure 1261:

2. Observe the residual and solution ratios as the simulation progresses.

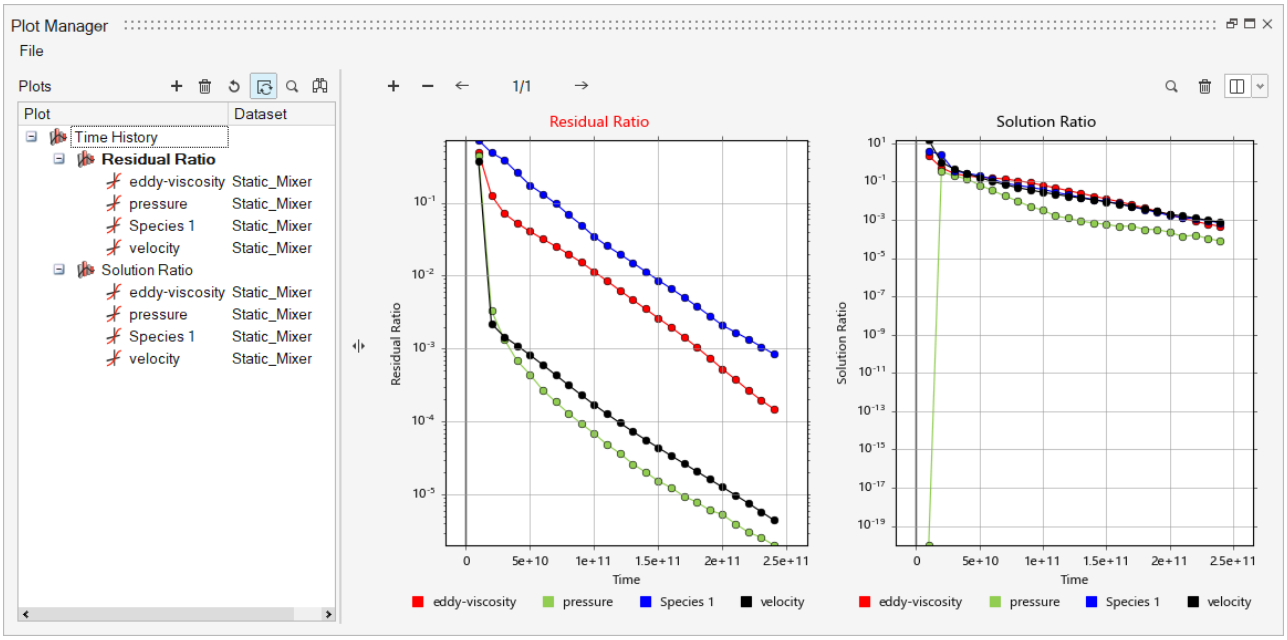


Figure 1262:

Post-Process with HyperMesh CFD Post

1. Navigate to the **Post** ribbon.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1263:

3. Open the AcuSolve `.log` file in your problem directory to load the results for post-processing.
4. Verify that the default step is set to **Last** in the **Reader Options** dialog and then click **Import**.

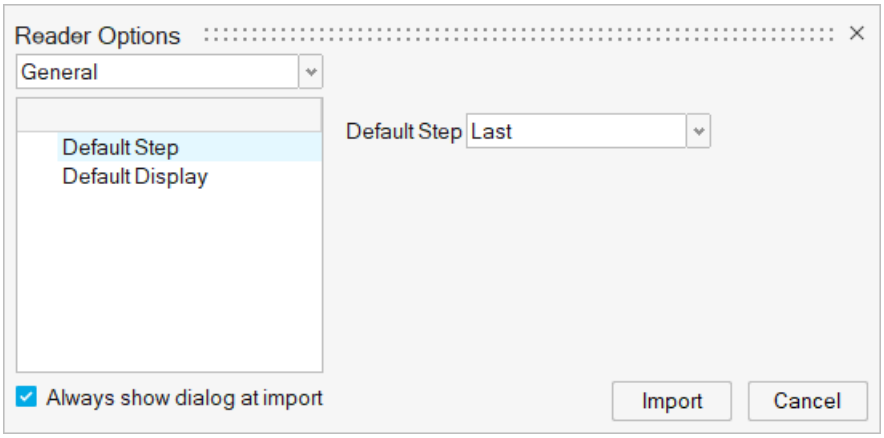


Figure 1264:

The solid and all the surfaces are loaded in the Post Browser.

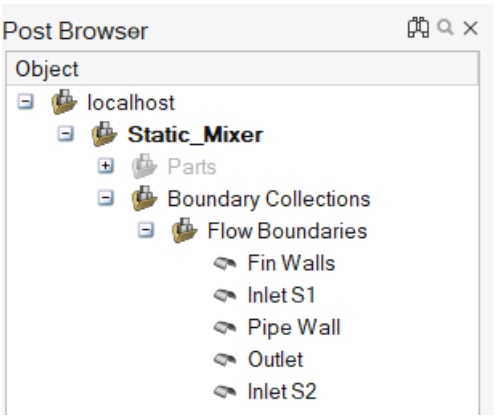


Figure 1265:

5. In the Post Browser, right-click **Pipe Wall** and select **Edit**. Set the transparency to around 80%.

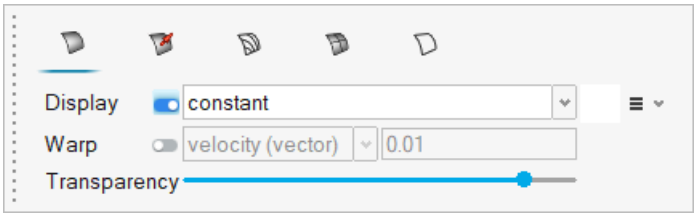


Figure 1266:

6. Similarly, edit the transparency of Fin Walls to around 30% .

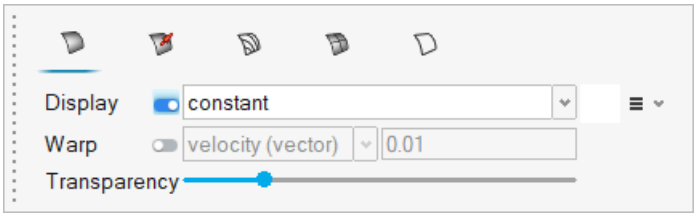


Figure 1267:

7. Hide **Outlet** and **Inlet S2**.
8. Click the **Slice Planes** tool.

Figure 1268:

9. Select the x-z plane in the modeling window and then click **Calculate**.

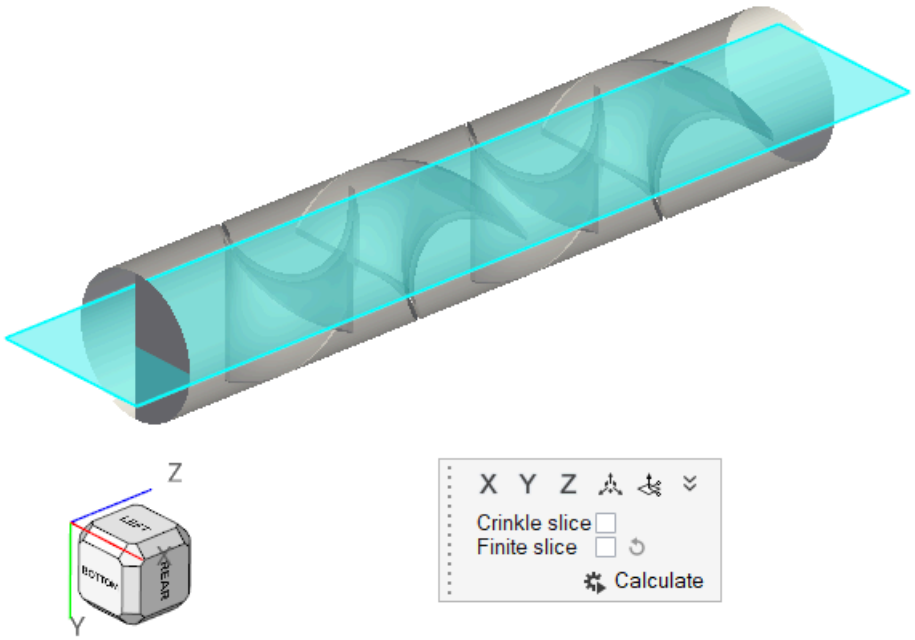


Figure 1269:

10. In the display properties microdialog, set the display to **species 1** and activate the **Legend** toggle.
11. Click to reset the legend.

12. Click ≡ and set the Colormap name to **Rainbow Uniform**.

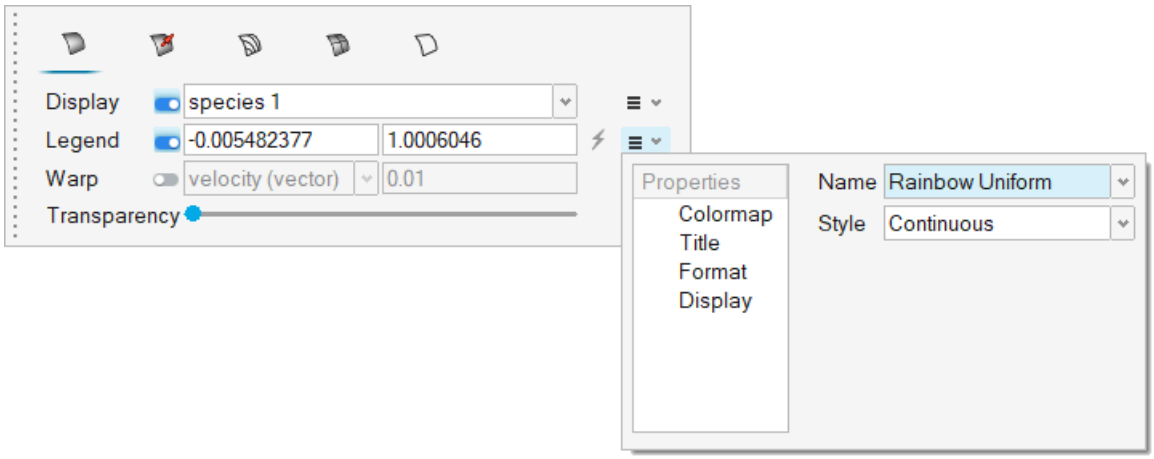


Figure 1270:

13. Click ✓ on the guide bar.
The plot below shows the mixing effectiveness as the maximum value of species 1 is reduced to 0.7 at the outlet, while the minimum value of species 1 is increased to 0.3.

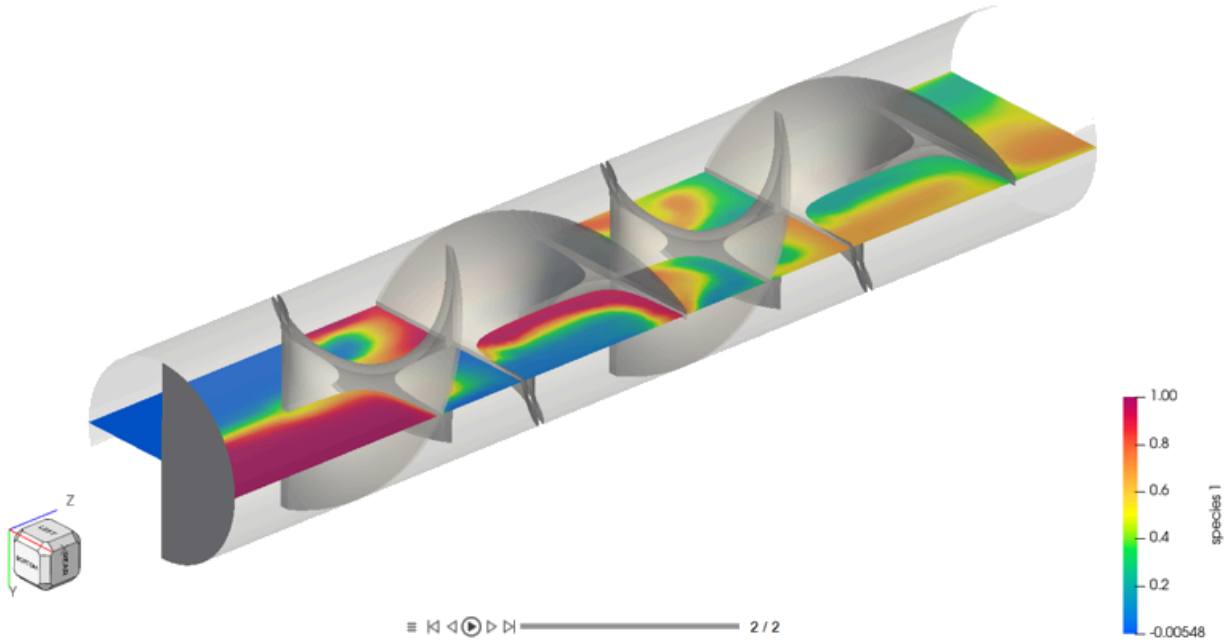


Figure 1271:

Prepare Particle Trace Attributes for AcuTrace

Now that the steady-state simulation is complete, you can use the finite mass particle tracer to simulate micro-particles of SiO_2, which are often used to add strength to the epoxy.

Define Particle Trace Parameters for Static Analysis

1. Navigate to the **AcuTrace** ribbon.
2. Click the **Setup** tool.

Figure 1272:

3. Click **Problem definition** and set the Particle type to **Finite Mass**.
4. For the AcuSolve working directory, select the **ACUSIM.DIR** (flow results) folder in your problem directory.
5. Make sure the Flow field type set to **Static**.
6. Disable **Mesh motion** if it not disabled already.

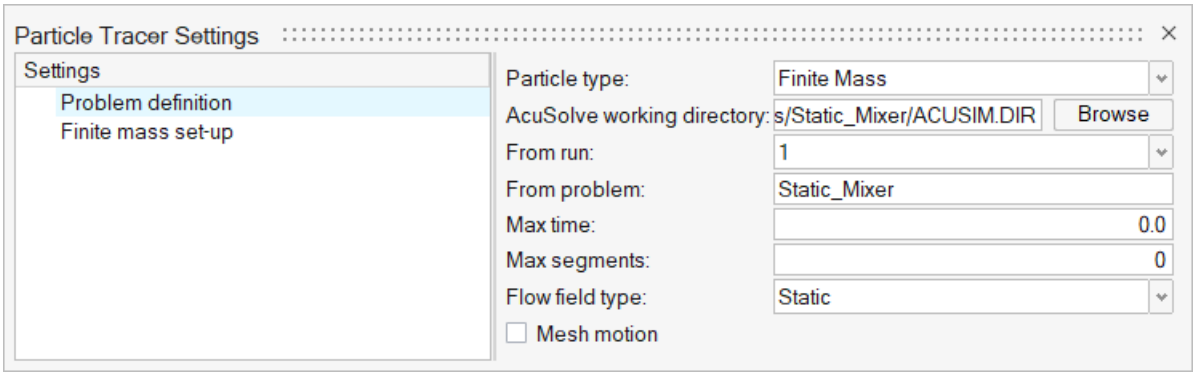


Figure 1273:

7. Click **Finite mass set-up** and leave all the parameters as shown below.

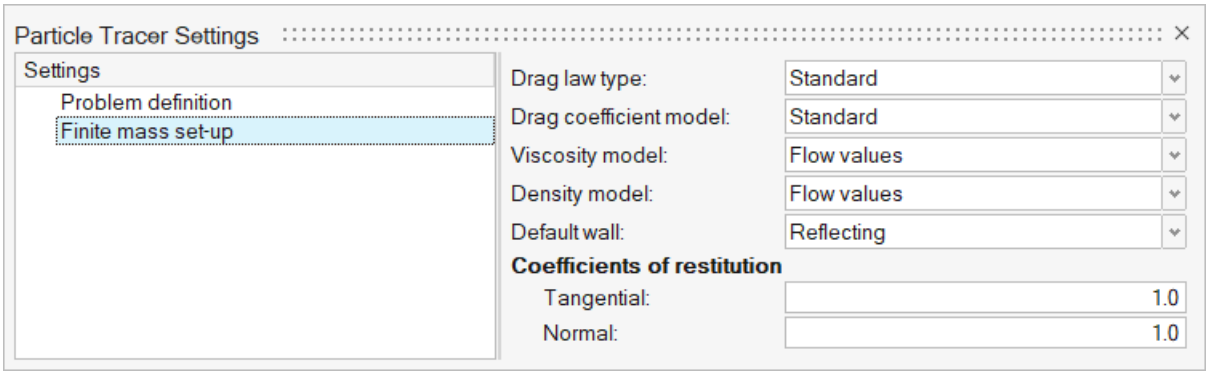


Figure 1274:

8. Close the dialog.

Define Particle Seeds

In the next steps, you will define the particle seeds that are moving into the flow regime.

1. Click the **Seeds** tool.

Figure 1275:

2. In the **Seed location** window, click 🗁.
3. Browse to the location where you saved `seeds1.csv` and `seeds2.csv`.
4. Select `seeds1.csv` file and click **Open**.
   This accounts for the particles being fed from Inlet S1.

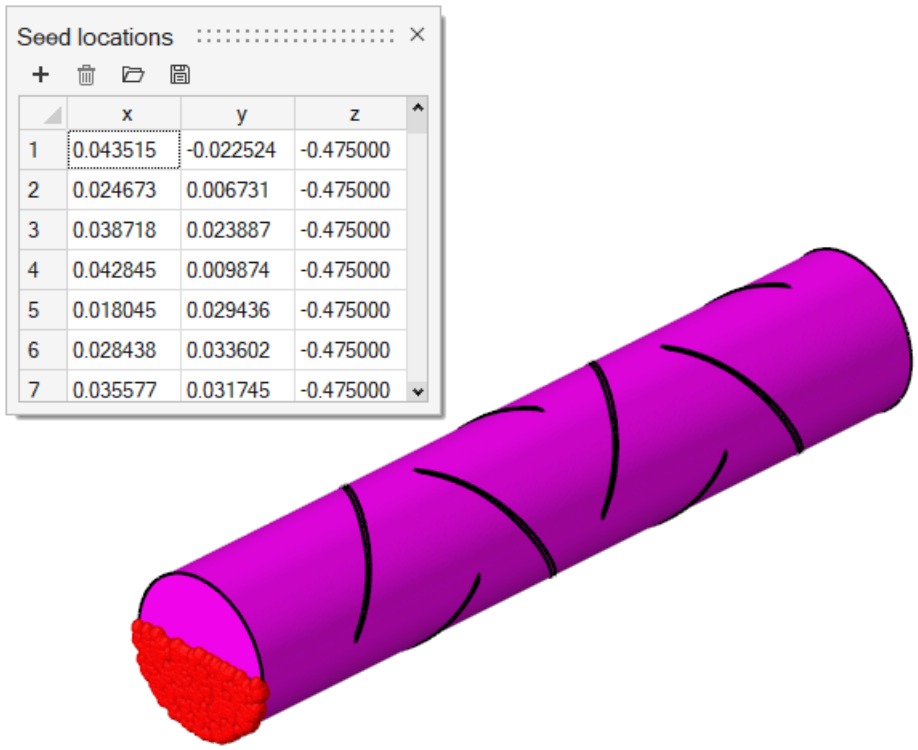


Figure 1276:

5. Click **Parameters** on the guide bar and enter the following details.

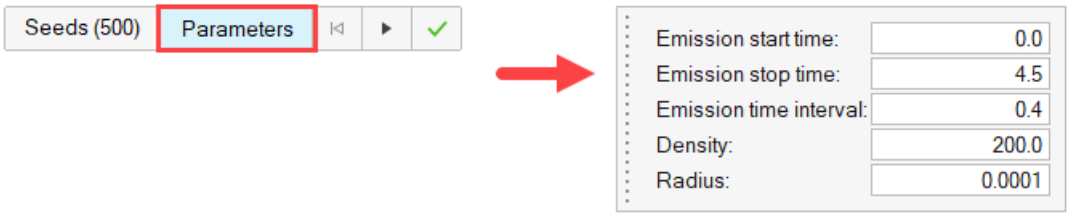


Figure 1277:

6. Click ▸ on the guide bar to execute the changes.
7. Right-click on **Particle Seeds** in the legend and select **Rename**. Change the name to `Inlet S1 Seeds`.
8. Repeat the same process and open `seeds2.csv`.

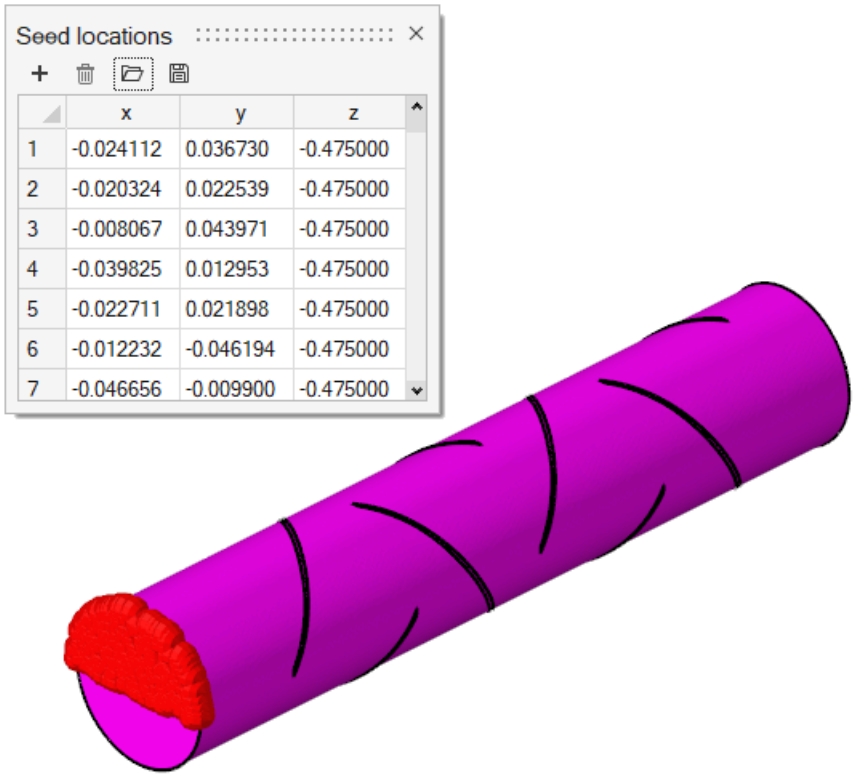


Figure 1278:

9. Click **Parameters** on the guide bar and enter the following details.

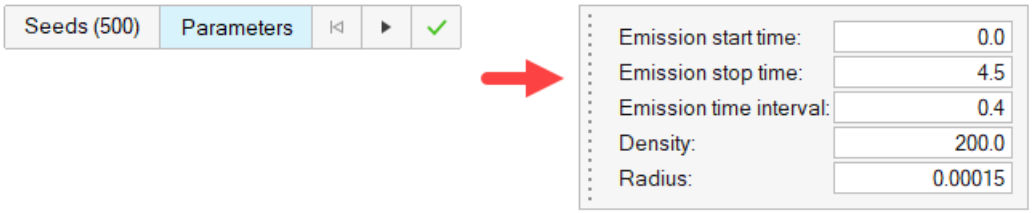


Figure 1279:

10. Click ▸ on the guide bar to execute the changes.
11. Right-click on **Particle Seeds** in the legend and select **Rename**. Change the name to `Inlet S2 Seeds`.
12. On the guide bar, click ✓ to execute the command and exit the tool.

Define Finite Mass Boundary Conditions

1. Click the **Reflecting** tool.

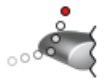


Figure 1280:

2. In the **Boundary Conditions** window, select the **Pipe Wall** and then set the Coefficients of restitution values to `0.2`.

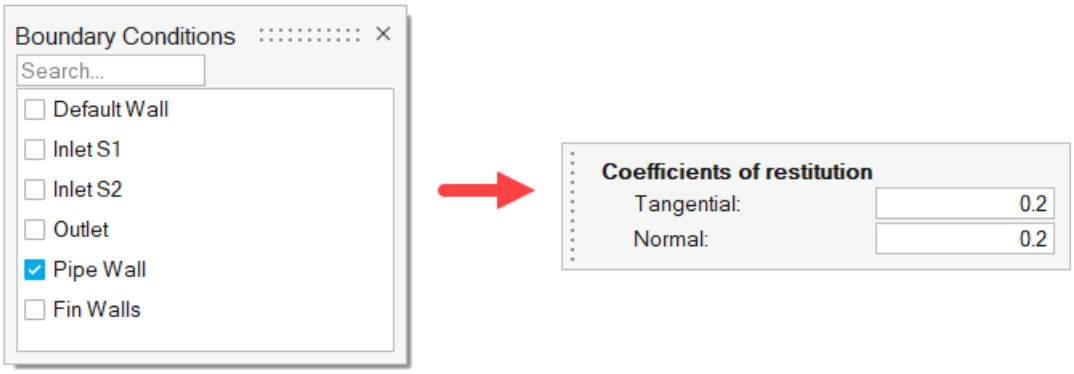


Figure 1281:

3. Click ▸ on the guide bar to execute the changes.
4. Right-click on **Reflecting Wall** in the legend and select **Rename**. Change the name to `Pipe Wall`.
5. In the **Boundary Conditions** window, select the **Fin Walls** and then set the Coefficients of restitution values to `0.8`.

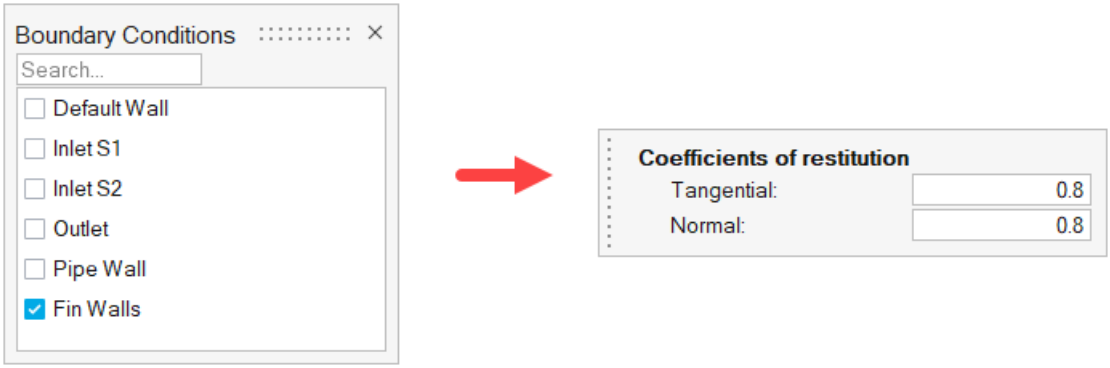


Figure 1282:

6. Click ▸ on the guide bar to execute the changes.
7. Right-click on **Reflecting Wall** in the legend and select **Rename**. Change the name to `Fin Walls`.

8. On the guide bar, click ✓ to execute the command and exit the tool.
9. Click the **Stopping** tool.

Figure 1283:

10. In the **Boundary Conditions** window, select the **Outlet**.
11. Click ▸ on the guide bar then ✓.

Define Output Parameters

1. Click the **Output** tool.

Figure 1284:

2. Check **Trace output** and enter the values shown below.

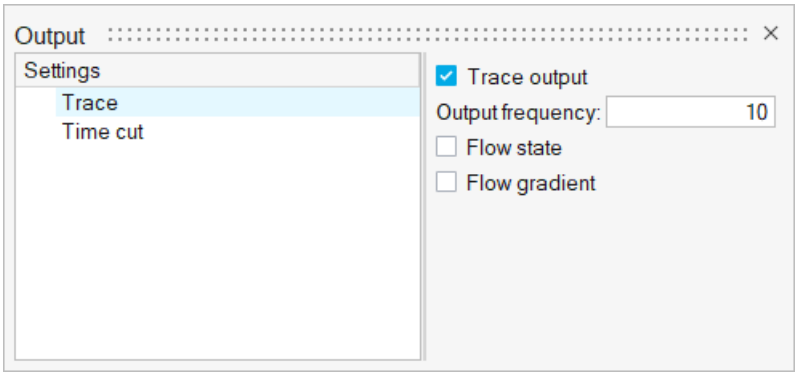


Figure 1285:

3. Click the **Time cut** setting and enter the values shown below.

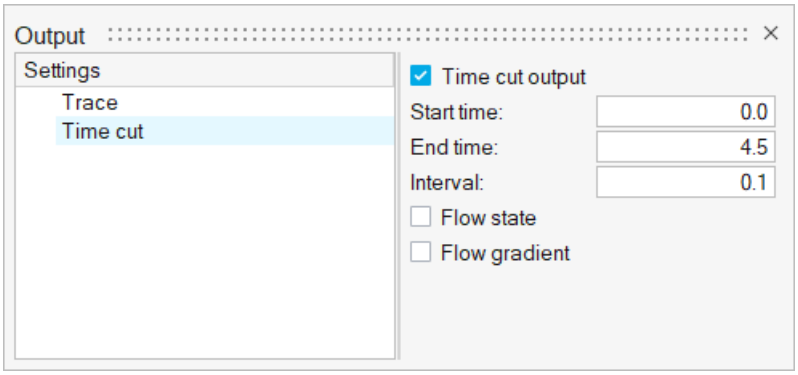


Figure 1286:

4. Close the dialog.

Run AcuTrace

1. Click the **Run** tool.

Figure 1287:

2. Check that `acuRunTrace.bat` is pointing to the correct path: `<installation_directory>\hwcfdsolvers\acusolve\win64\bin\acuRunTrace.bat`.
3. Set the Parallel processing option to **Intel MPI**.
4. Optional: Set the number of processors to **4** or **8** based on availability.
5. Leave the remaining options as default and click **Run** to launch AcuTrace.

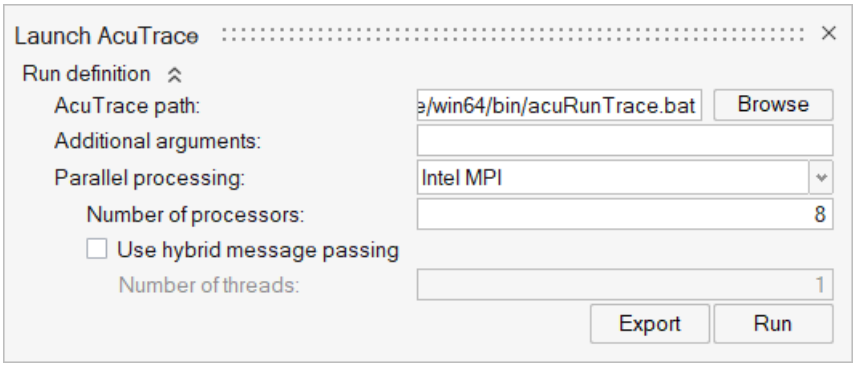


Figure 1288:

Convert Results for Ensight

Once the run is complete, you need to convert the results so that they can be read in HyperMesh CFD Post. To do this, run the AcuTransTrace utility. This tool can be used to convert data for Ensight.

1. Click the **Convert** tool.

Figure 1289:

The **AcuTrace Data Translator** window opens.

2. Browse to the location where the AcuTrace `.log` file (`*.tlog`) is located.
3. Set the Output format to **Ensight**.
4. Set Data type to **Time Cut**.

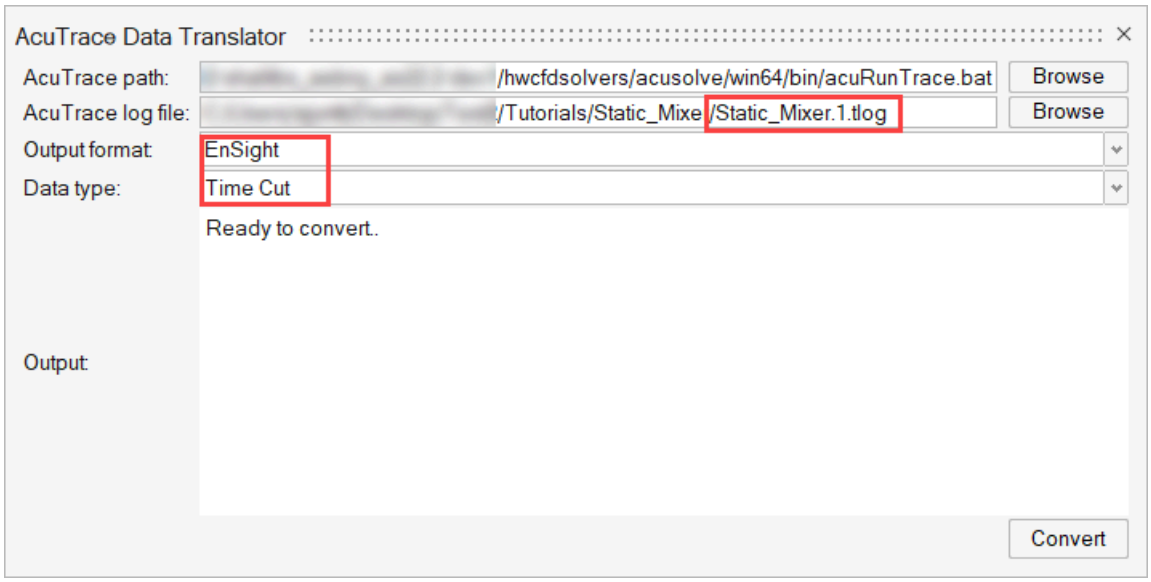


Figure 1290:

5. Click **Convert**.
   Once the conversion is successful, it generates two Ensight result files.

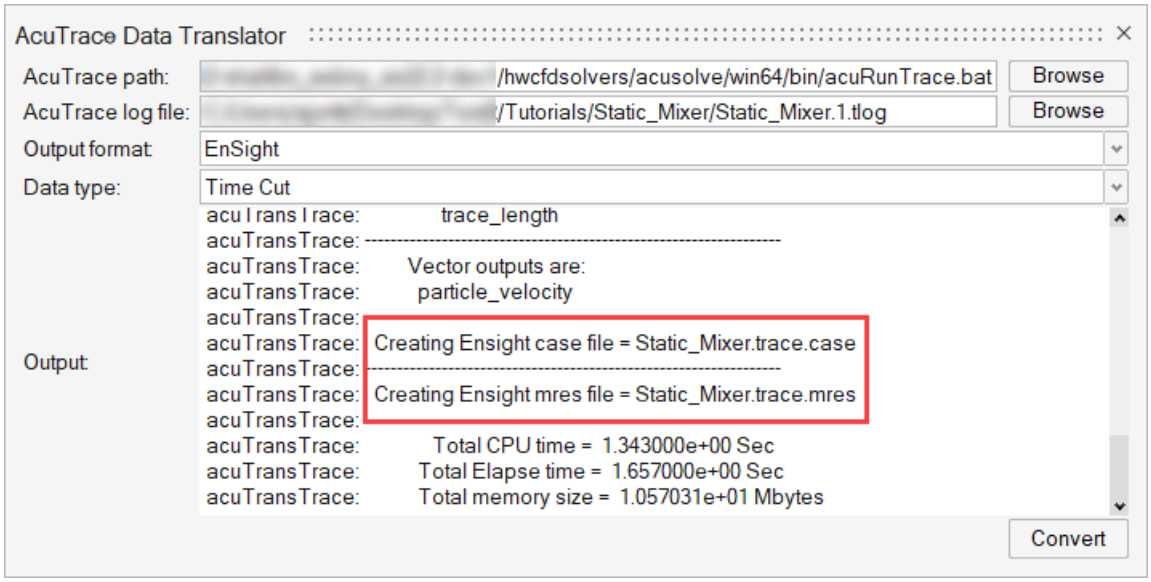


Figure 1291:

6. Close the dialog.

Post-Process the Ensight File

1. Navigate to the **Post** ribbon.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1292:

3. Open the Ensight case file (`static_mixer.trace.case`) in your problem directory to load the results for post-processing.

> **Note:** A warning opens wherein if some results were previously loaded it will try to remove the results first then load the current results. Click **Continue**.

The solid and all the surfaces along with the Particles are loaded in the Post Browser.

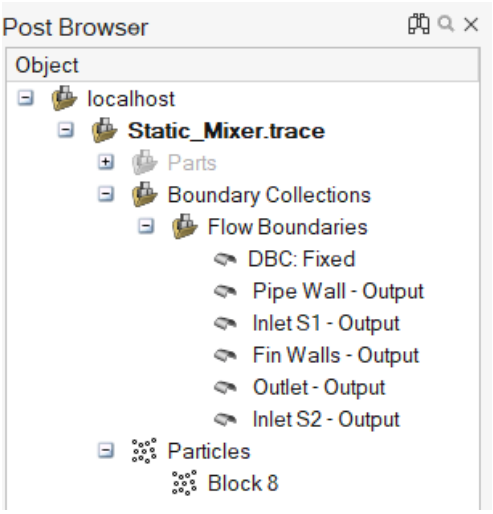


Figure 1293:

4. In the Post Browser, turn off the display for all surfaces except for Pipe Wall and Fin Walls.
5. Set the transparency for Fin Walls to around 30% and the transparency for Pipe Wall to around 80%.
6. Right-click **Block8** under Particles and select **Edit**.
7. Set the display to **Radius**. Click ≡ and set the point radius to `0.003`.

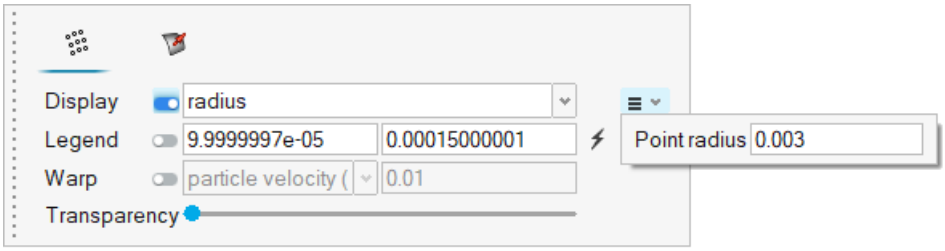


Figure 1294:

8. Activate the **Legend** toggle and click ⚡ to refresh the range.
9. Click ≡ and set the colormap name to **Rainbow Desaturated**, the colormap style to **Filled**, and the number of colors to `2`.

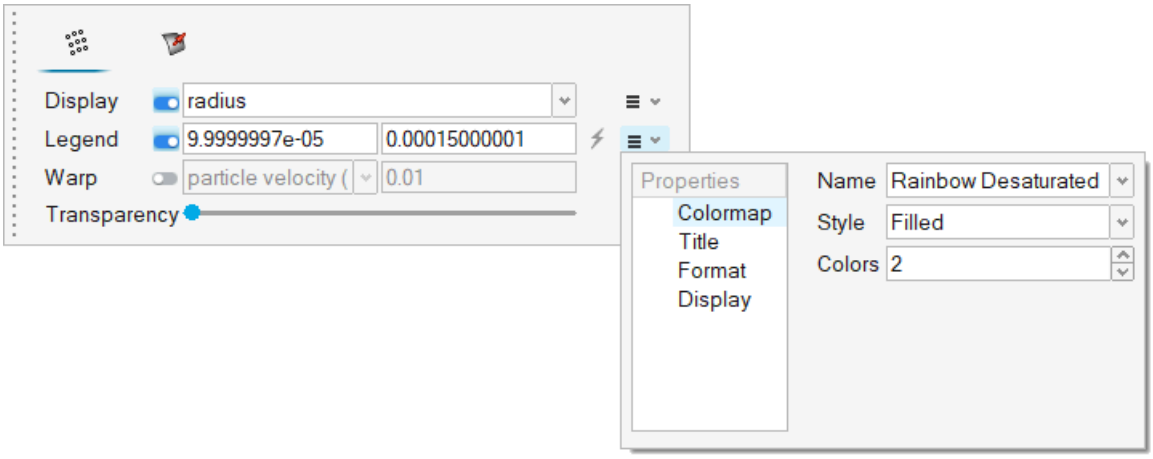


Figure 1295:

10. Switch to the **Display** properties. Set the Orientation to **Horizontal** and select the **Lower Center** button.

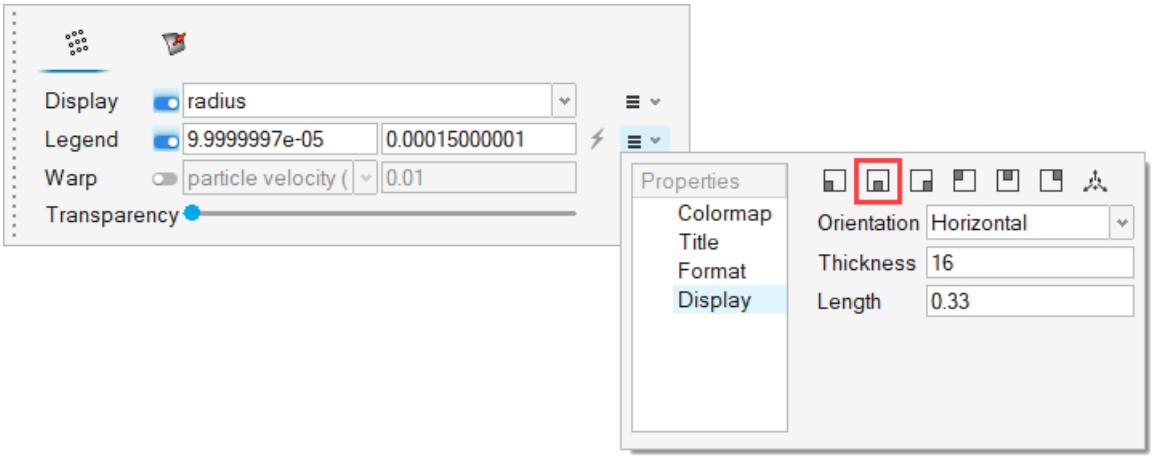


Figure 1296:

11. Click the **Isometric** face on the View Cube to align the model.

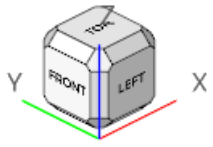


Figure 1297:

12. Rotate the model as shown in below figure such that you can view the particles movement inside the domain more precisely.

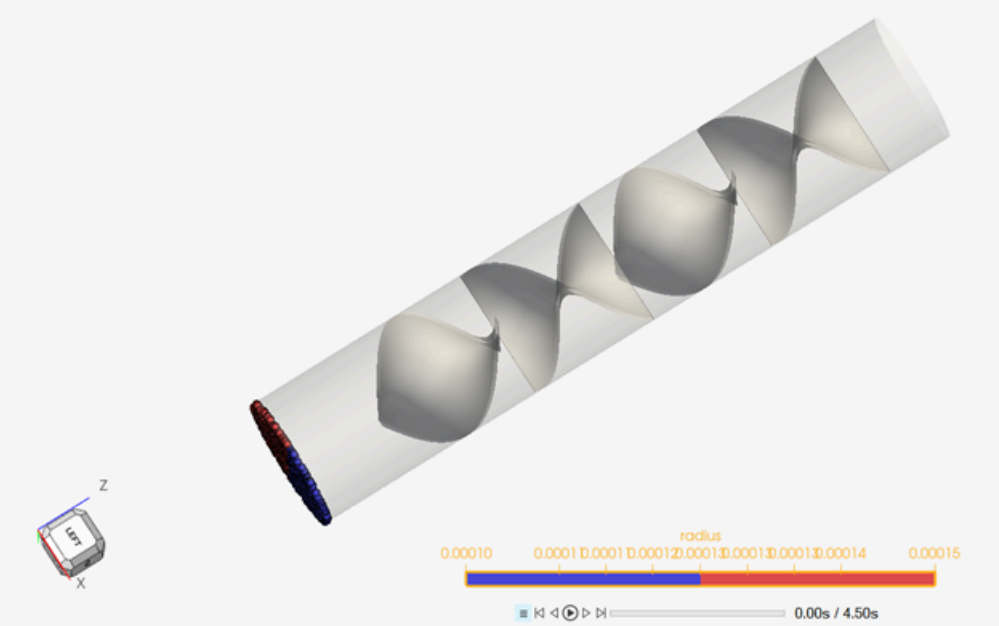


Figure 1298:

13. Click ▶ at the bottom of the modeling window to view a live animation of the flow.
14. Save the animation.
    a) Go to **File** > **Screen Capture** > **Advanced Screen Capture**.
    b) Click ≡ and uncheck **Include mouse cursor**.
    c) Set the frame rate to `30`.
    d) Click ⌖ on the toolbar then drag over the area you want to record.
    e) Click ◉ to begin recording and the same icon to stop recording.
    f) Name the file and save it.

Summary

In this tutorial, you successfully set up and solved for a steady simulation of a static mixer to visualize the particle flow. You started the tutorial by opening a database in HyperMesh CFD and setting up the simulation parameters to compute the species mixing within the mixer. Then, you set up the particle seeds and ran AcuTrace to generate the particles flow within the static mixer. Lastly, you converted the data using AcuTransTrace to visualize the particles flow in HyperMesh CFD Post.

ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up and running a basic sequential coupling (one-way steady) simulation using AcuSolve and EDEM. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6100_windshifter.hm`

Note: This tutorial does not cover the steps related to geometry cleanup and mesh settings.

Problem Description

The problem to be solved is shown schematically in Figure 1299. It is a windshifter model in which air enters the domain from the inlet at the bottom at a very high velocity (20 m/sec) and exits through the outlet. When particles are introduced into the domain through the particle inlet, the lighter particles get carried away with the air and the heavier particles exit through the opening at the bottom. Since the concentration of the particles is sparse, the effect of the particles on the fluid field is not considered for this simulation. Hence, one-way coupling between AcuSolve and EDEM is used to simulate the separation of particles; only the effect of the fluid forces on the particles is considered.

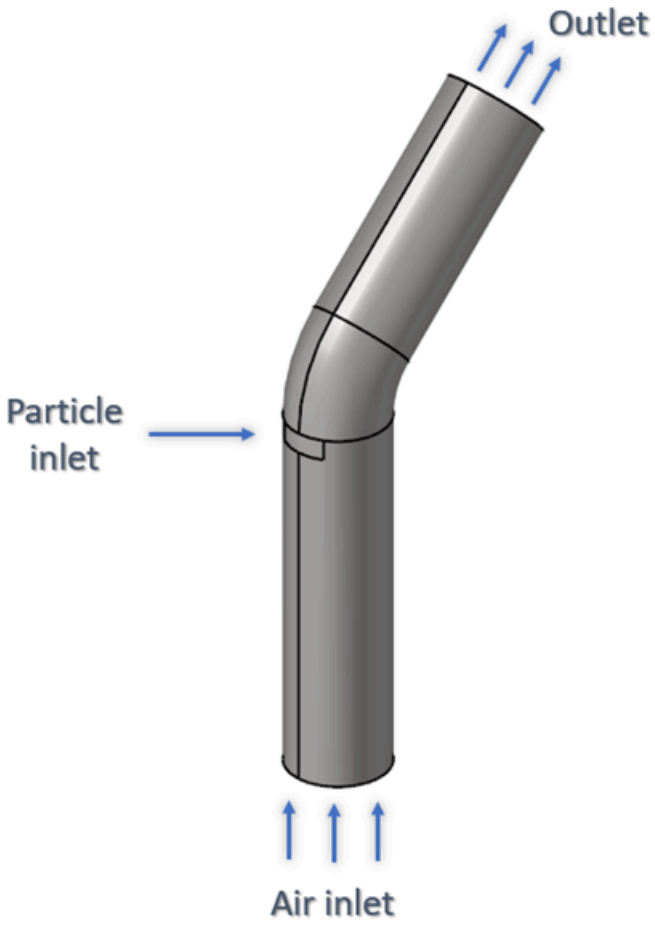


Figure 1299:

The model consists of a cylindrical pipe with a 45-degree bend. The radius of the pipe is 0.25 m and the particle inlet is located midway through the length of the pipe. To simplify things, the particle inlet is considered as a wall for the CFD simulation. The workflow for an AcuSolve-EDEM sequential coupling simulation is shown below.

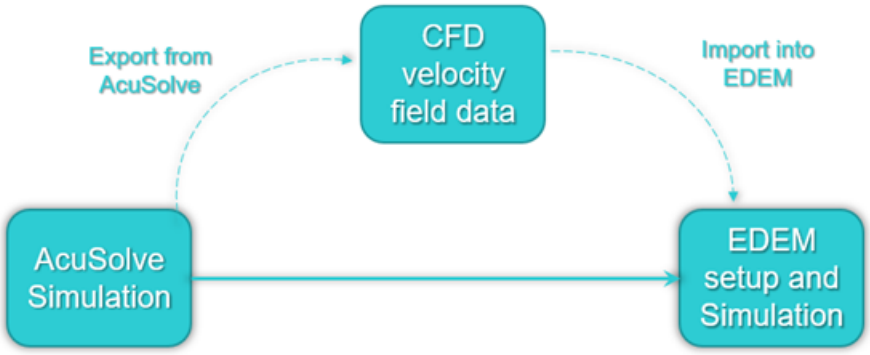


Figure 1300:

The tutorial consists of two parts:

1. AcuSolve simulation
2. EDEM simulation

The setup for the AcuSolve simulation is done using HyperMesh CFD. Once the AcuSolve simulation is complete, the velocity nodal data is exported into CGNS format using the AcuTrans utility available in AcuSolve. This field data will be imported into EDEM and will be used to calculate the drag forces on the particles due to the fluid.

Two different bulk materials used in the EDEM simulation and their properties are listed below:

Name	**Density (kg/m^3)**	**Particle radius (m)**	**Average weight of individual particle (kg)**	**Rate of generation (particles per sec)**
Heavy particle	900	0.03	0.004	100
Light particle	100	0.03	0.0004	100

The fluid drag forces on the particles are calculated using the Schiller-Naumann drag model. The drag force on the particle is calculated using the relative velocity of the particle with respect to the local fluid velocity. This particle body force will used to update the location of the particles and then the new drag forces are calculated. This loop is repeated until the end of the simulation.

Part 1 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1301:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6100_windshifter.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Save the database as `windshifter_sequential` in the same directory as the other input files.

   This will be the working directory and all the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1302:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

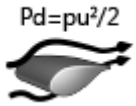


Figure 1303:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Verify that the **Incompressible** option is selected under Single phase flow.
   b) Activate the **Export geometry to EDEM** check-box.
   c) Set Time marching to **Steady**.
   d) Select **Spalart-Allmaras** as the Turbulence model.

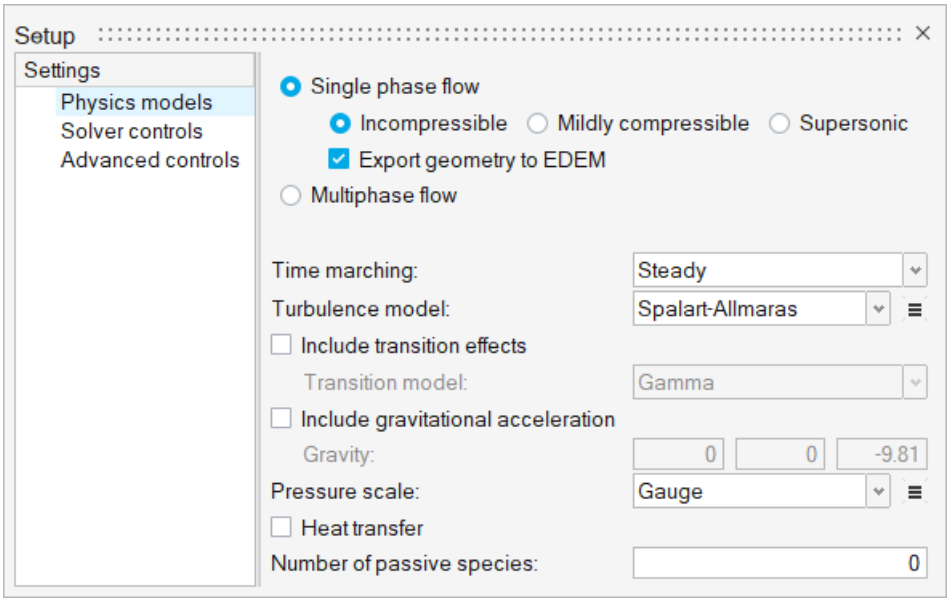


Figure 1304:

3. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1305:

2. Verify that Air has been assigned as the material.

3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1306:

2. Click on the inlet face highlighted in the figure below and enter a value of `20` m/s for the Average velocity in the microdialog.

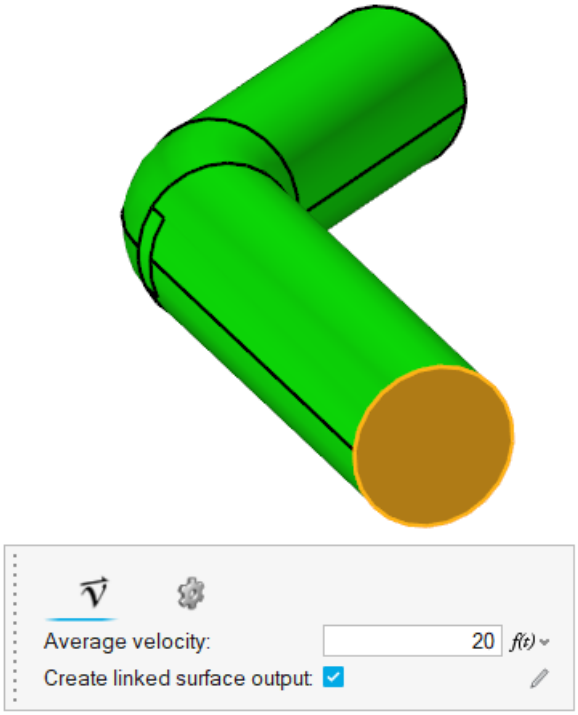


Figure 1307:

3. On the guide bar, click ✓ to execute the command and exit the tool.
4. Click the **Outlet** tool.

Figure 1308:

5. Select the face highlighted in the figure below and then click ✓ on the guide bar.

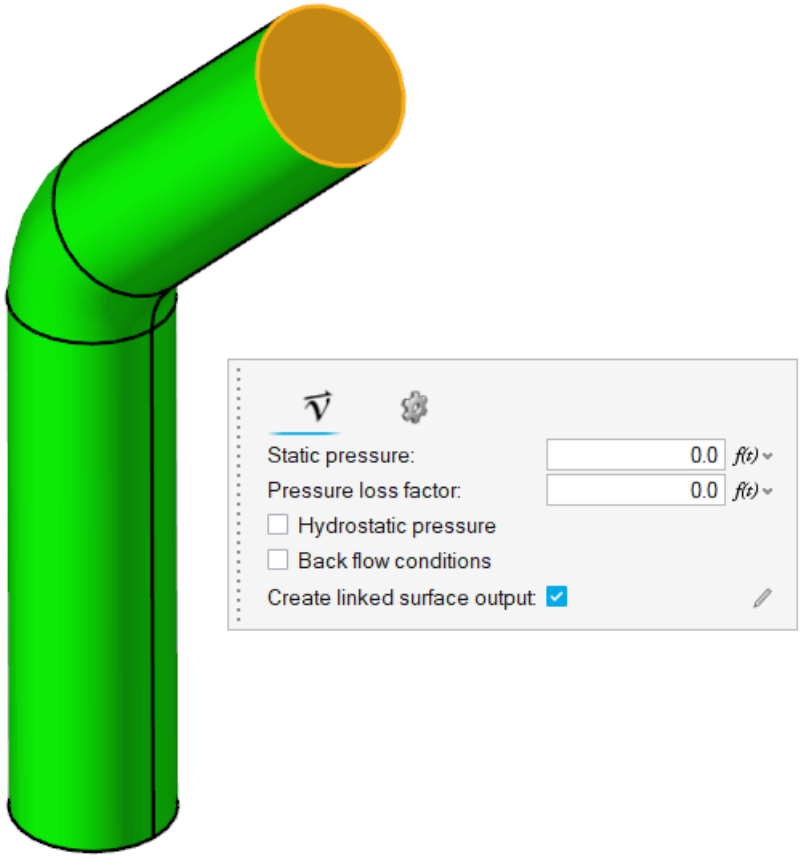


Figure 1309:

6. Click the **No Slip** tool.

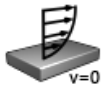


Figure 1310:

7. Select the face highlighted in the figure below.

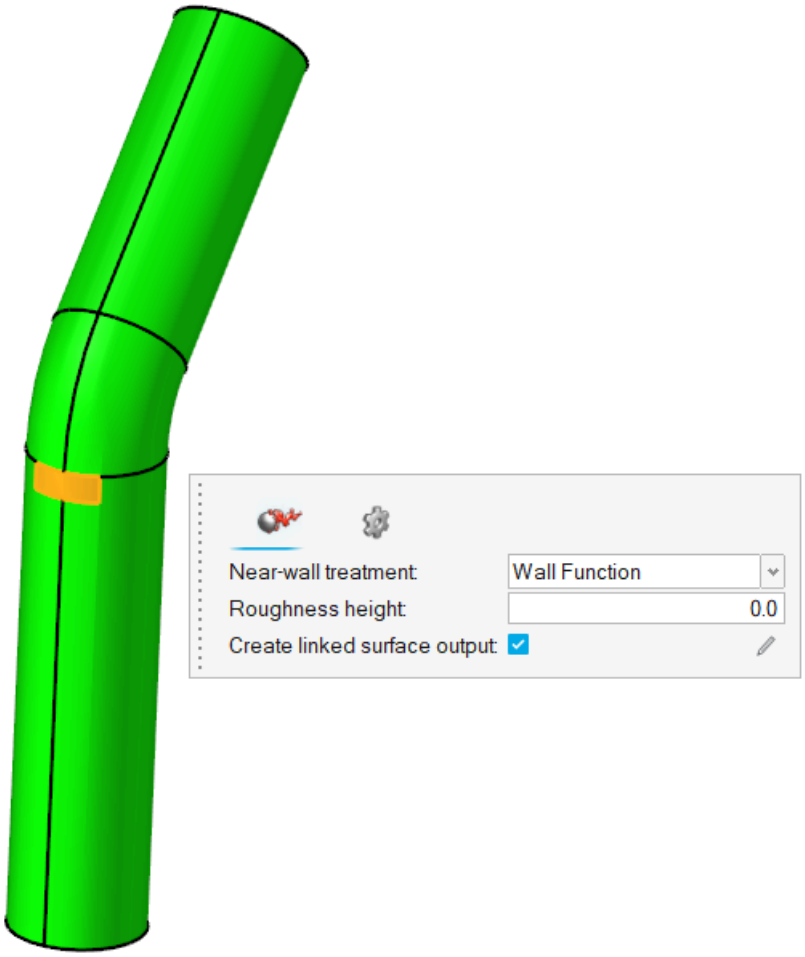


Figure 1311:

8. In the Boundaries legend, double-click **Wall** and rename it to `Particle_inlet`.
   You will use this surface as a reference geometry to create a particle factory in EDEM. Hence, you are placing this surface in a different geometry group.
9. Click ✓ on the guide bar.
10. Save the model.

Generate the Mesh

To focus on the solver setup, the mesh settings are predefined in the input file given to you.

1. From the Mesh ribbon, click the **Volume** tool.

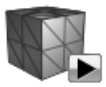


Figure 1312:

The **Meshing Operations** dialog opens.

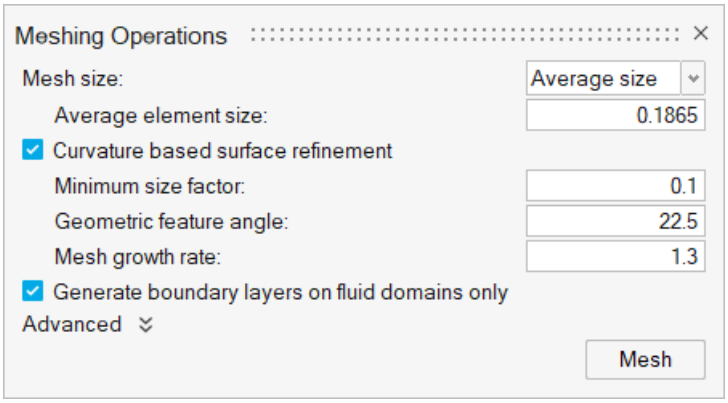


Figure 1313:

2. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

3. Save the model.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1314:

The **Launch AcuSolve** dialog opens.

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Click **Run** to launch AcuSolve.

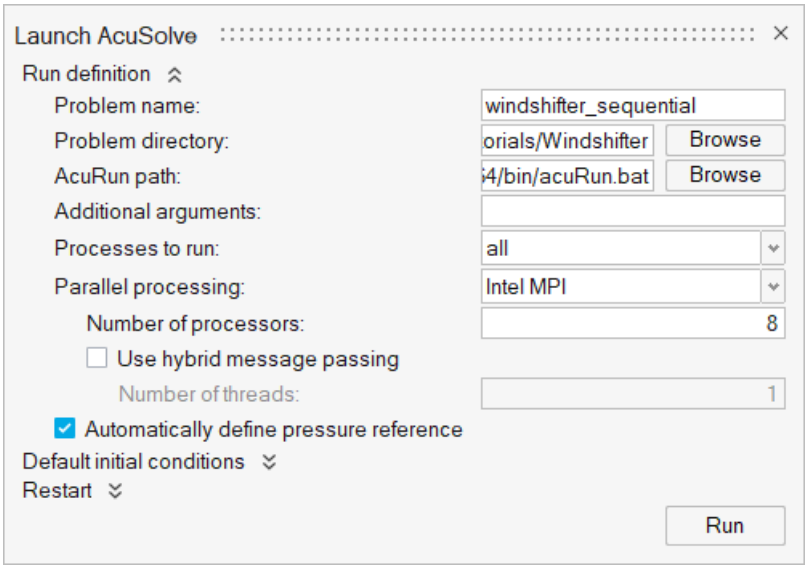


Figure 1315:

Once the AcuSolve simulation is launched, HyperMesh CFD also exports the EDEM files which contain the geometry. The surfaces are organized based on the CFD setup.

Export Velocity Field Data

1. From the Solution ribbon, click the **Convert** tool.

Figure 1316:

The **AcuSolve Data Translator** dialog opens.
2. In the dialog, click **Browse**, navigate to the AcuSolve problem directory, and open the `windshifter_sequential.1.Log` file.
3. Set the Output format to **CGNS**.
4. Select **velocity** from the list of variables.
5. Click **Convert** to export the velocity nodal data in CGNS format.

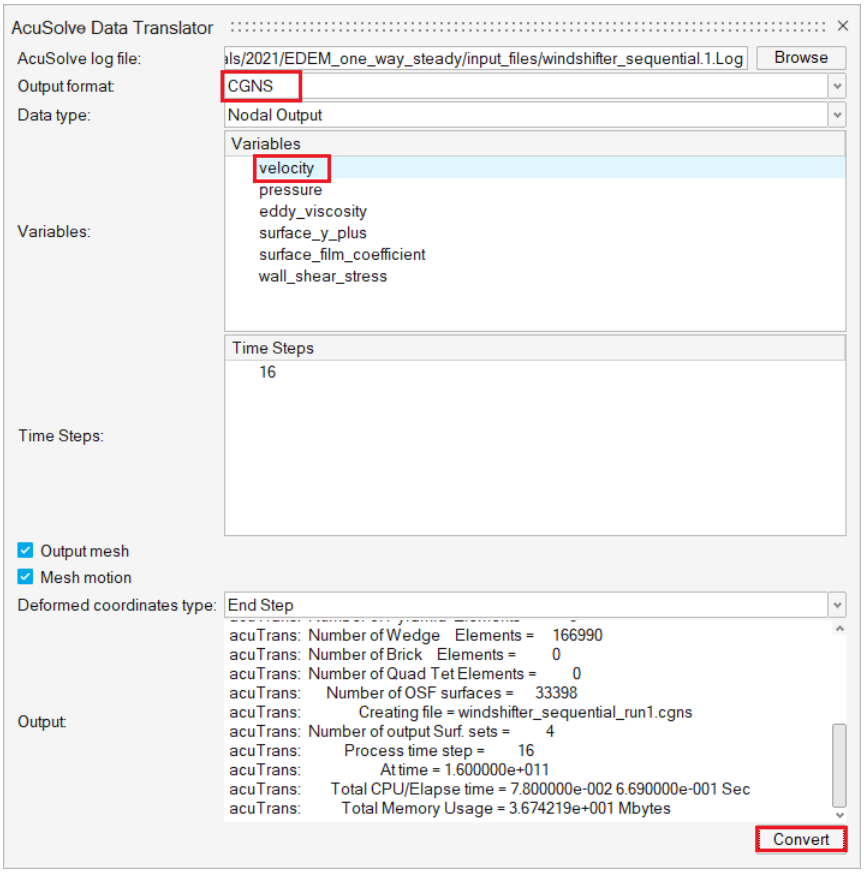


Figure 1317:

Once the process is successful, the same information is printed in the Output field and a CGNS file is created in the AcuSolve problem directory.

Post-Process the Results with HM-CFD Post

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Select the **Left** face on the view cube to align the model to the x-z plane.

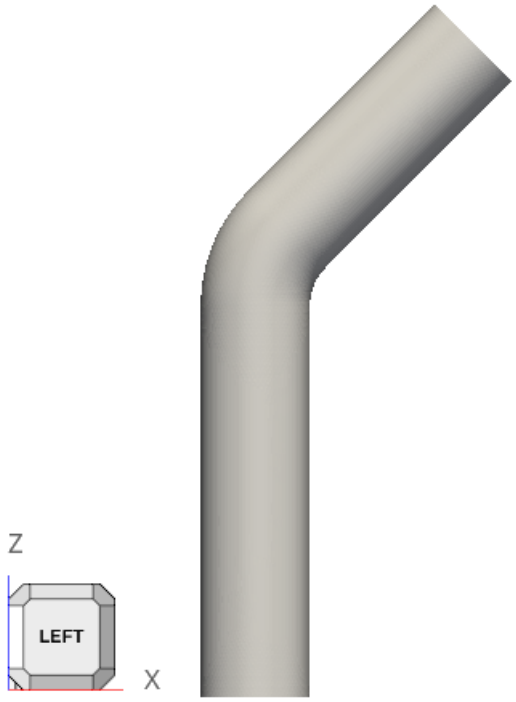


Figure 1318:

5. In the Post Browser, turn off the display of the boundary surfaces by clicking on the icon next to Flow Boundaries.

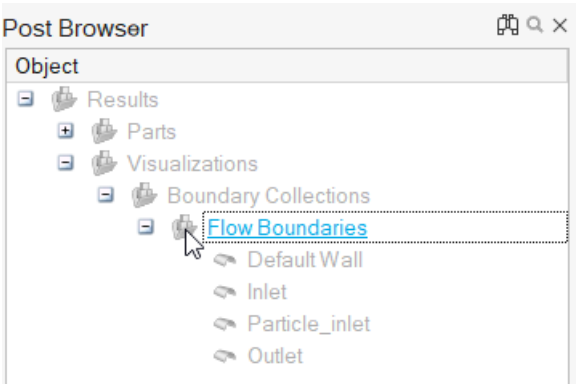


Figure 1319:

6. Click the **Slice Planes** tool.

Figure 1320:

7. In the modeling window, click on the plane that is parallel to the screen (the x-z plane).
8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
10. Click and set the Colormap name to **Rainbow Uniform**.

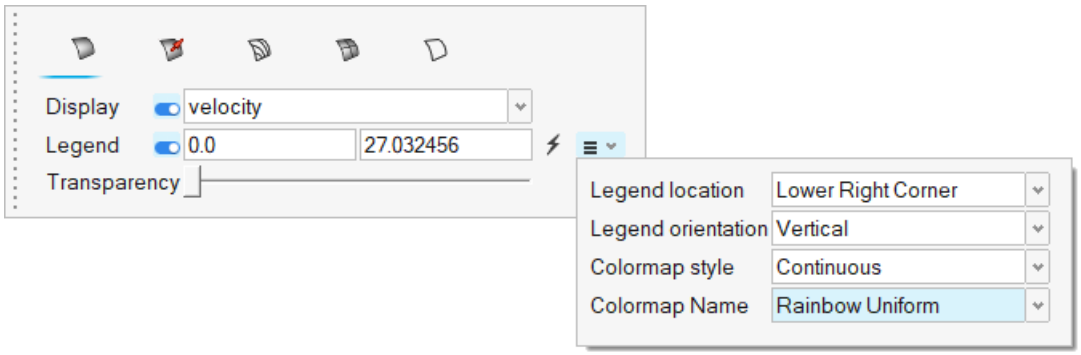


Figure 1321:

11. Click on the guide bar.

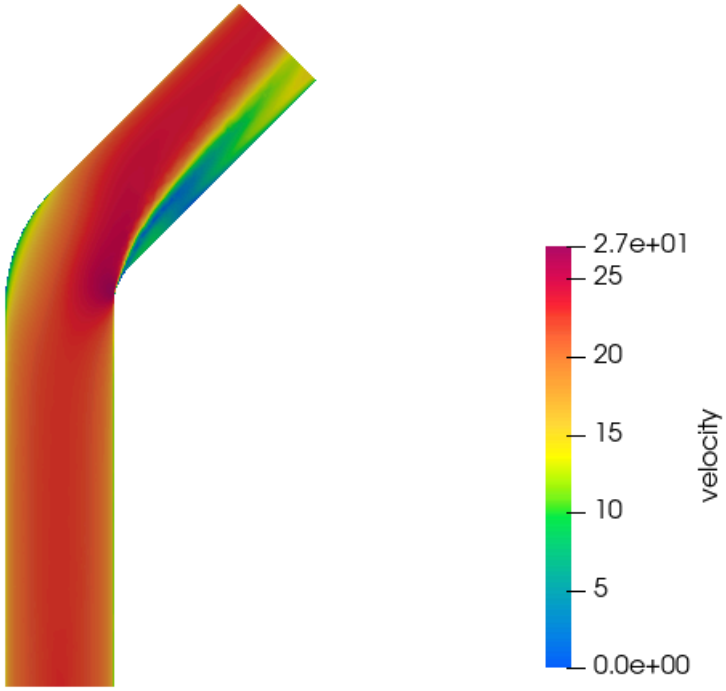


Figure 1322:

Part 2 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** . The user interface in EDEM is divided into three tabs: Creator, Simulator and Analyst. The Creator is used to setup and initialize your model. It is where you import particles and geometries and define the other model parameters. The Simulator is where you configure and control the EDEM simulation engine, and where you can observe the progress of your simulation. The Analyst is the post-processor used to analyze and visualize the results of your simulation.

The steps required to setup and run a basic CFD-DEM one-way coupling are explained below. Refer to the EDEM help documentation for more details.

Open the EDEM Input Deck

As mentioned earlier, when the AcuSolve simulation was launched, HyperMesh CFD created a set of EDEM files in the problem directory. You will open that EDEM input deck and setup the DEM simulation.

1. In the Creator tab in EDEM, go to **File** > **Open**.
2. In the dialog, browse to the AcuSolve problem directory and open the `windshifter_sequential.dem` file located in the EDEM folder.
   The geometry is loaded.
3. Click the **Environment** tab under the Creator Tree, uncheck **Auto Update from Geometry**, and then check the box again to fit the geometry within the boundary.

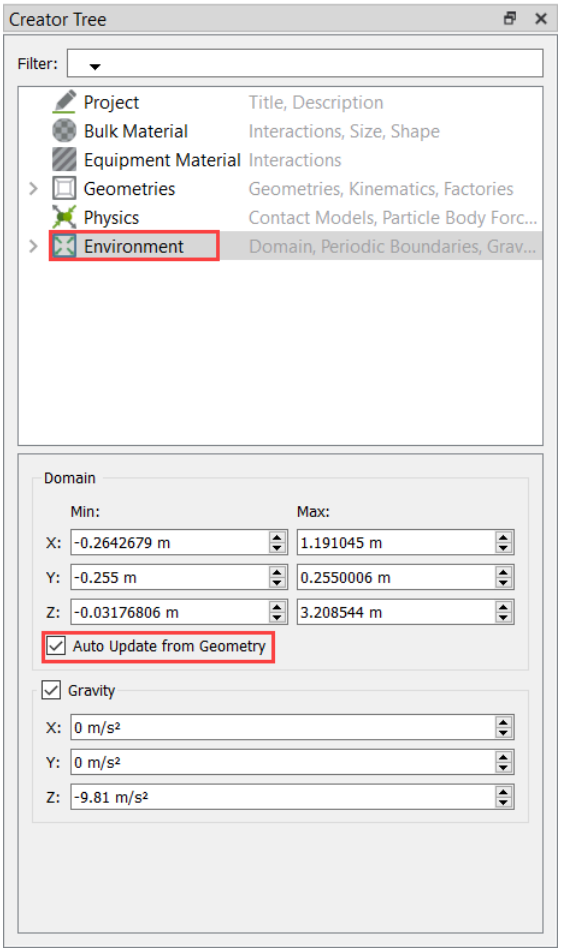

Creator Tree
Filter:
Project
Title, Description
Bulk Material
Interactions, Size, Shape
Equipment Material
Interactions
Geometries
Geometries, Kinematics, Factories
Physics
Contact Models, Particle Body Forc...
Environment
Domain, Periodic Boundaries, Grav...
Domain
Min:
Max:
X:
-0.2642679 m
1.191045 m
Y:
-0.255 m
0.2550006 m
Z:
-0.03176806 m
3.208544 m
Auto Update from Geometry
Gravity
X:
0 m/s²
Y:
0 m/s²
Z:
-9.81 m/s²

Figure 1323:

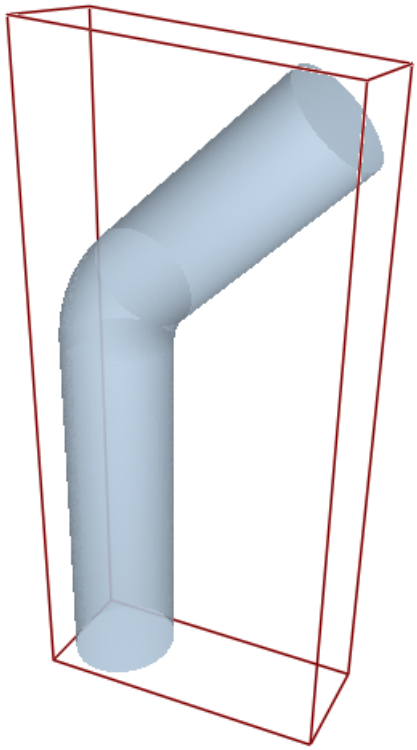


Figure 1324:

Define the Bulk Materials and Equipment Material

In this step, you will define the material models for the heavy and light bulk material and the equipment material.

1. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.
2. Rename the material to `Heavy`.
3. In the Creator Tree, set the Solids Density property to `900` kg/m^3.
   You will use the default values for other properties for this tutorial.

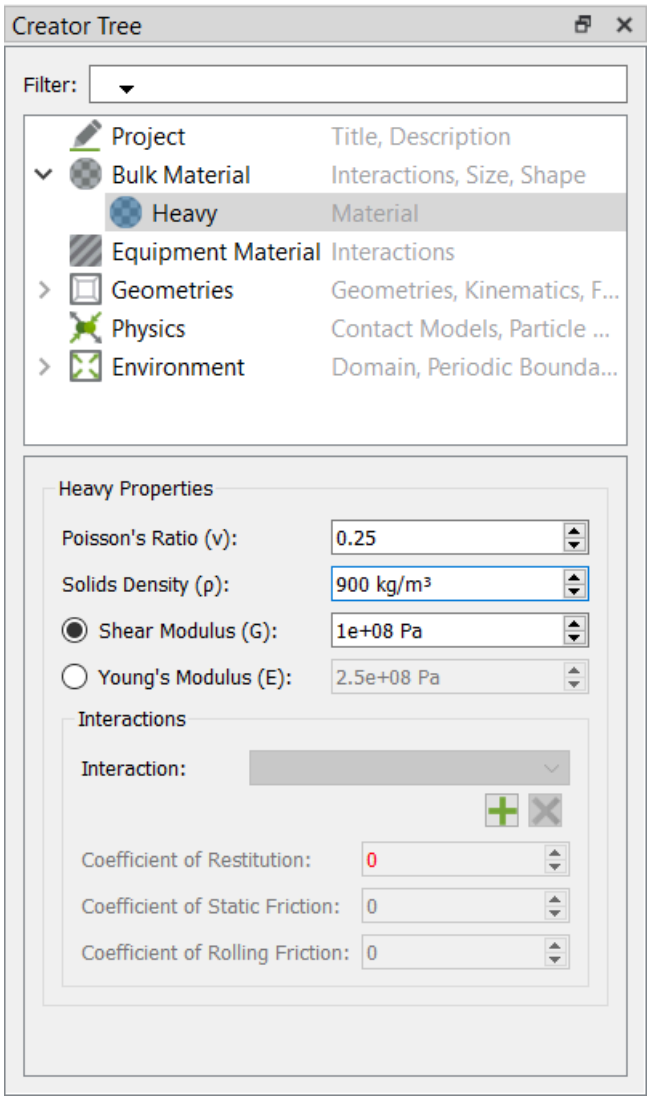


Figure 1325:

4. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, click **OK**.
5. In the Creator Tree, right click **Heavy** and select **Add Shape from Library** > **Dual Sphere Shape**.
6. Rename the particle to `Heavy particle`.
7. Under Heavy particle, click **Properties**.
8. In the Heavy particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

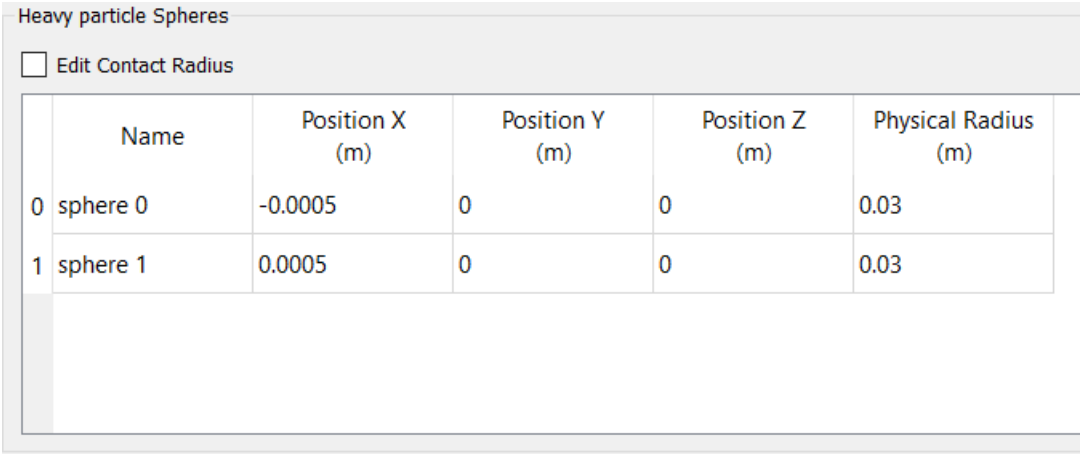


Figure 1326:

9. In the Creator Tree, click **Calculate Properties**.

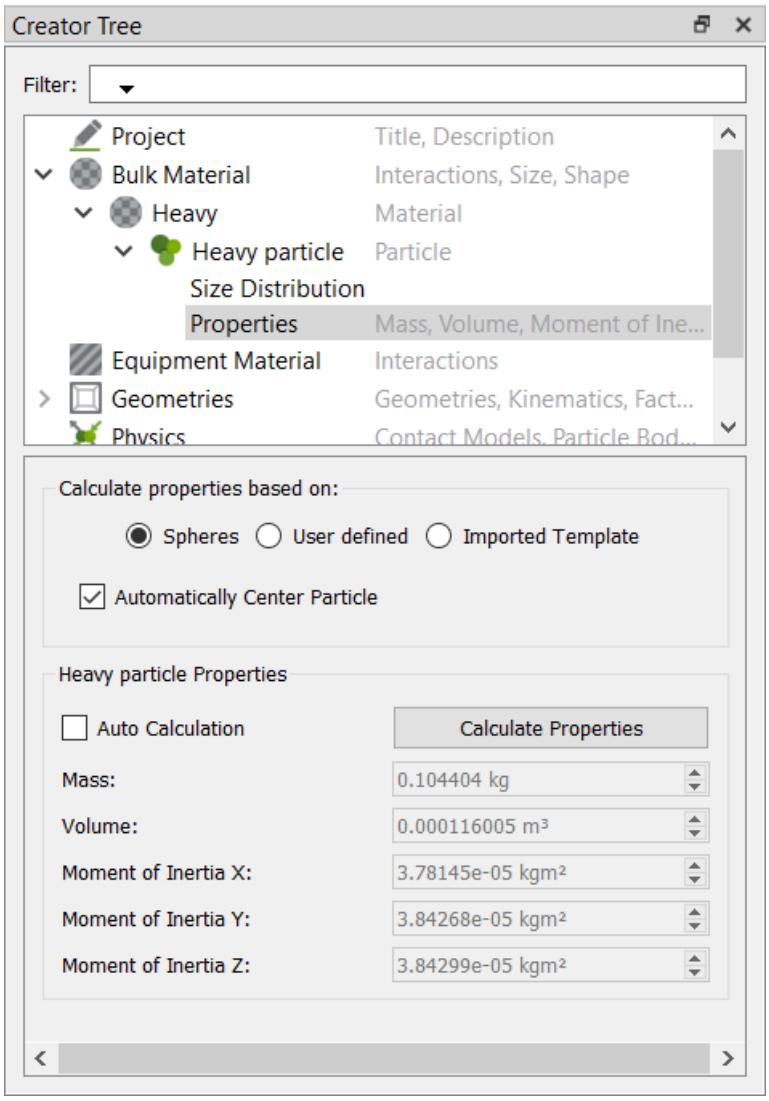


Figure 1327:

10. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.

11. Rename the material to `Light`.

12. In the Creator Tree, set the Solids Density property to `100` kg/m^3.

You will use the default values for other properties for this tutorial.

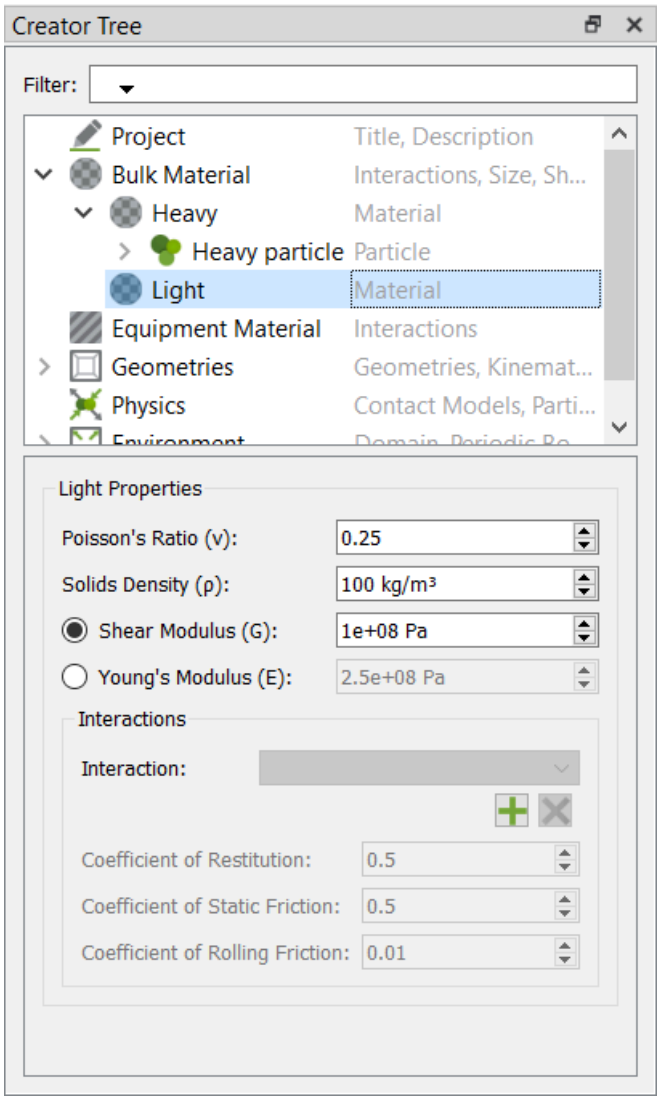


Figure 1328:

13. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.
14. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.
15. In the Creator Tree, right click on **Light** and select **Add Shape from Library** > **Dual Sphere Shape**.
16. Rename the particle to `Light particle`.
17. Under Light particle, click **Properties**.
18. In the Light particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

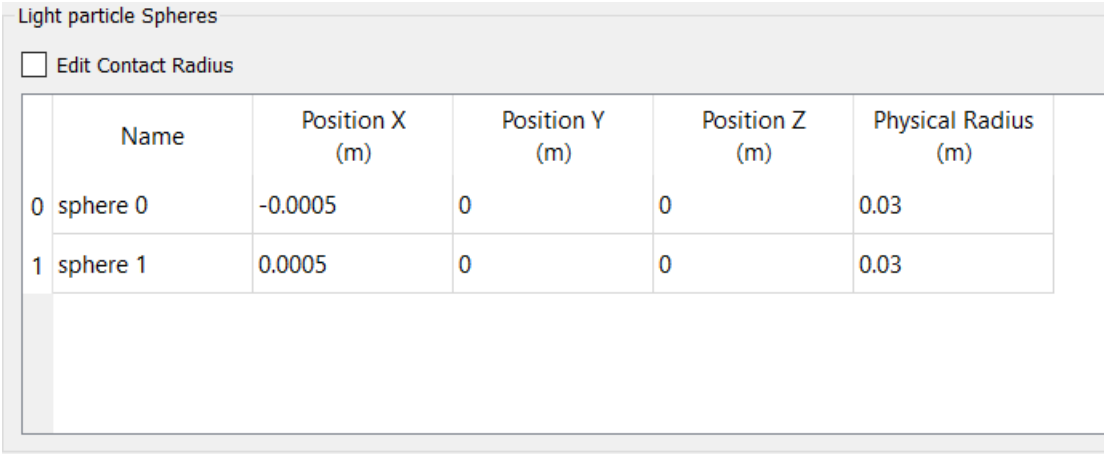


Figure 1329:

19. In the Creator Tree, click **Calculate Properties**.

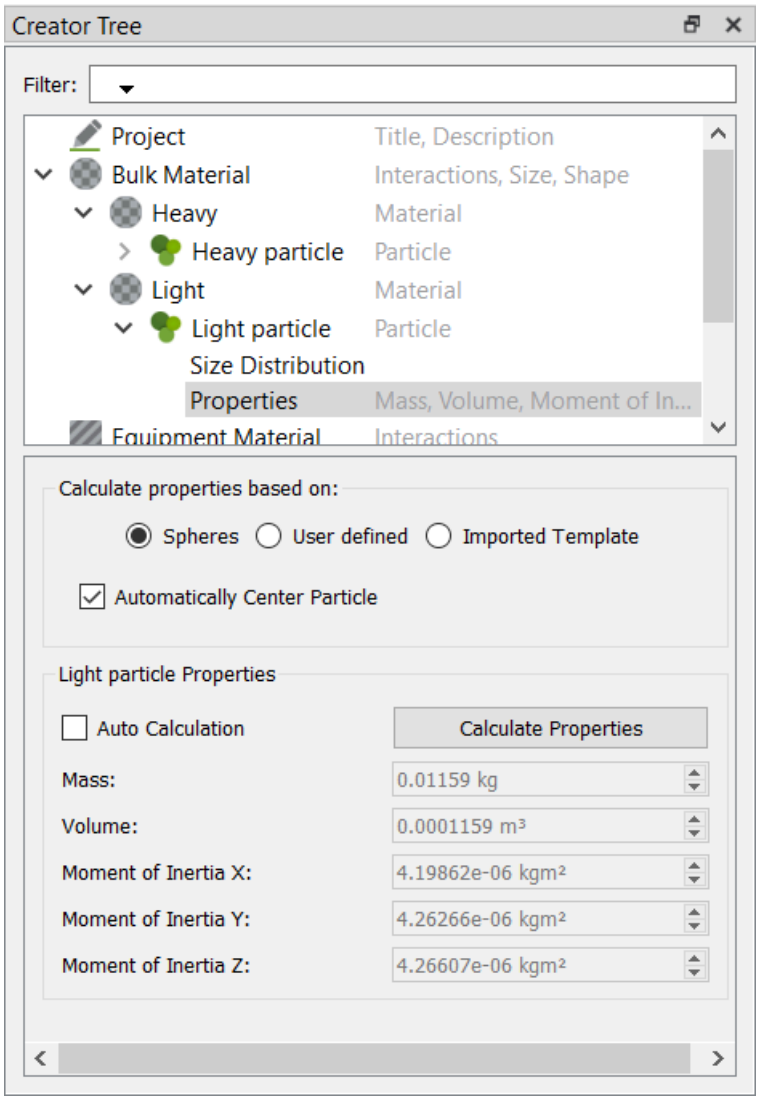


Figure 1330:

20. In the Creator tree, right-click **Equipment Material** and select **Add Equipment Material**. Rename it to `Steel`.

21. Set the Density to `7800` kg/m^3.

22. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.

23. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.

24. Save the model.

Create the Particle Factory

1. Expand **Geometries** under the Creator Tree tab. Next, right-click the **Particle inlet** surface group and select **Copy Geometry** > **Single Copy**.
2. Rename the new geometry section to `Particle_factory`.
3. Right-click the **Default Wall** geometry section in the Creator Tree and select **Merge Geometry(s)**.
4. In the **Merge Geometry** dialog, select the **Particle_inlet** and then click **OK**.
5. Click the **Default Wall** geometry section, set the Type to **Physical**, and the Material to **Steel** (if not set already).

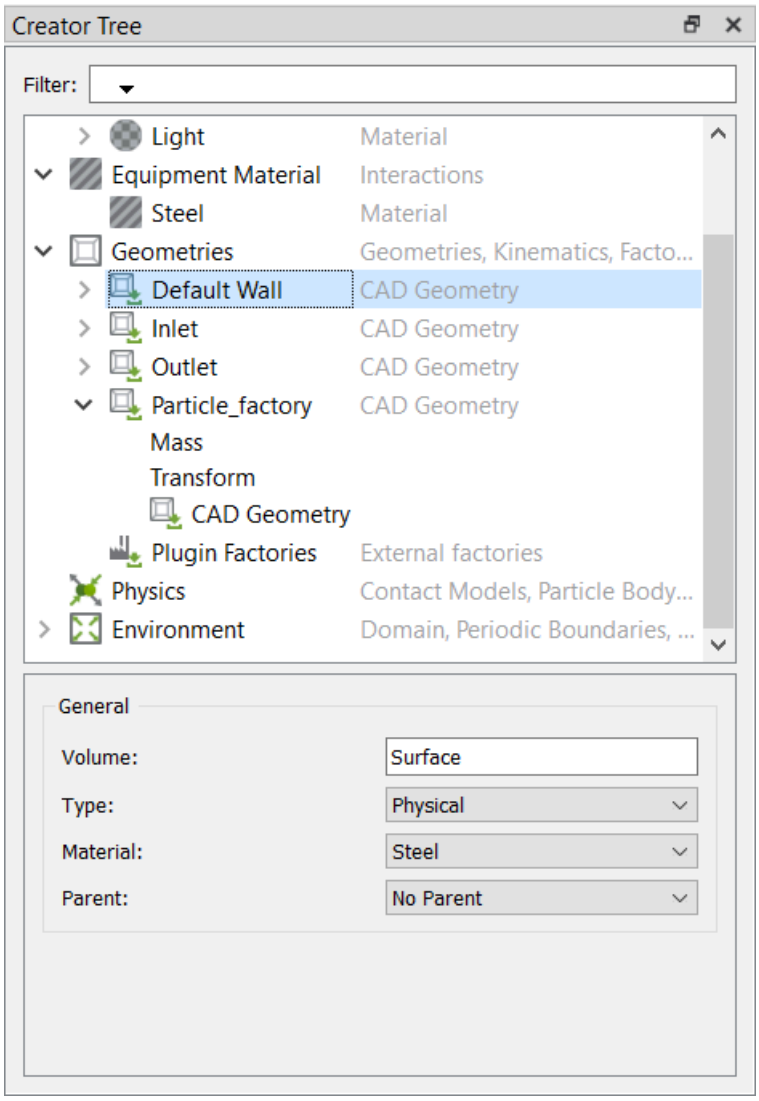


Figure 1331:

6. In the Creator Tree, click the **Inlet** section and change the Type to **Virtual**.
7. Similarly, change the Type to **Virtual** for the Outlet and Particle_factory sections.
8. Under Particle_factory, click **Transform**. Set the X-Position to `0.035` m

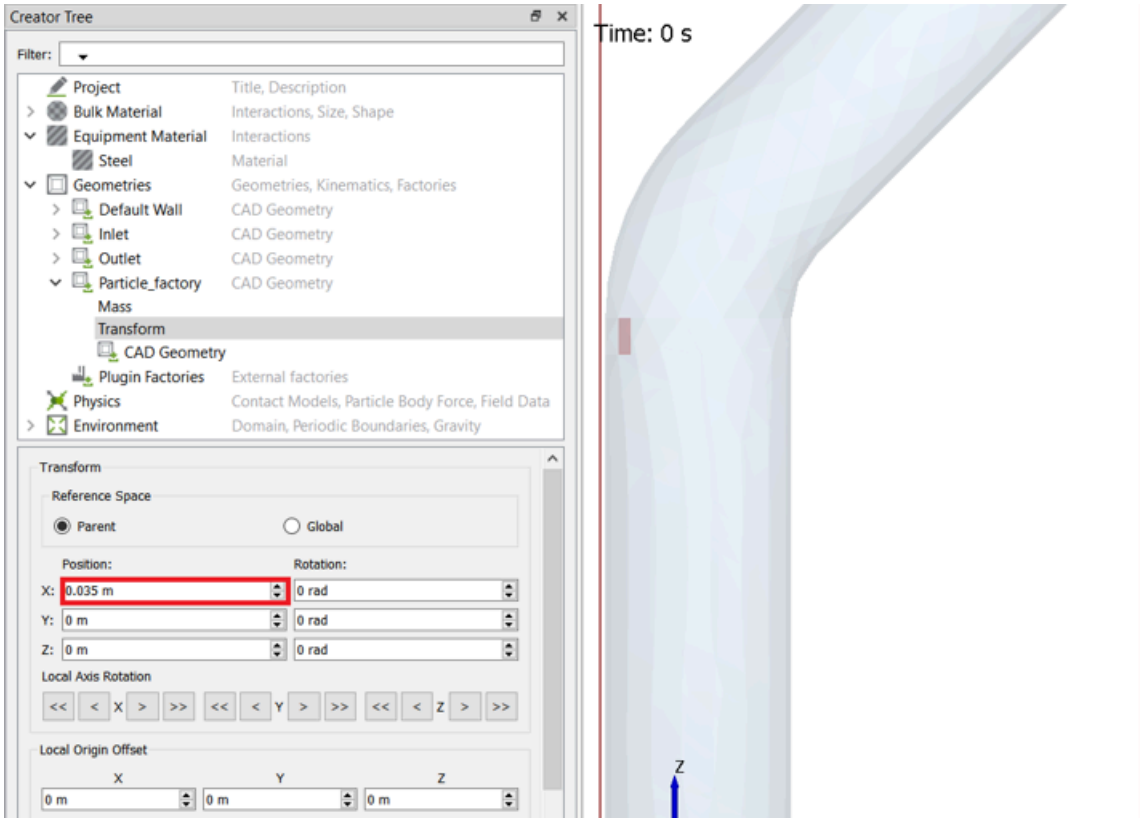


Figure 1332:

> **Note:** Set the Opacity value to 0.2 to see the transformed surface location inside the pipe geometry.
>
> This is done to make sure that the particles are generated inside the fluid domain.

Define the Particle Factory

Now that the bulk material, geometry sections, and equipment materials are defined, you need to create a particle factory to generate the particles. You will create one factory for each bulk material.

1. In the Creator Tree, right-click **Particle_factory** and select **Add Factory**.
2. Rename the new factory to `Heavy factory`.
3. Set the particle generation parameters as shown in the figure below.

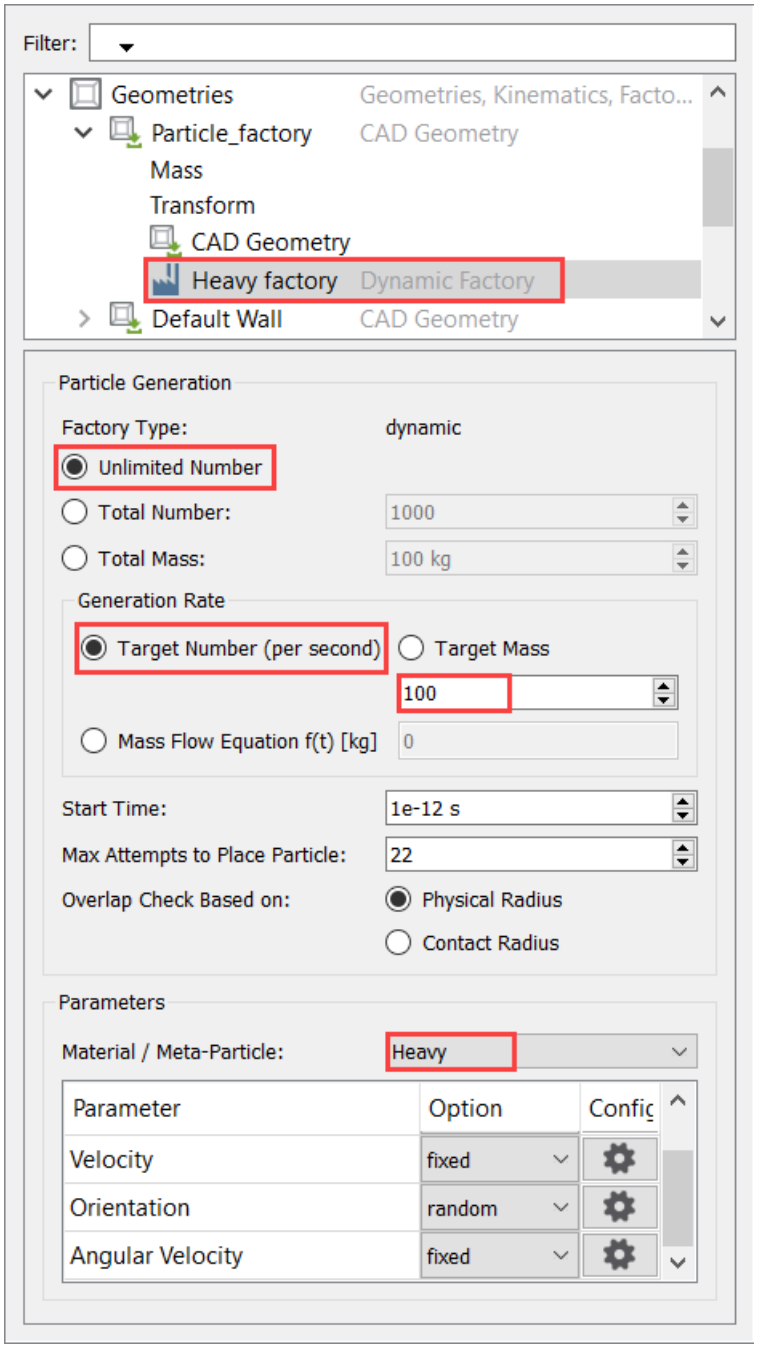


Figure 1333:

4. Click ⚙ besides Velocity, set the X-velocity to `1` m/s, and then click **OK**.
5. Repeat steps 1-4 to create another factory named `Light factory` using the same parameters but with **Light** as the Material.

Define the Physics and Import CFD Field Data

In this step, you will define the physics models for particle collisions and the particle body force.

1. In the Creator Tree, click **Physics**.
2. Click the **Interaction** drop-down menu and select **Particle Body Force**.

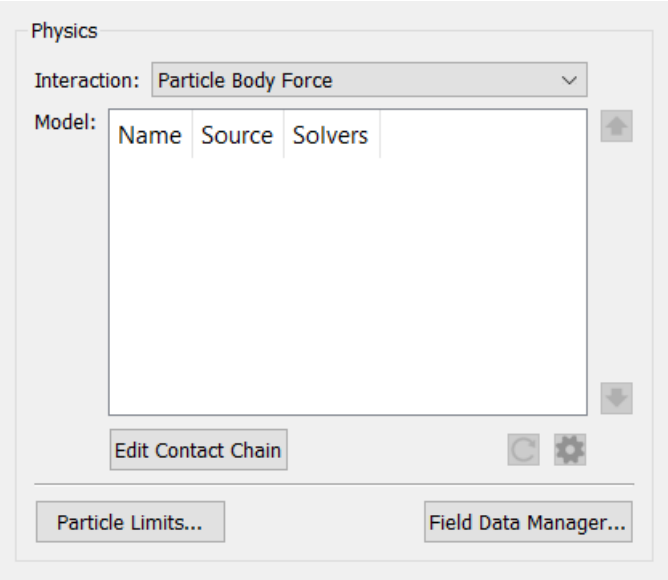


Figure 1334:

3. Click **Edit Contact Chain**.
4. In the dialog, select the **Schiller and Naumann Drag** model and then click **OK**.
5. In the Physics tab, click **Schiller and Naumann Drag** and then click .
6. In the **Schiller and Naumann Drag Parameter Values** dialog, enter the values as shown in the figure below and then click **OK**.

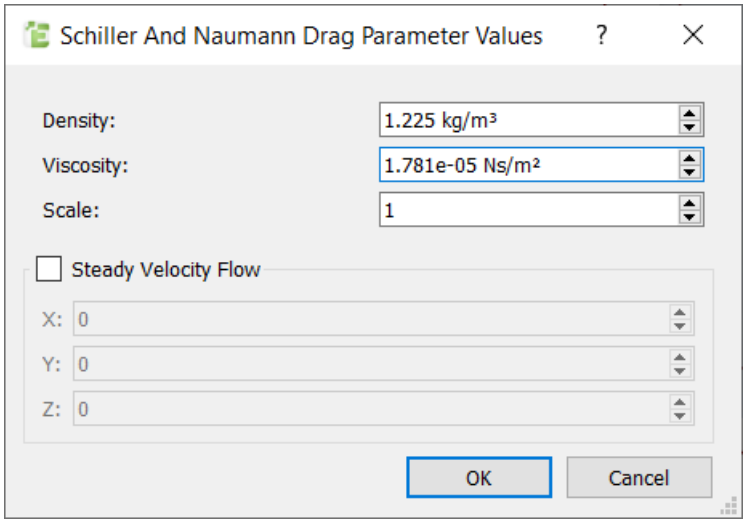


Figure 1335:

7. In the Physics tab, click the **Field Data Manager**.
8. In the **Field Data Manager** dialog, click then browse to the AcuSolve problem directory where you saved the CGNS file exported using the CFD field data. Open `windshifter_sequential_run1.cgns`.
9. Once the data import is complete, double-click **Velocity** and rename it to `velocity`. (The letter 'v' should be in lower case). Then, close the dialog.

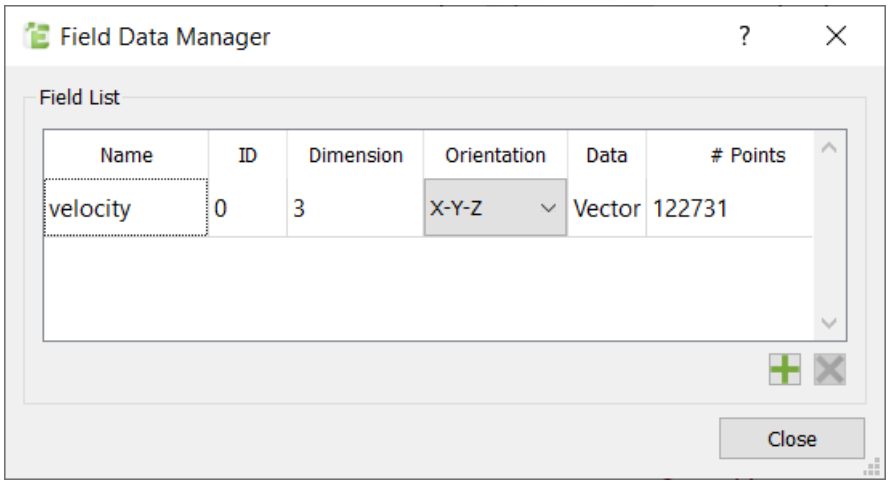


Figure 1336:

10. Save the EDEM deck.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).
   When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.
3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings and Run the Simulation

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and activate the **Auto Time Step** checkbox (if not set already).
3. Set the Total Time to `1` s and the Target Save Interval to `0.01` s.
4. Set the Cell Size to `4` R min.

   Generally, a value in the range of 3-6 R_{min} is recommended as the optimum cell size. The cell size in EDEM doesn't have any impact on the accuracy of the simulation and affects only the run time.
5. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

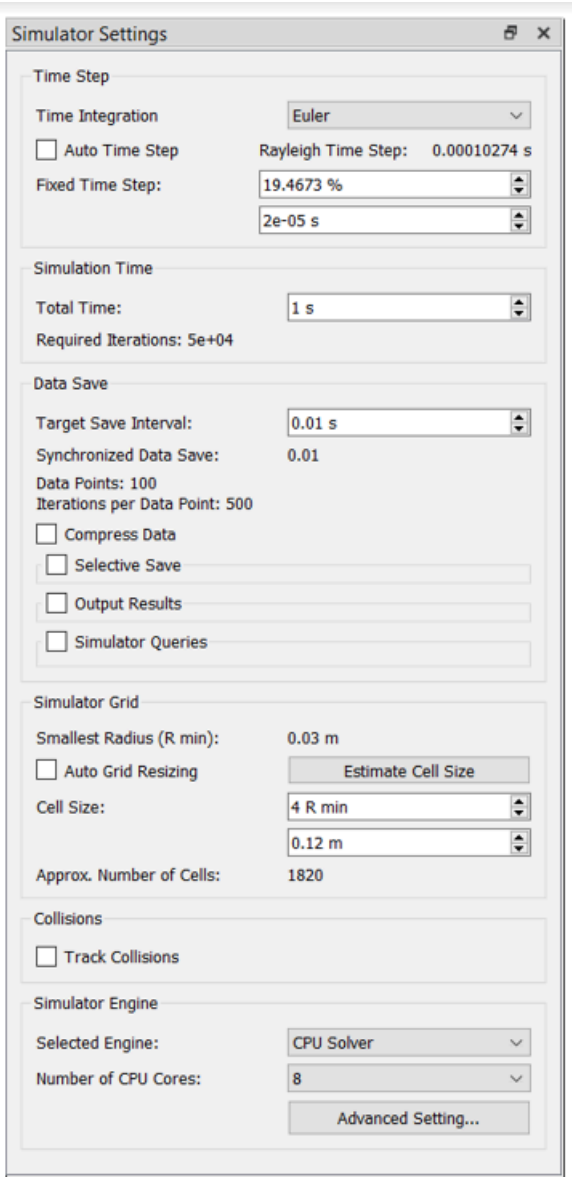


Figure 1337:

6. Once the simulation settings have been defined, click to start the EDEM simulation.

Analyze the Results

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then click **Default Wall**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

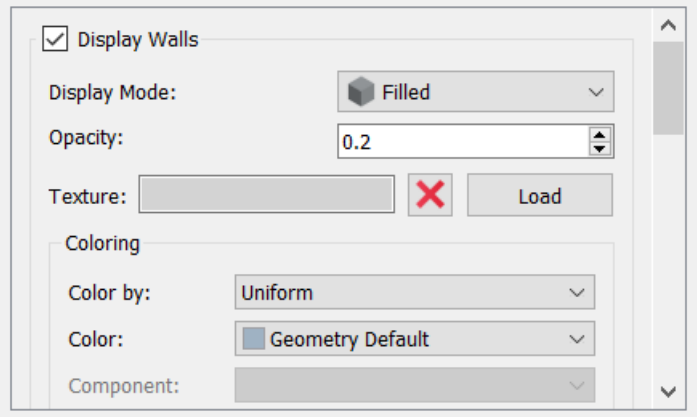


Figure 1338:

4. In the Analyst Tree, expand **Particles** and click **Heavy particle**.
5. Change the display color to **Magenta**.

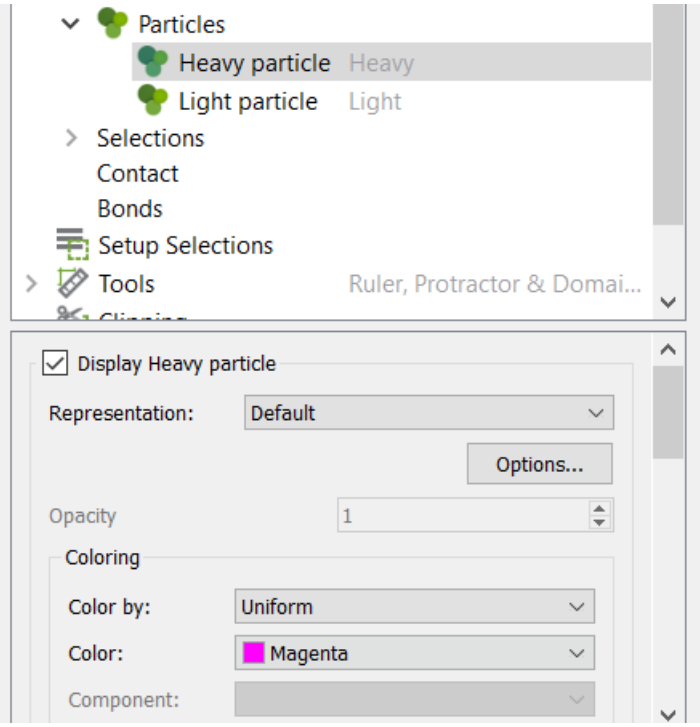


Figure 1339:

6. Click **Light particle** and set the display color to **Green**.

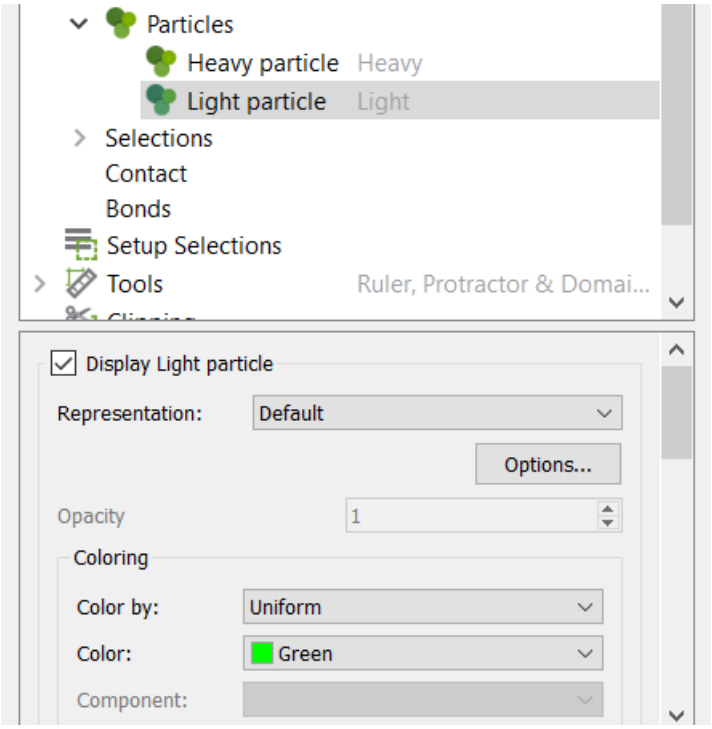


Figure 1340:

7. On the menu bar, set the time to `0` by clicking:

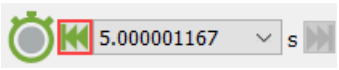


Figure 1341:

8. Set the View plane to **+ Y**.

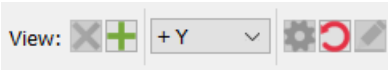


Figure 1342:

9. In the Viewer window, set the Playback Speed to `0.1x` then click on the play icon to play the particle flow animation.

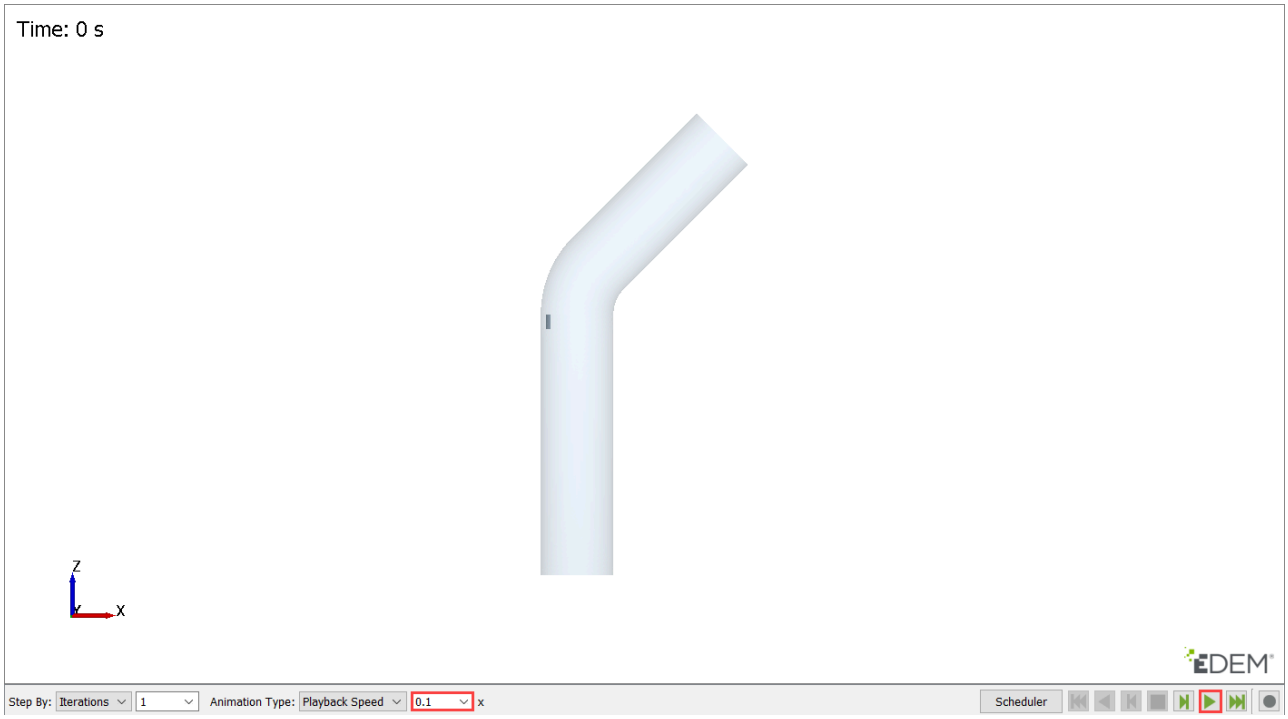


Figure 1343:

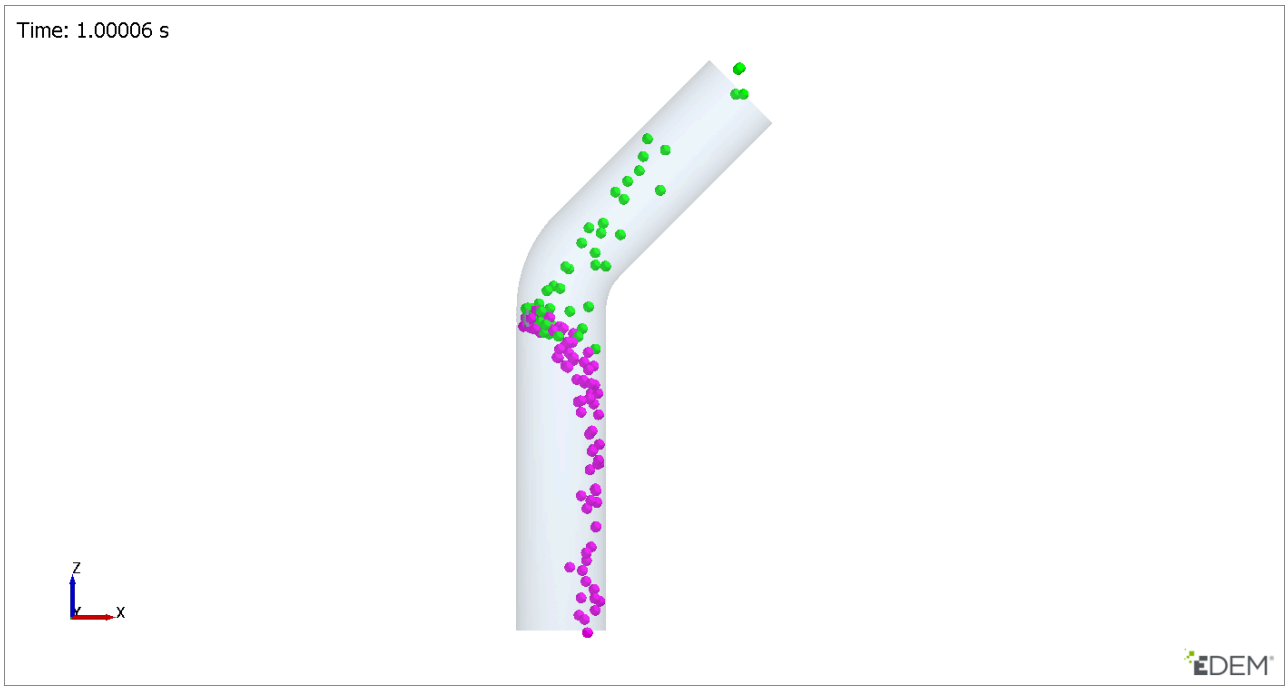


Figure 1344:

Observe that the lighter particles (green) get carried by the fluid and escape the domain through the outlet at the top and the heavier particles (magenta) stay inside the domain for a longer time while some of them fall through the bottom of the pipe.

Summary

In tutorial you learned how to setup a basic AcuSolve-EDEM sequential (one-way steady) coupling problem. In the first part, you set up and solved a steady state flow simulation using AcuSolve and then exported the CFD field data using AcuTrans. Next, you set up the EDEM model and imported the CFD field data using the Field Data Manager. Once the EDEM simulation was completed, you learned how to view results and create animations in EDEM Analyst.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/5i1tw8ekab

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6101 Particle Separation in a Windshifter using AcuSolve - EDEM Unidirectional Coupling

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up and running a basic unidirectional coupling (one-way steady) simulation using AcuSolve and EDEM. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6101_windshifter.hm`

Problem Description

The problem to be solved is shown schematically in Figure 1345. It is a windshifter model in which air enters the domain from the inlet at the bottom at a very high velocity (20 m/sec) and exits through the outlet. When particles are introduced into the domain through the particle inlet, the lighter particles get carried away with the air and the heavier particles exit through the opening at the bottom. Since the concentration of the particles is sparse, the effect of the particles on the fluid field is not considered for this simulation. Hence, one-way coupling between AcuSolve and EDEM is used to simulate the separation of particles; only the effect of the fluid forces on the particles is considered.

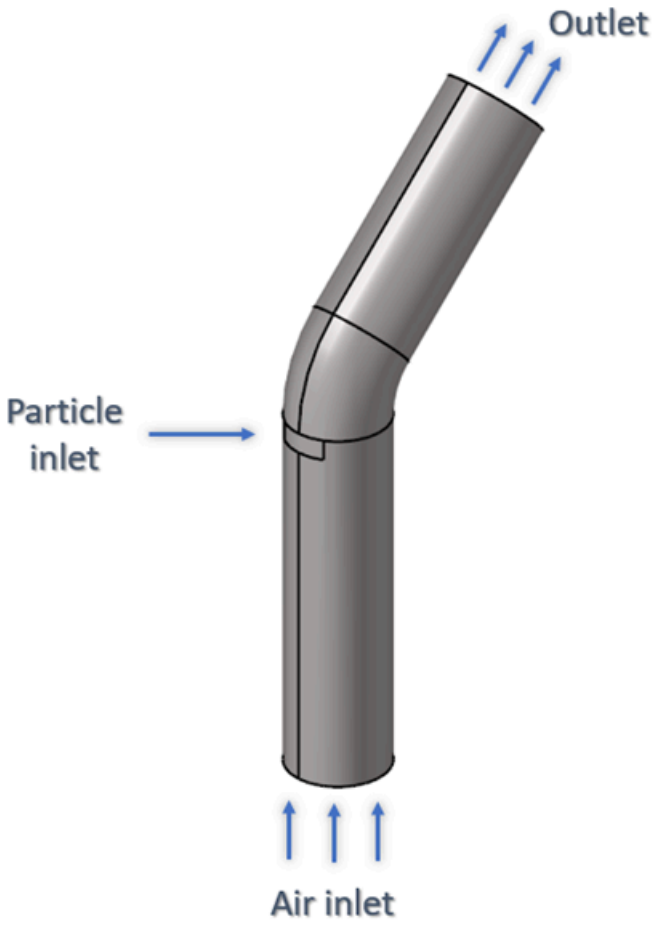


Figure 1345:

The model consists of a cylindrical pipe with a 45-degree bend. The radius of the pipe is 0.25 m and the particle inlet is located midway through the length of the pipe.

The workflow for an AcuSolve-EDEM unidirectional coupling simulation is shown below.

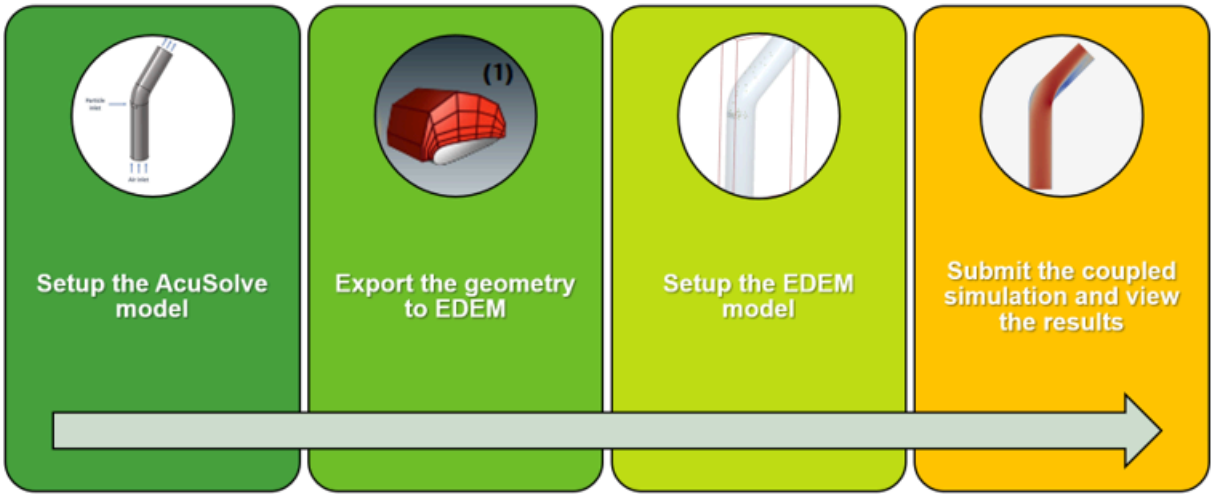


Figure 1346:

Accordingly, the tutorial consists of two parts:

1. AcuSolve setup and geometry export
2. EDEM setup and simulation

The AcuSolve model will be set up using HyperMesh CFD. Once the AcuSolve setup is complete, the EDEM deck with the geometry will be exported from HyperMesh CFD. This input deck will be opened in EDEM and will be used to complete the EDEM setup. Once the EDEM deck is set up, you will launch the coupled simulation.

Two different bulk materials used in the EDEM simulation and their properties are listed below:

Name	**Density (kg/m^3)**	**Particle radius (m)**	**Average weight of individual particle (kg)**	**Rate of generation (particles per sec)**
Heavy particle	900	0.03	0.1	100
Light particle	100	0.03	0.01	100

The fluid drag forces on the particles are calculated using the Schiller-Naumann drag model. AcuSolve receives the particle position and velocity information from EDEM and calculates the drag forces on the particles. This fluid forces information is sent back to EDEM, which will be used to update the location and velocities of the particles. This loop is repeated until the end of the simulation.

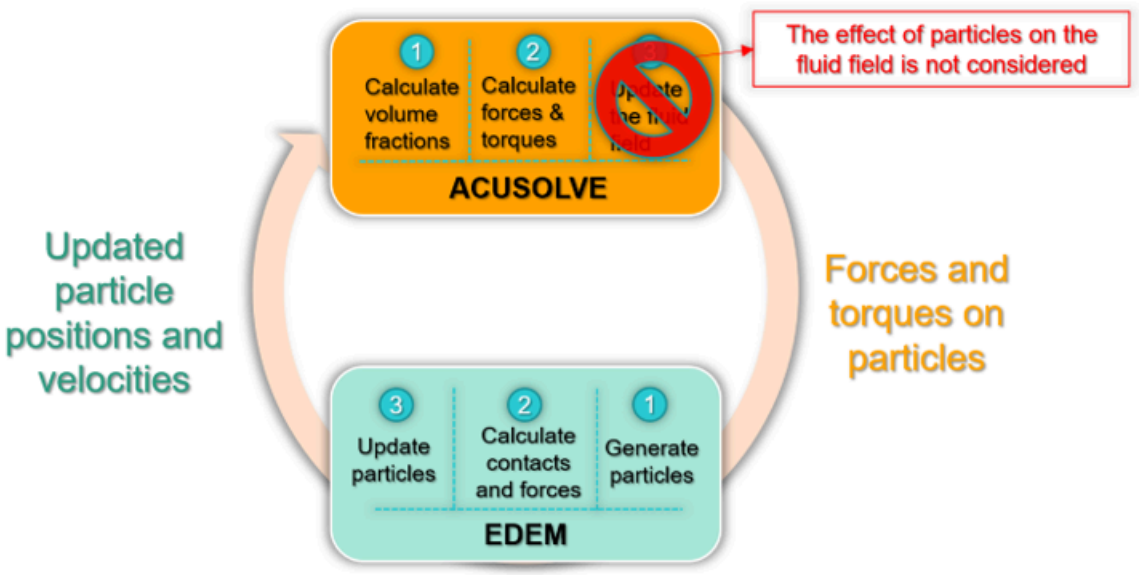


Figure 1347:

Part 1 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1348:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6101_windshifter.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Save the database as `windshifter_unidirectional` in the same directory as the other input files.

   This will be the working directory and all the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1349:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

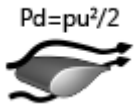
   

   Figure 1350:

   The **Setup** dialog opens.
2. Under the Physics models setting:
   a) Select the **Multiphase flow** radio button.
   b) Change the Multifluid type to **Unidirectional EDEM Coupling**.
   c) Set the Eulerian material to **Air-EDEM Particle -1way**, if not set already.
   d) Set Time step size and Final time to `0.001` and `1`, respectively.
   e) Select **Spalart-Allmaras** as the Turbulence model.
   f) Verify that Gravity is set to `0, 0, -9.81`.

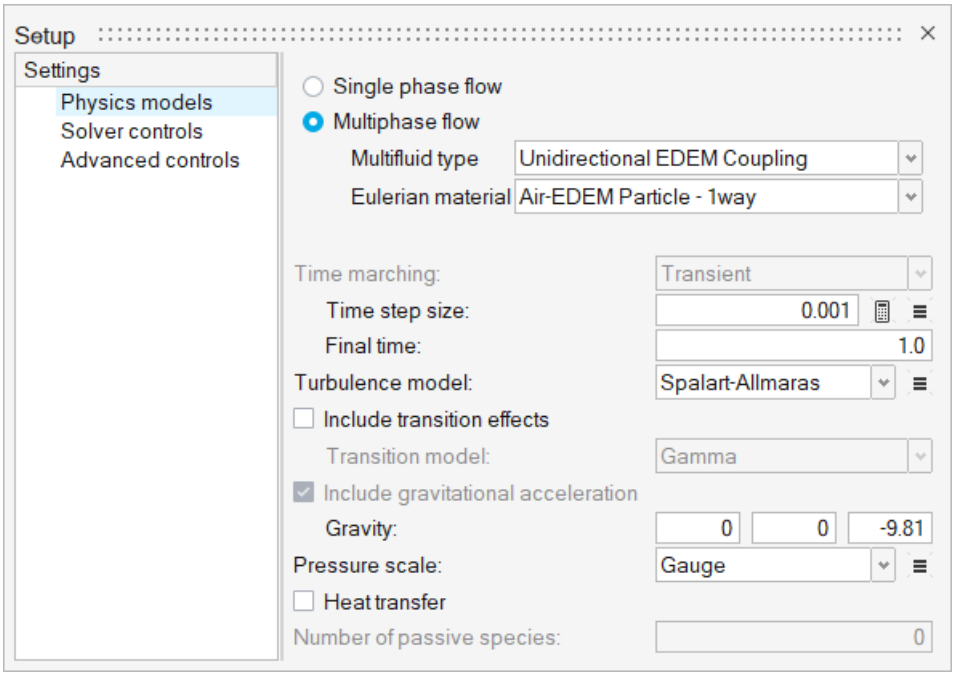


Figure 1351:

3. Click the **Solver controls** setting and set the Minimum and Maximum stagger iterations to `2` and `4`, respectively.

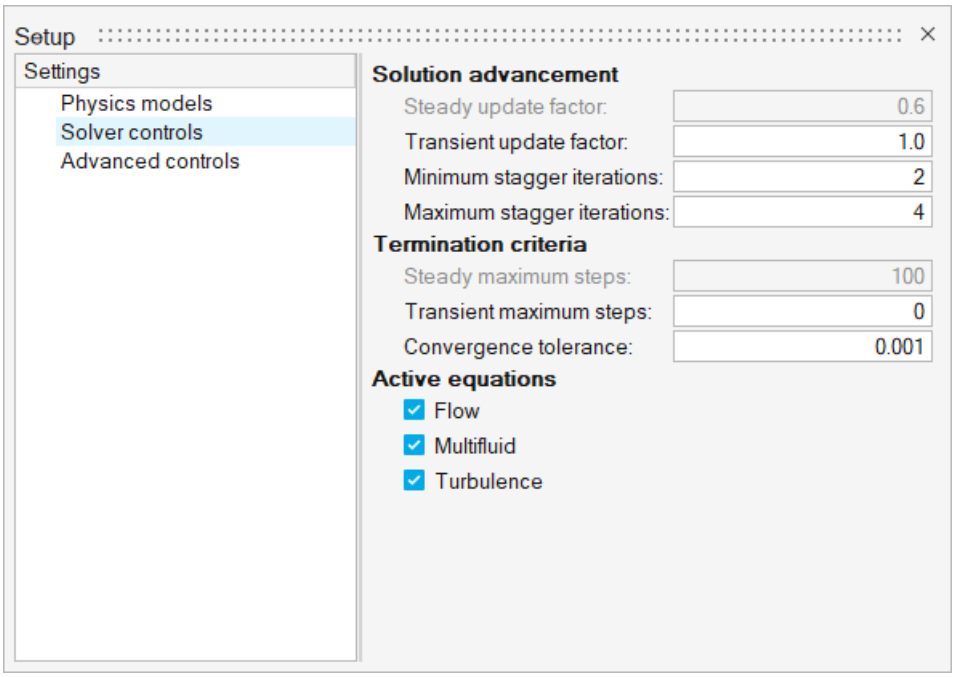


Figure 1352:

4. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1353:

2. Verify that Air-EDEM Particle - 1way has been assigned as the material.
3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1354:

2. Click on the inlet face highlighted in the figure below. In the microdialog, enter a value of `20` m/s for the Average velocity and set the Carrier fluid volume fraction to `1.0`.

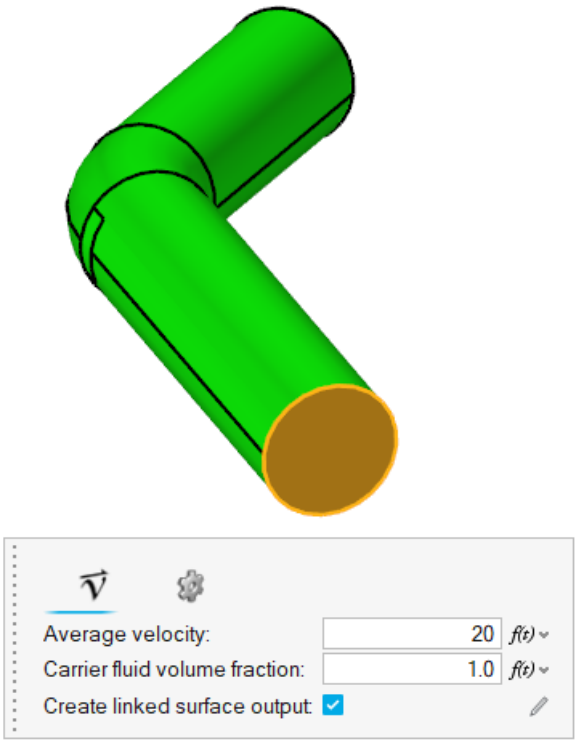


Figure 1355:

3. On the guide bar, click ✓ to execute the command and exit the tool.
4. Click the **Outlet** tool.

Figure 1356:

5. Select the face highlighted in the figure below and then click ✓ on the guide bar.

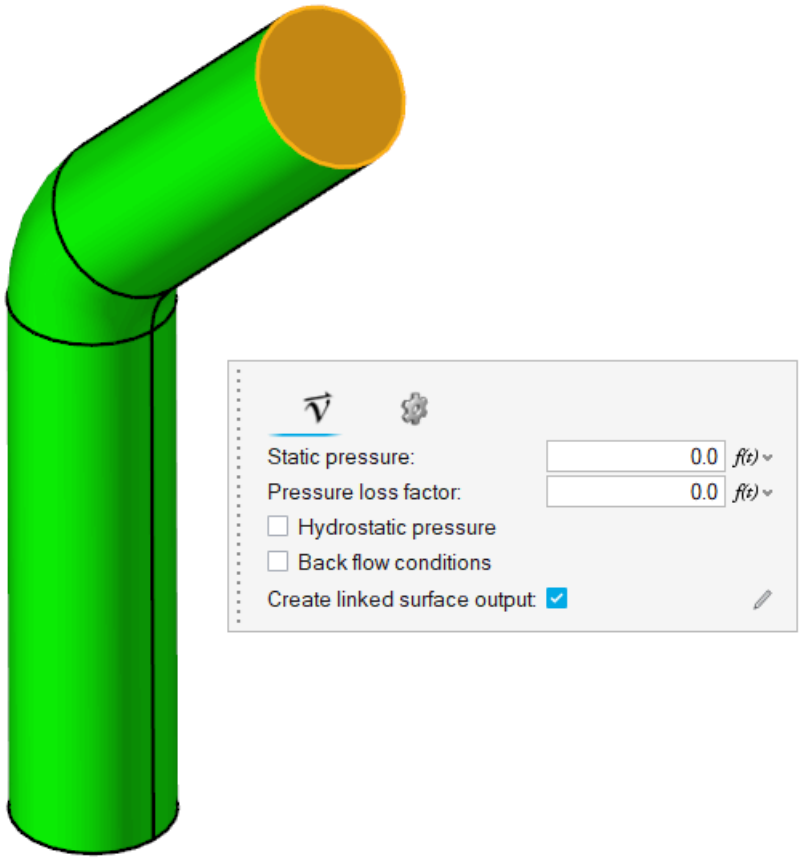


Figure 1357:

6. Click the **No Slip** tool.

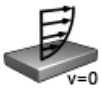


Figure 1358:

7. Select the face highlighted in the figure below.

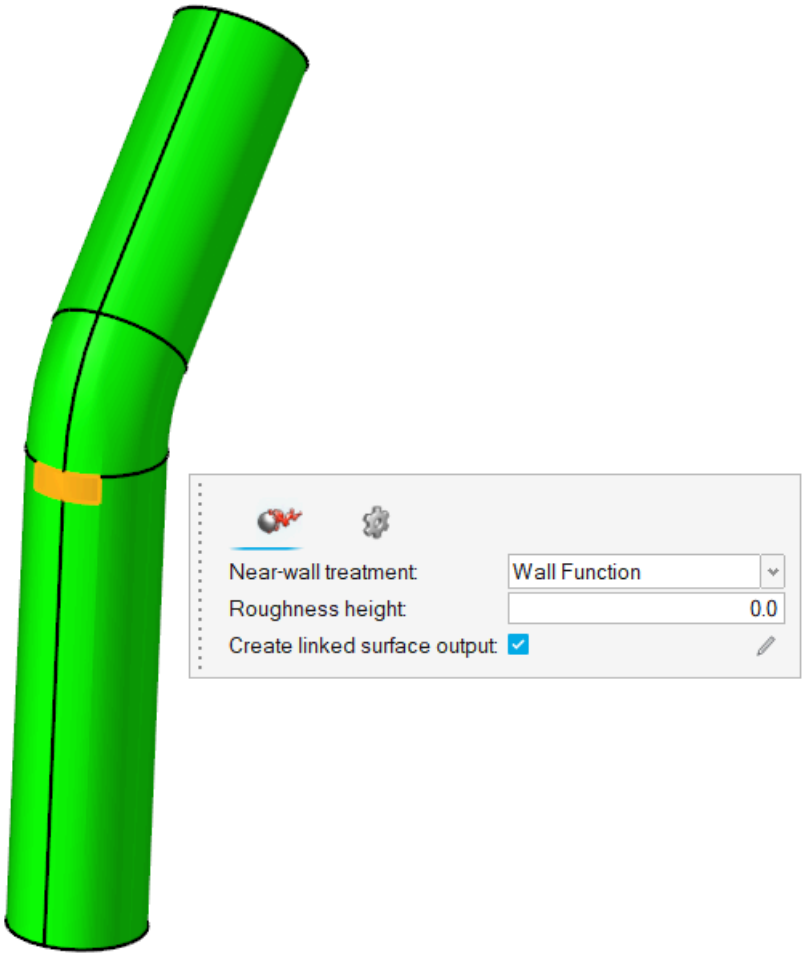


Figure 1359:

8. In the Boundaries legend, double-click **Wall** and rename it to `Particle_inlet`.
   You will use this surface as a reference geometry to create a particle factory in EDEM. Hence, you are placing this surface in a different geometry group.
9. Click ✓ on the guide bar.
10. Save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

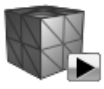


Figure 1360:

The **Meshing Operations** dialog opens.

2. Change the Average element size to `0.3`.

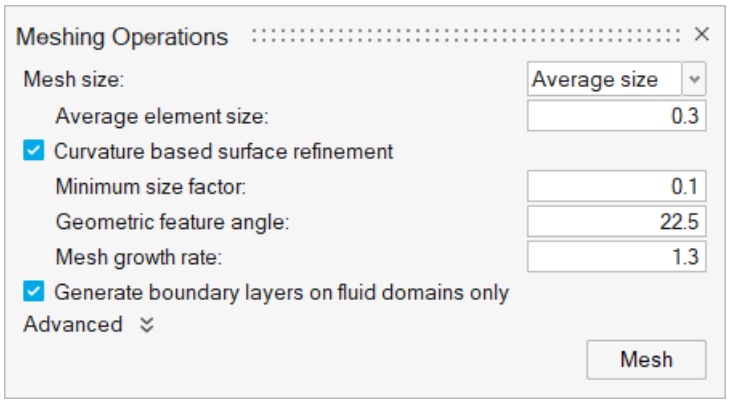


Figure 1361:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

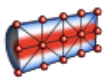


Figure 1362:

The **Field Output** dialog opens.

2. Check the box for **Write Initial Conditions**.
3. Set the Time step interval to `10`.

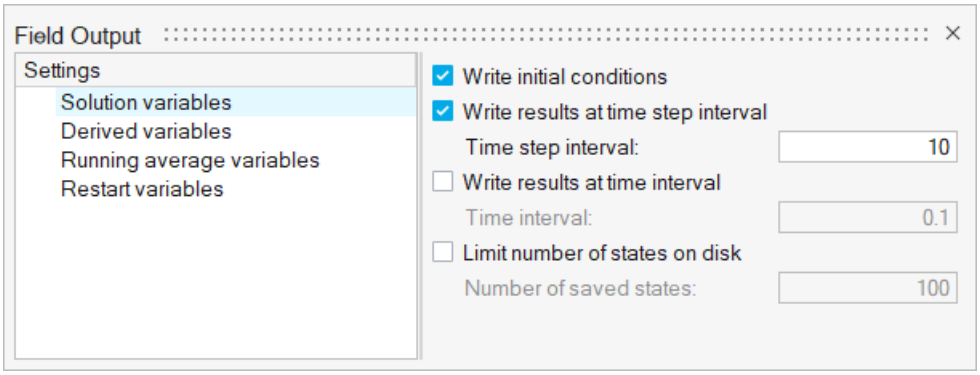


Figure 1363:

Export the Solver Deck

1. From the menu bar, go to **File** > **Export** > **Solver Deck**.
2. Name the file `windshifter_unidirectional` and make sure that **AcuSolve (*.inp)** is selected as the file type.
3. Click **Save**.

The AcuSolve input files and the EDEM input deck with the geometry sections are created. You will use this EDEM deck to set up the DEM simulation deck.

Part 2 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

As mentioned earlier, when the AcuSolve simulation was launched, HyperMesh CFD created a set of EDEM files in the problem directory. You will open that EDEM input deck and setup the DEM simulation.

1. In the Creator tab in EDEM, go to **File** > **Open**.
2. In the dialog, browse to the AcuSolve problem directory and open the `windshifter_unidirectional.dem` file located in the EDEM folder.
   The geometry is loaded.
3. Click the **Environment** tab under the Creator Tree, uncheck **Auto Update from Geometry**, and then check the box again to fit the geometry within the boundary.

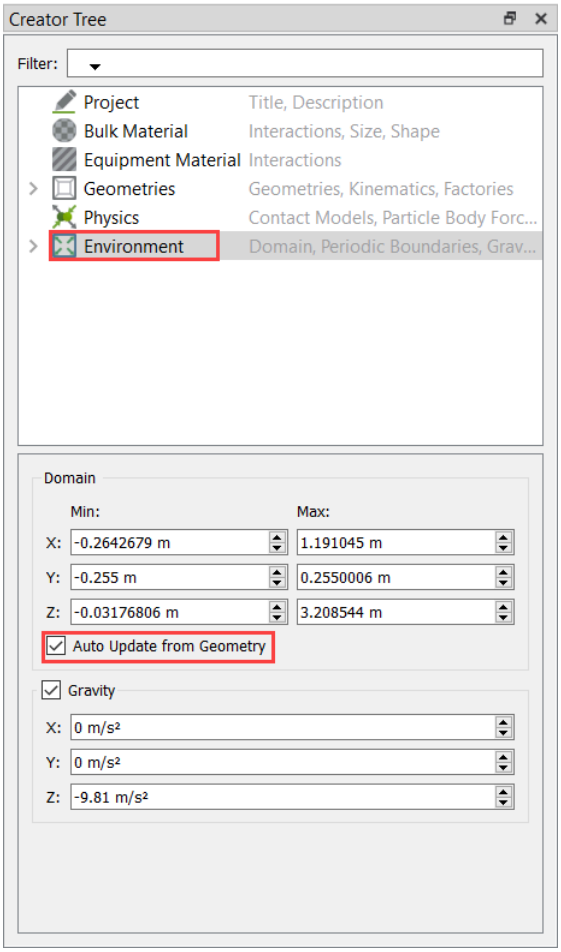

Creator Tree
Filter:
Project
Title, Description
Bulk Material
Interactions, Size, Shape
Equipment Material
Interactions
Geometries
Geometries, Kinematics, Factories
Physics
Contact Models, Particle Body Forc...
Environment
Domain, Periodic Boundaries, Grav...
Domain
Min:
Max:
X:
-0.2642679 m
1.191045 m
Y:
-0.255 m
0.2550006 m
Z:
-0.03176806 m
3.208544 m
Auto Update from Geometry
Gravity
X:
0 m/s²
Y:
0 m/s²
Z:
-9.81 m/s²

Figure 1364:

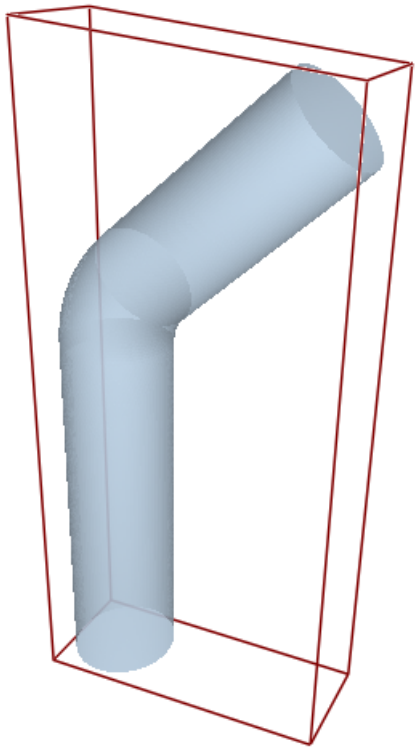


Figure 1365:

Define the Bulk Materials and Equipment Material

In this step, you will define the material models for the heavy and light bulk material and the equipment material.

1. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.
2. Rename the material to `Heavy`.
3. In the Creator Tree, set the Solids Density property to `900` kg/m^3.
   You will use the default values for other properties for this tutorial.

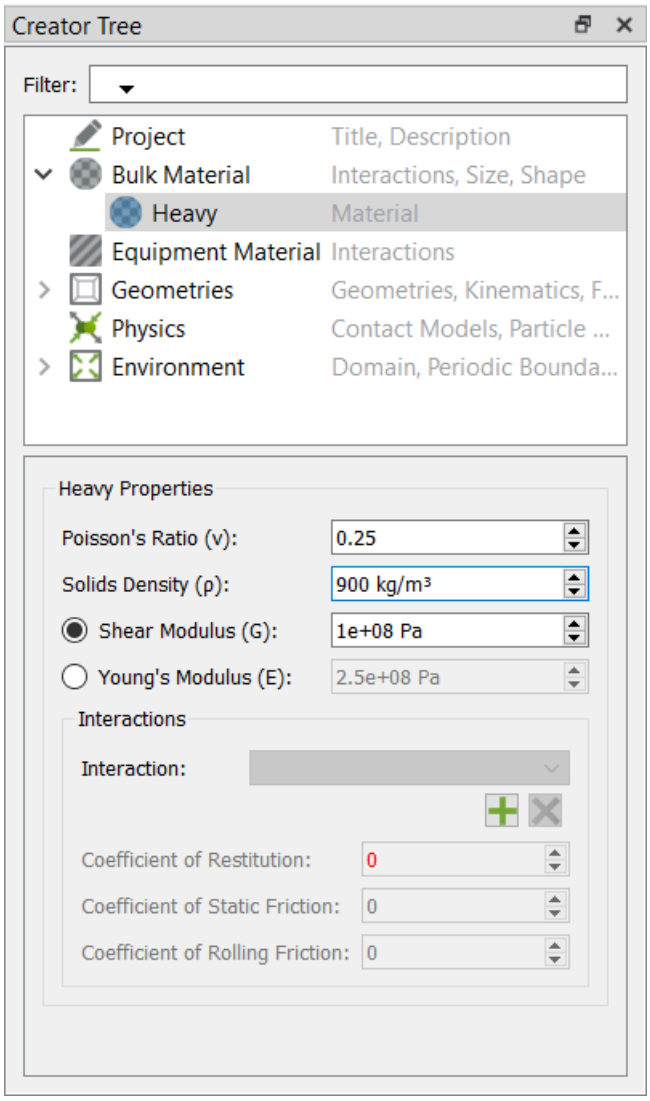


Figure 1366:

4. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, click **OK**.
5. In the Creator Tree, right click **Heavy** and select **Add Shape from Library** > **Dual Sphere Shape**.
6. Rename the particle to `Heavy particle`.
7. Under Heavy particle, click **Properties**.
8. In the Heavy particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

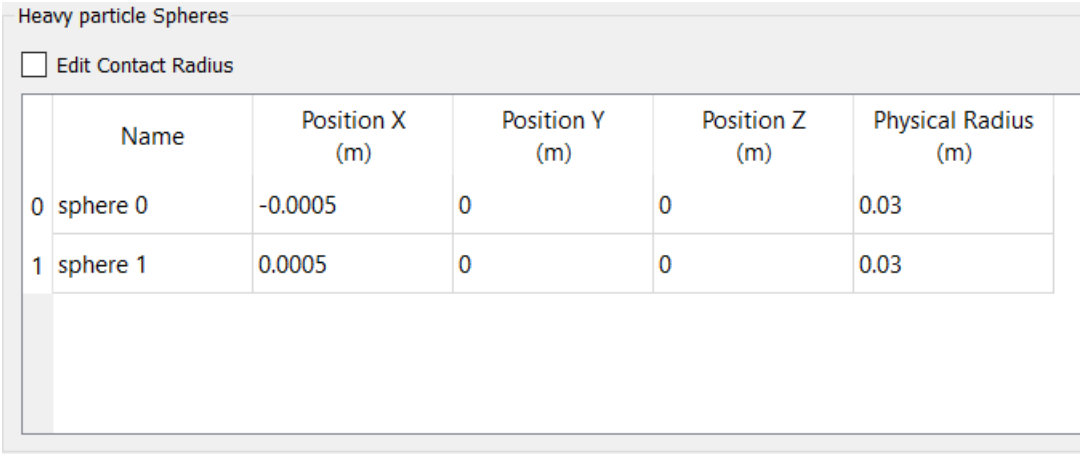


Figure 1367:

9. In the Creator Tree, click **Calculate Properties**.

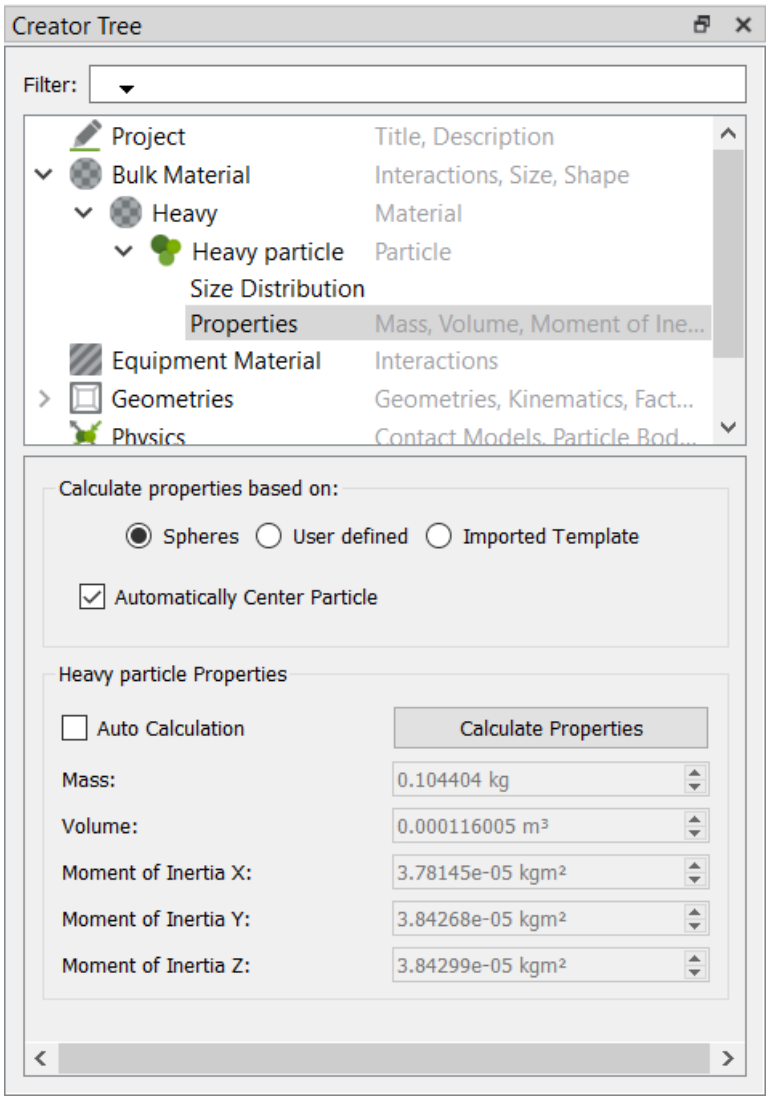


Figure 1368:

10. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.

11. Rename the material to `Light`.

12. In the Creator Tree, set the Solids Density property to `100` kg/m^3.

You will use the default values for other properties for this tutorial.

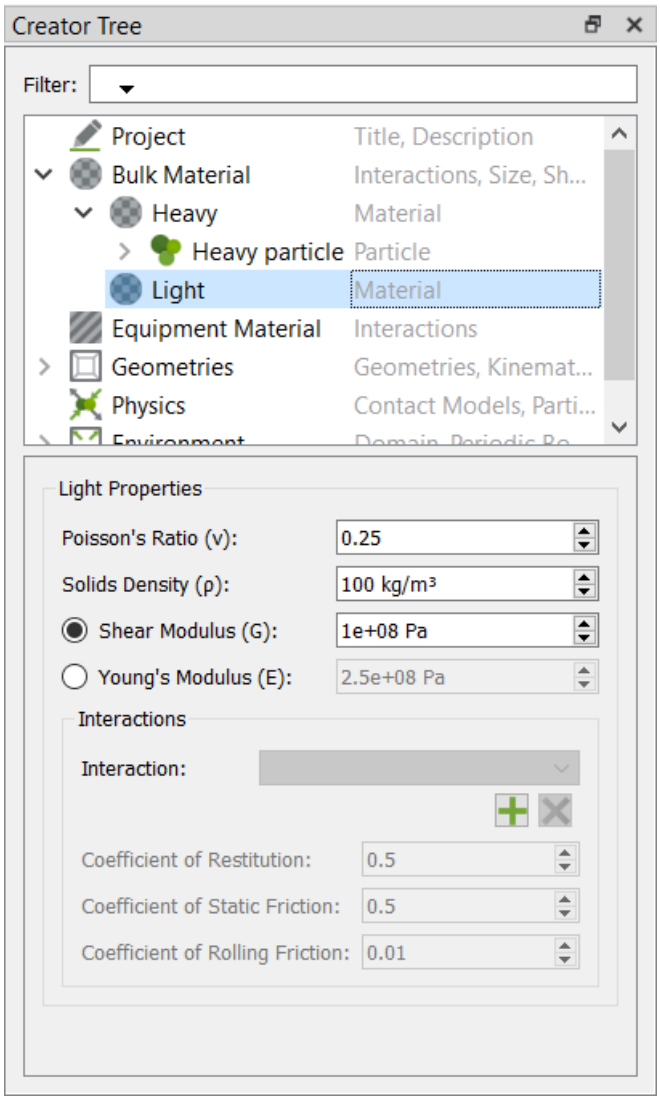


Figure 1369:

13. Click + below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.
14. Click + again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.
15. In the Creator Tree, right click on **Light** and select **Add Shape from Library** > **Dual Sphere Shape**.
16. Rename the particle to `Light particle`.
17. Under Light particle, click **Properties**.
18. In the Light particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

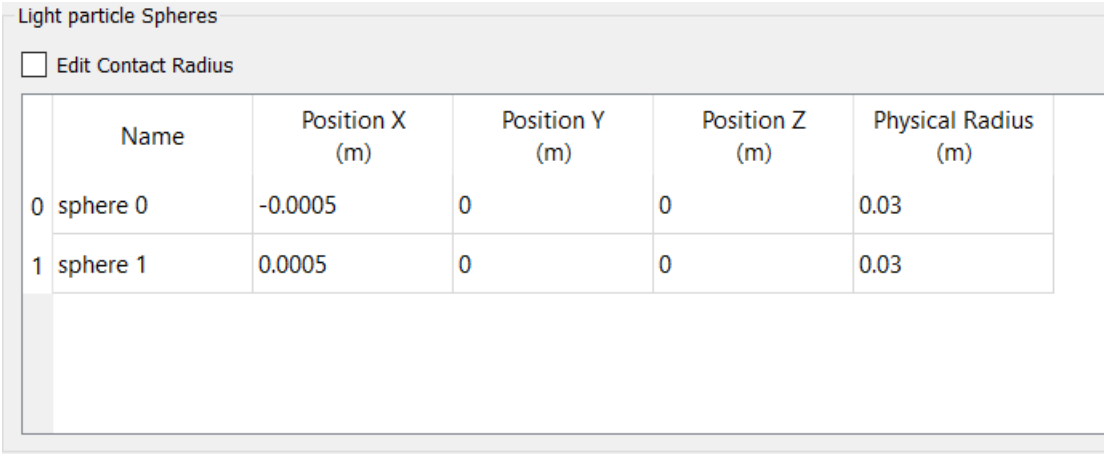


Figure 1370:

19. In the Creator Tree, click **Calculate Properties**.

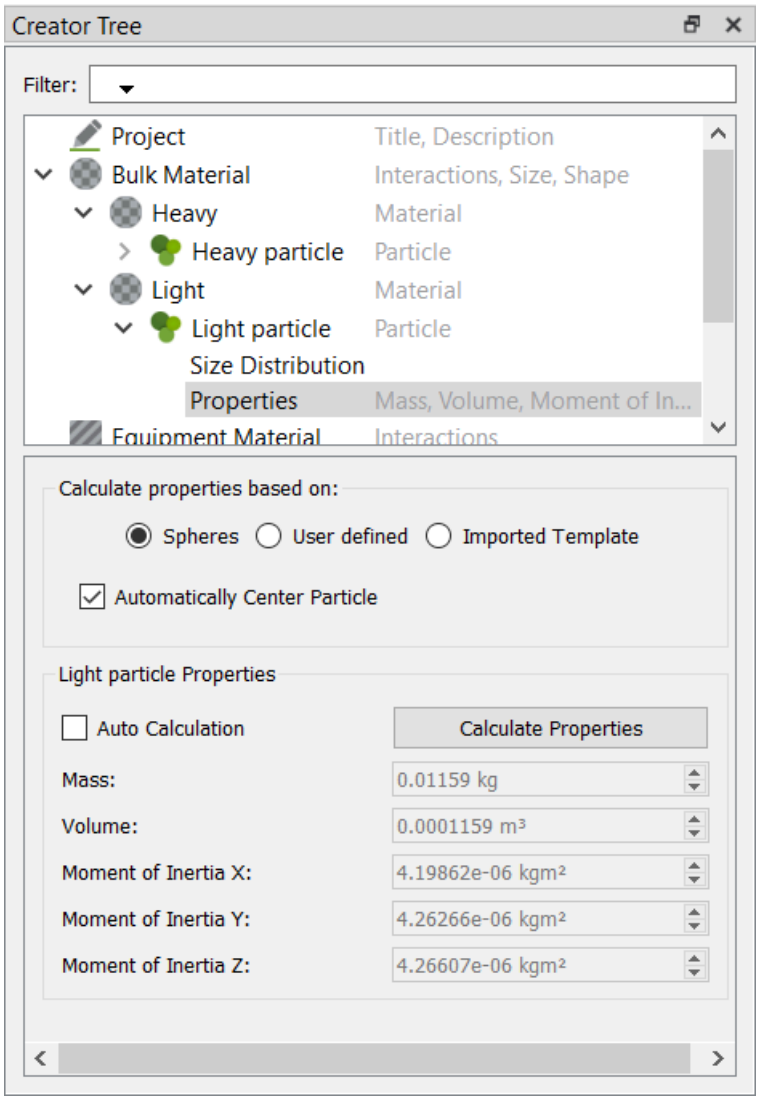


Figure 1371:

20. In the Creator tree, right-click **Equipment Material** and select **Add Equipment Material**. Rename it to `Steel`.

21. Set the Density to `7800` kg/m^3.

22. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.

23. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.

24. Save the model.

Create the Particle Factory

1. Expand **Geometries** under the Creator Tree tab. Next, right-click the **Particle inlet** surface group and select **Copy Geometry** > **Single Copy**.
2. Rename the new geometry section to `Particle_factory`.
3. Right-click the **Default Wall** geometry section in the Creator Tree and select **Merge Geometry(s)**.
4. In the **Merge Geometry** dialog, select the **Particle_inlet** and then click **OK**.
5. Click the **Default Wall** geometry section, set the Type to **Physical**, and the Material to **Steel** (if not set already).

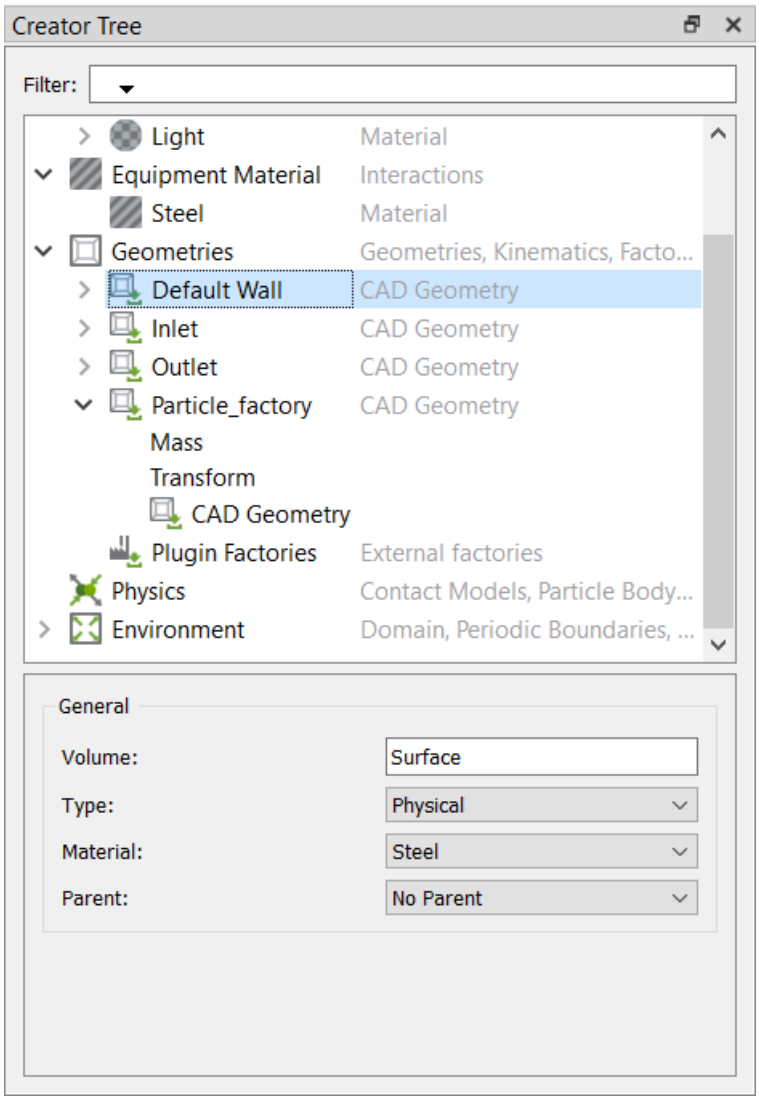


Figure 1372:

6. In the Creator Tree, click the **Inlet** section and change the Type to **Virtual**.
7. Similarly, change the Type to **Virtual** for the Outlet and Particle_factory sections.
8. Under Particle_factory, click **Transform**. Set the X-Position to `0.035` m

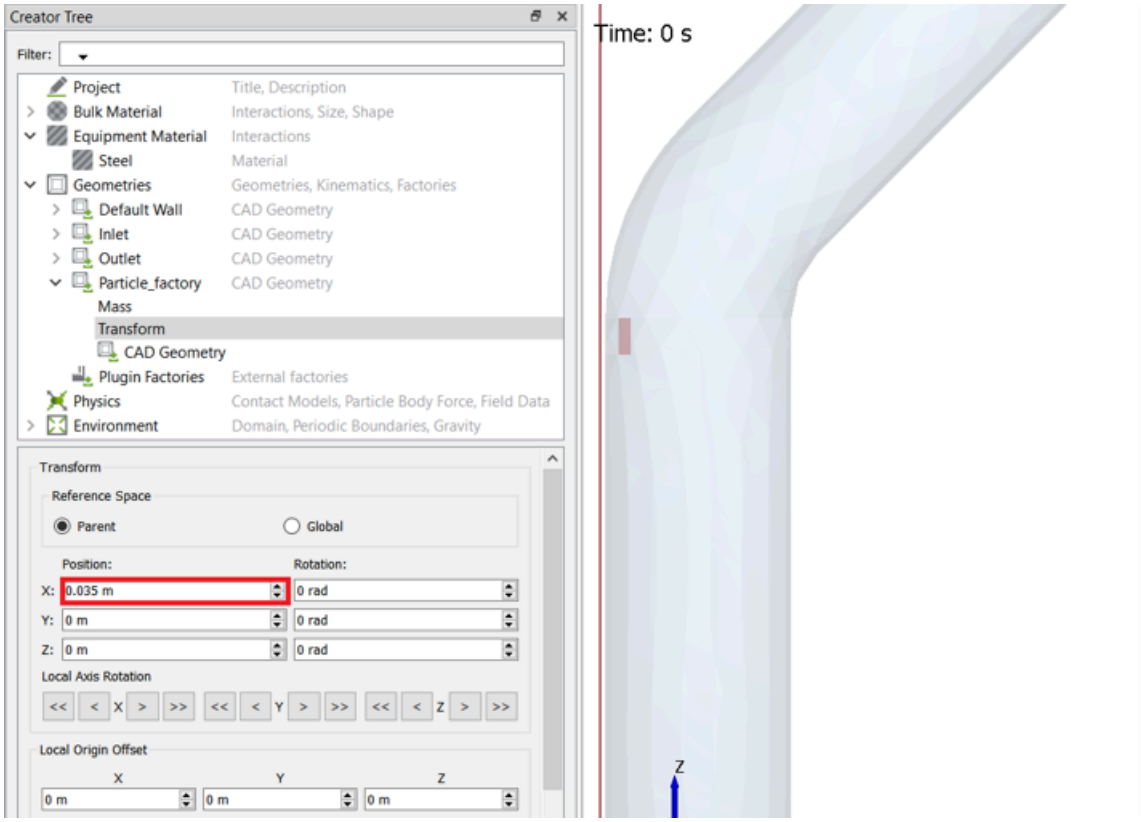


Figure 1373:

> **Note:** Set the Opacity value to 0.2 to see the transformed surface location inside the pipe geometry.
>
> This is done to make sure that the particles are generated inside the fluid domain.

Define the Particle Factory

Now that the bulk material, geometry sections, and equipment materials are defined, you need to create a particle factory to generate the particles. You will create one factory for each bulk material.

1. In the Creator Tree, right-click **Particle_factory** and select **Add Factory**.
2. Rename the new factory to `Heavy factory`.
3. Set the particle generation parameters as shown in the figure below.

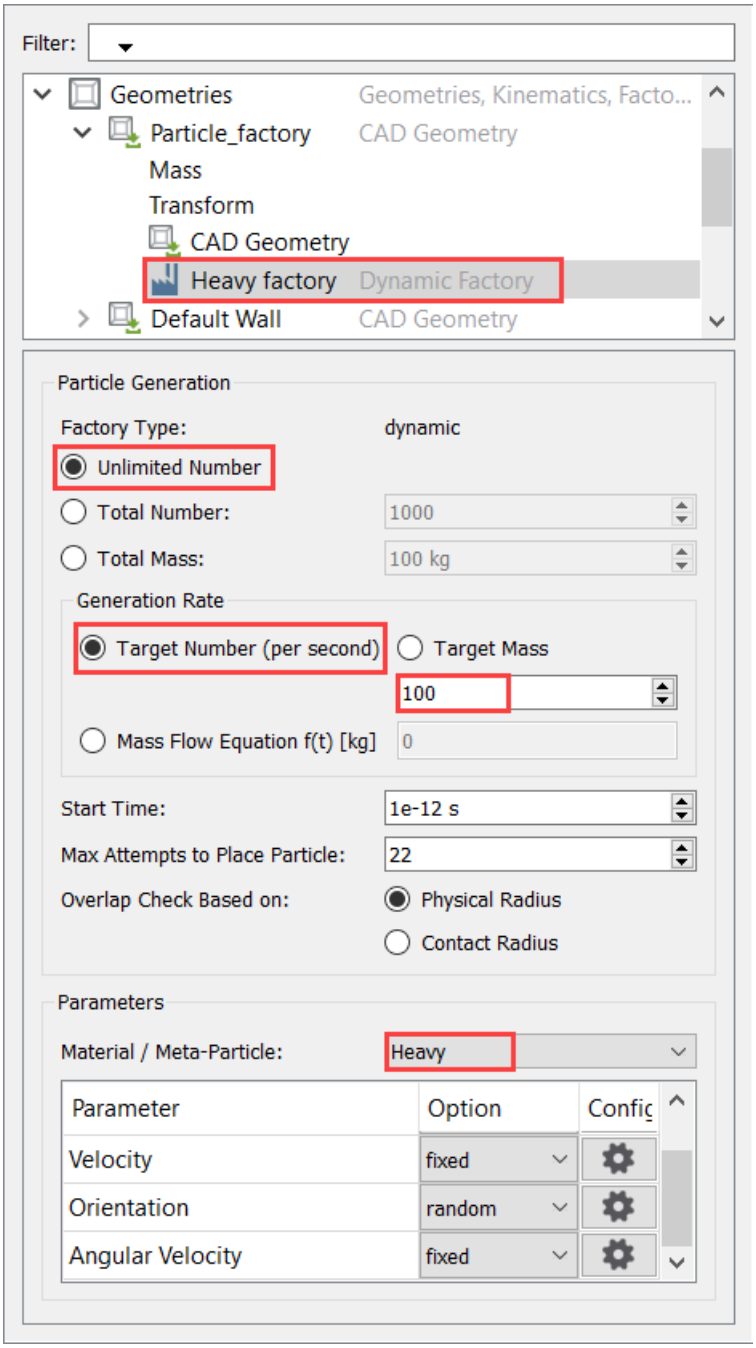


Figure 1374:

4. Click ⚙ besides Velocity, set the X-velocity to `1` m/s, and then click **OK**.
5. Repeat steps 1-4 to create another factory named `Light factory` using the same parameters but with **Light** as the Material.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).
   When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.
3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and deactivate the **Auto Time Step** checkbox.

   > **Note:** The Auto time step should never be used for AS-EDEM bidirectional coupled simulations.

3. Enter a value of `2e-5` for Fixed Time Step.

   When setting the time step for the DEM model, make sure that the CFD time step is an integer multiple of DEM time step. In this case, the CFD time step is 1e-3, which is 50 times the DEM time step. This ensures that the physical time in both the solvers match at the end of each data exchange interval. Also, the recommended range for the CFD-to-DEM time step ratio is 50-200 for most cases.
4. Set the Total Time to `1` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `4` R min.

   Generally, a value in the range of 3-6 R_{min} is recommended as the optimum cell size. The cell size in EDEM does not have any impact on the accuracy of the simulation and affects only the run time.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

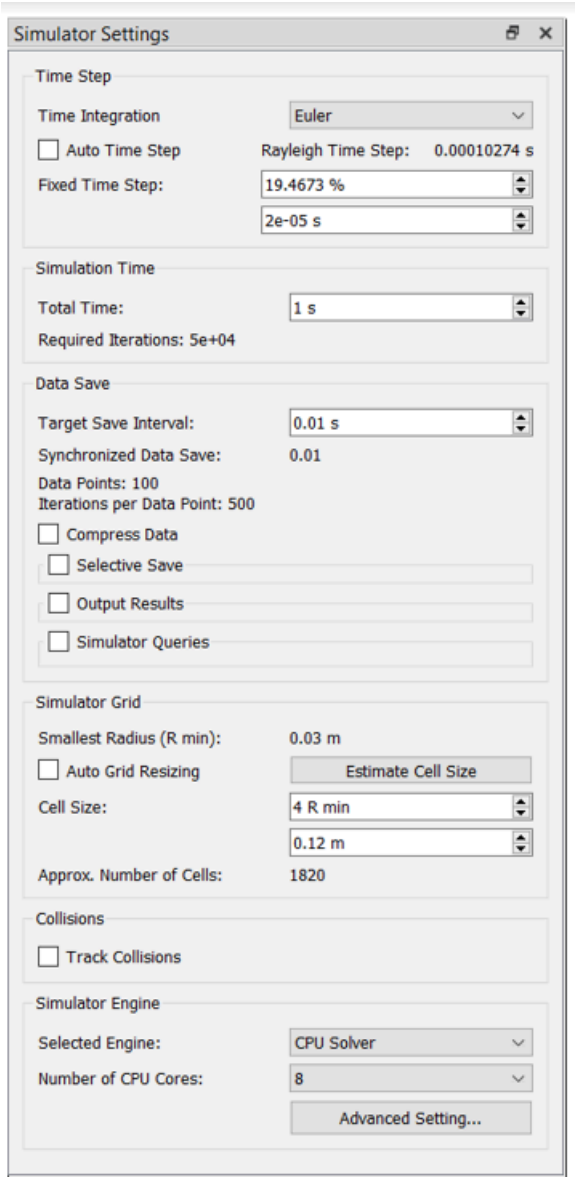


Figure 1375:

7. Once the simulation settings have been defined, save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1376:

Once the Coupling server is activated, the icon changes.

Figure 1377:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1378:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Optional: Set the number of processors to **4** or **8** based on availability.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
7. Click **Run** to launch AcuSolve.

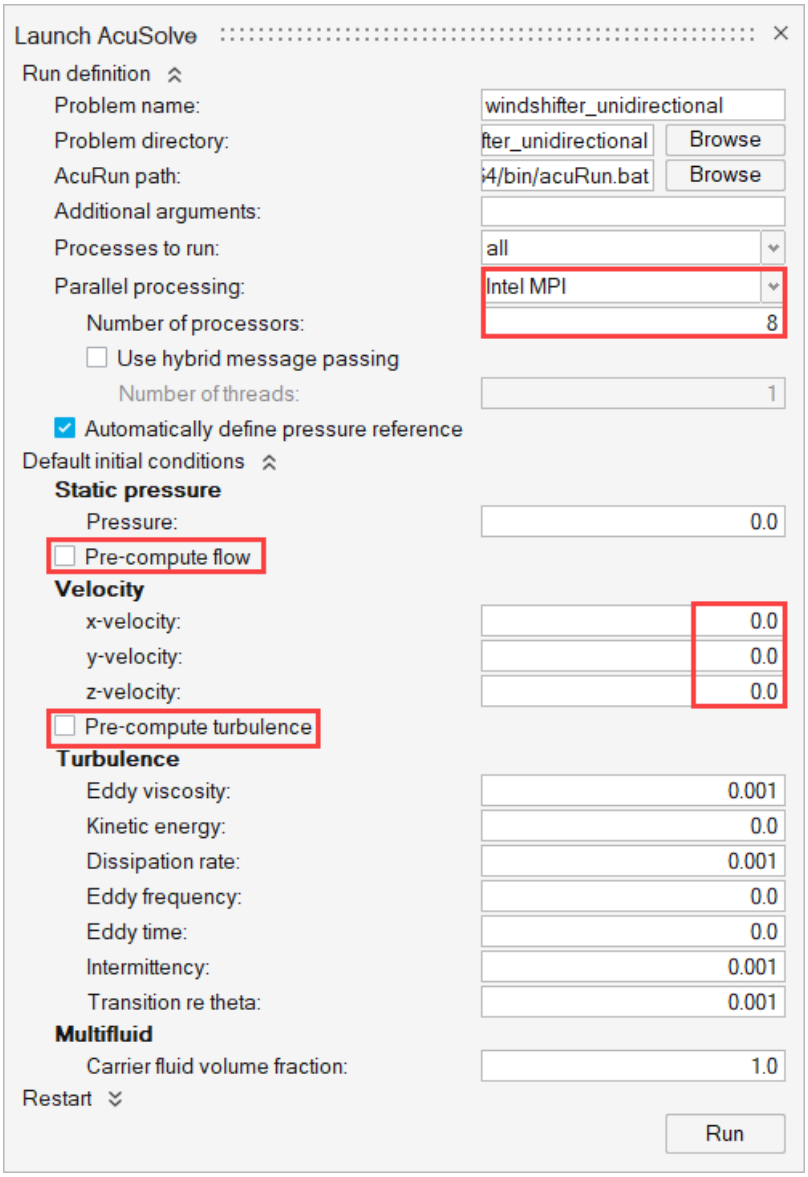


Figure 1379:

If the coupling between AcuSolve and EDEM is successful, a message will be printed in the AcuSolve Log file before the first time step.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 7.320300e+001 8.723200e+001 Sec
acuSolve: Input Memory Usage        = 6.800000e+001 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Interface thickness is 1.456277e-001
acuSolve: Init. CPU/Elapse time     = 1.078000e+000 1.710000e-001 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 1.000000e-003 ; time= 0.000000e+000
```

Figure 1380:

Analyze the Results

AcuSolve Post-Processing

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all of the surfaces are loaded in the Post Browser.
4. Select the **Left** face on the view cube to align the model to the x-z plane.

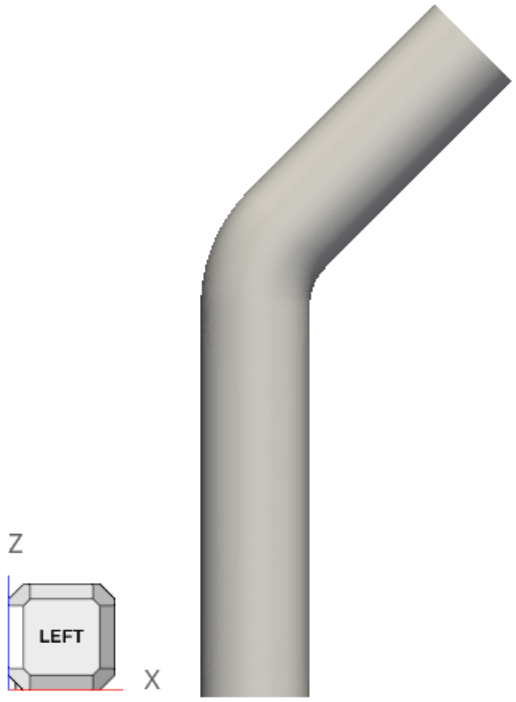


Figure 1381:

5. In the Post Browser, turn off the display of the boundary surfaces by clicking on the icon next to Flow Boundaries.

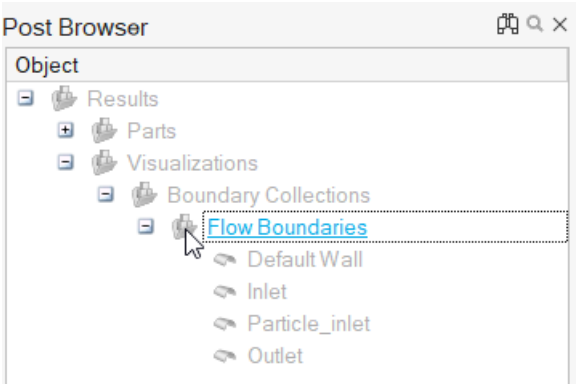


Figure 1382:

6. Click the **Slice Planes** tool.

Figure 1383:

7. In the modeling window, click on the plane that is parallel to the screen (the x-z plane).
8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
10. Change the upper bound of the legend to `36`.
11. Click and set the Colormap name to **Rainbow Uniform**.

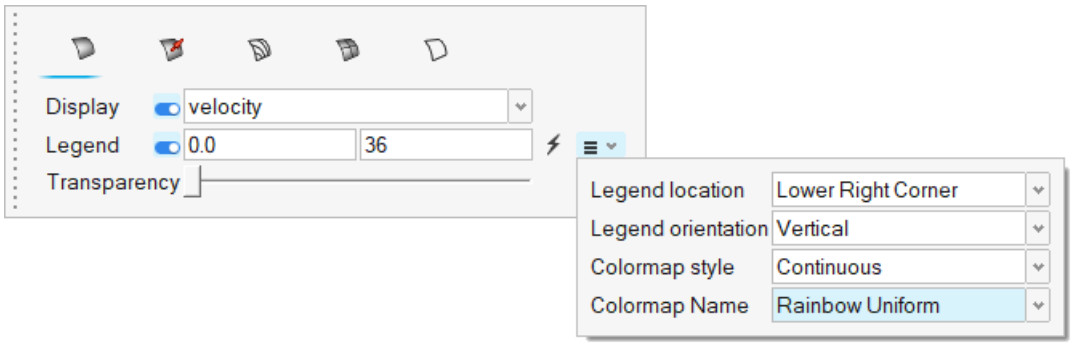


Figure 1384:

12. Click on the guide bar.
    The velocity contour should look like the one shown below at the beginning of the simulation.

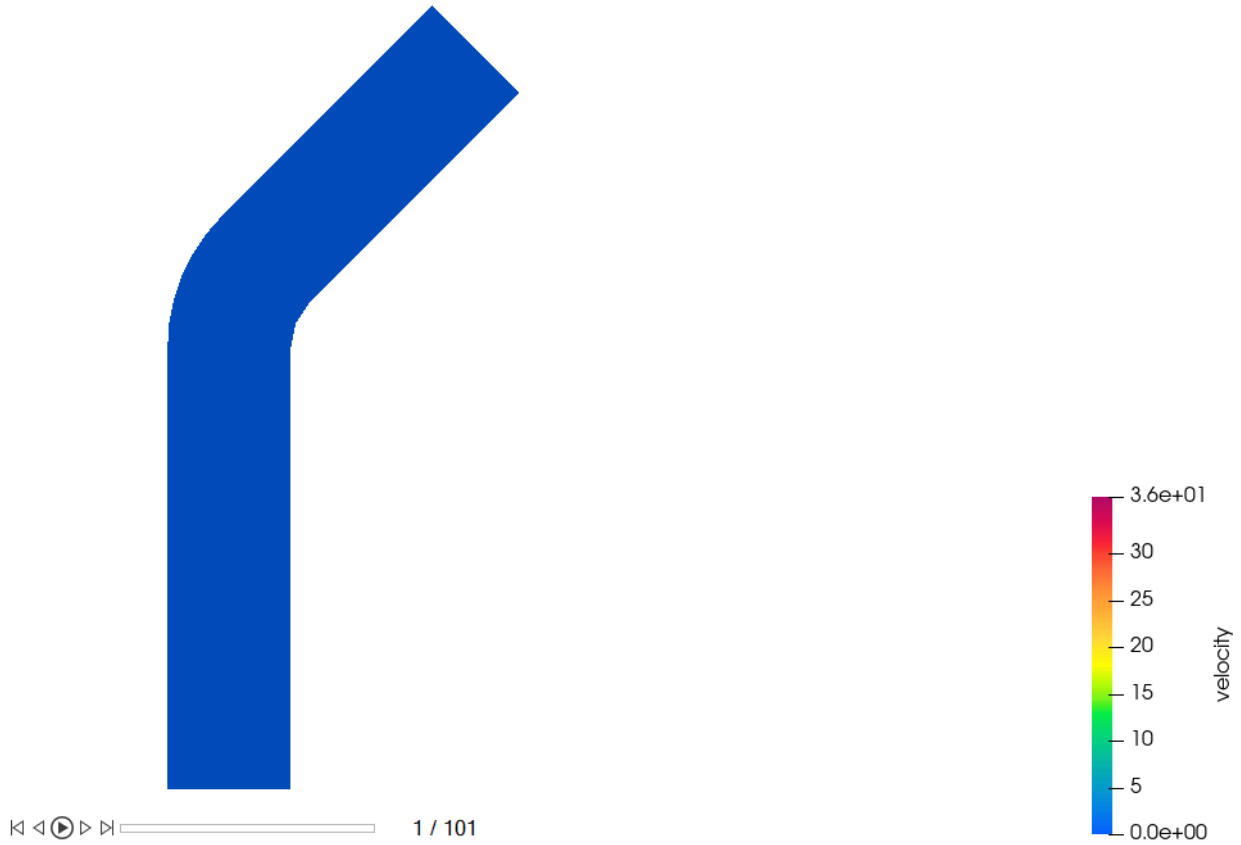


Figure 1385:

13. Plot the contours at 0.25s, 0.5s, 0.75s, and 1.0s by dragging the slider on the bottom to the 26th, 51th, 76th, and 101st frames.

Since the effect of particles on the pressure and velocity fields is not considered for unidirectional coupling, the velocity contours should look similar for all the time frames mentioned above.

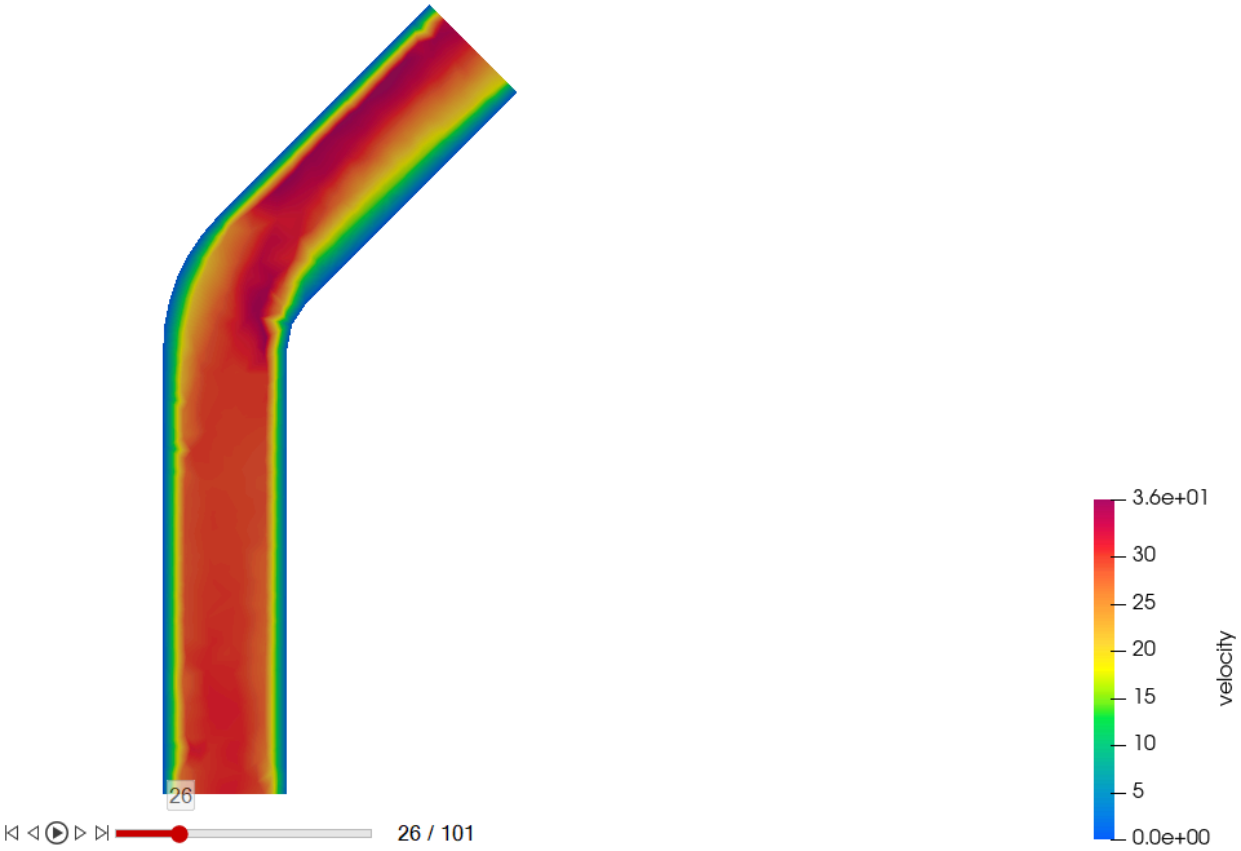


Figure 1386:

14. Right-click **Slice Plane 1** in the Post Browser and select **Edit**.

15. In the microdialog, change the Display variable from velocity to **volume fraction edem particle** and set the upper bound of the legend to `0.186`.

16. Click ✓ on the guide bar.

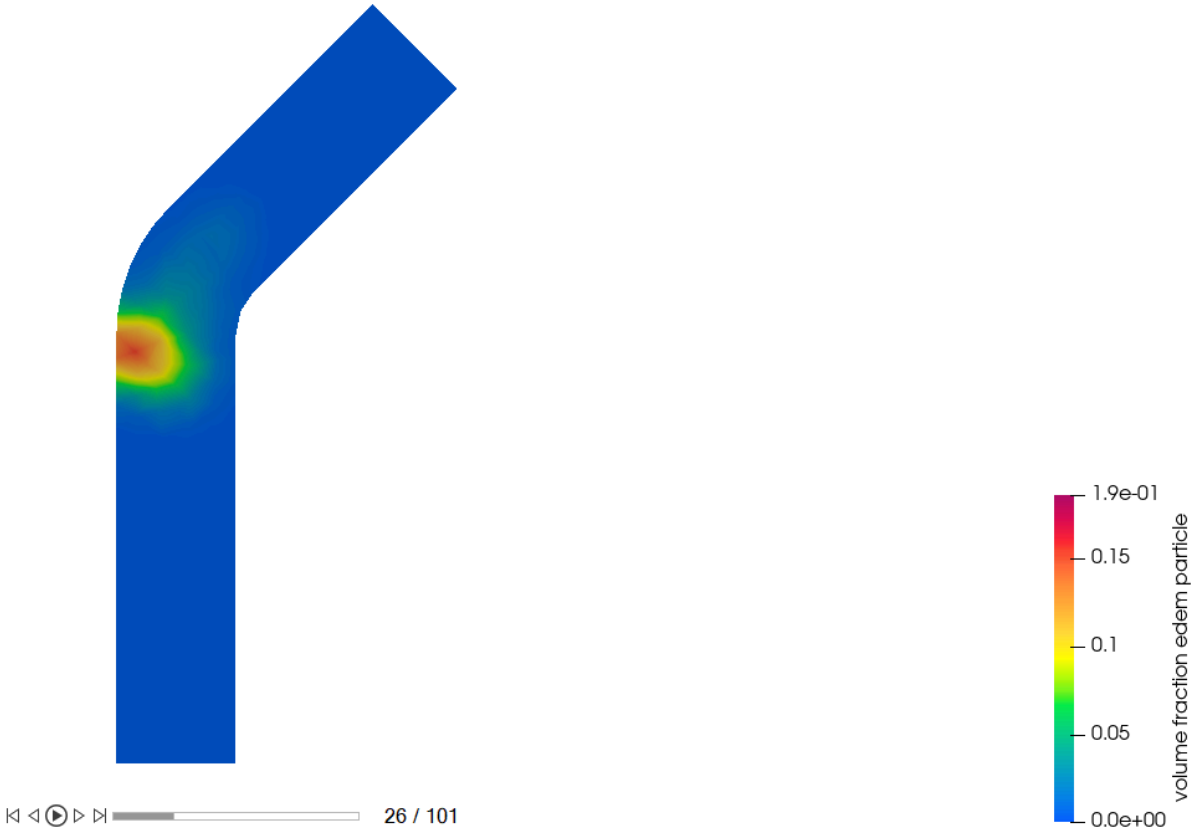


Figure 1387:

17. Plot the contours at 0.5s (step=51/101), 0.75s (step=76/101), and 1.0s (step = 101/101) to see the volume fraction edem particle distribution inside the fluid domain.

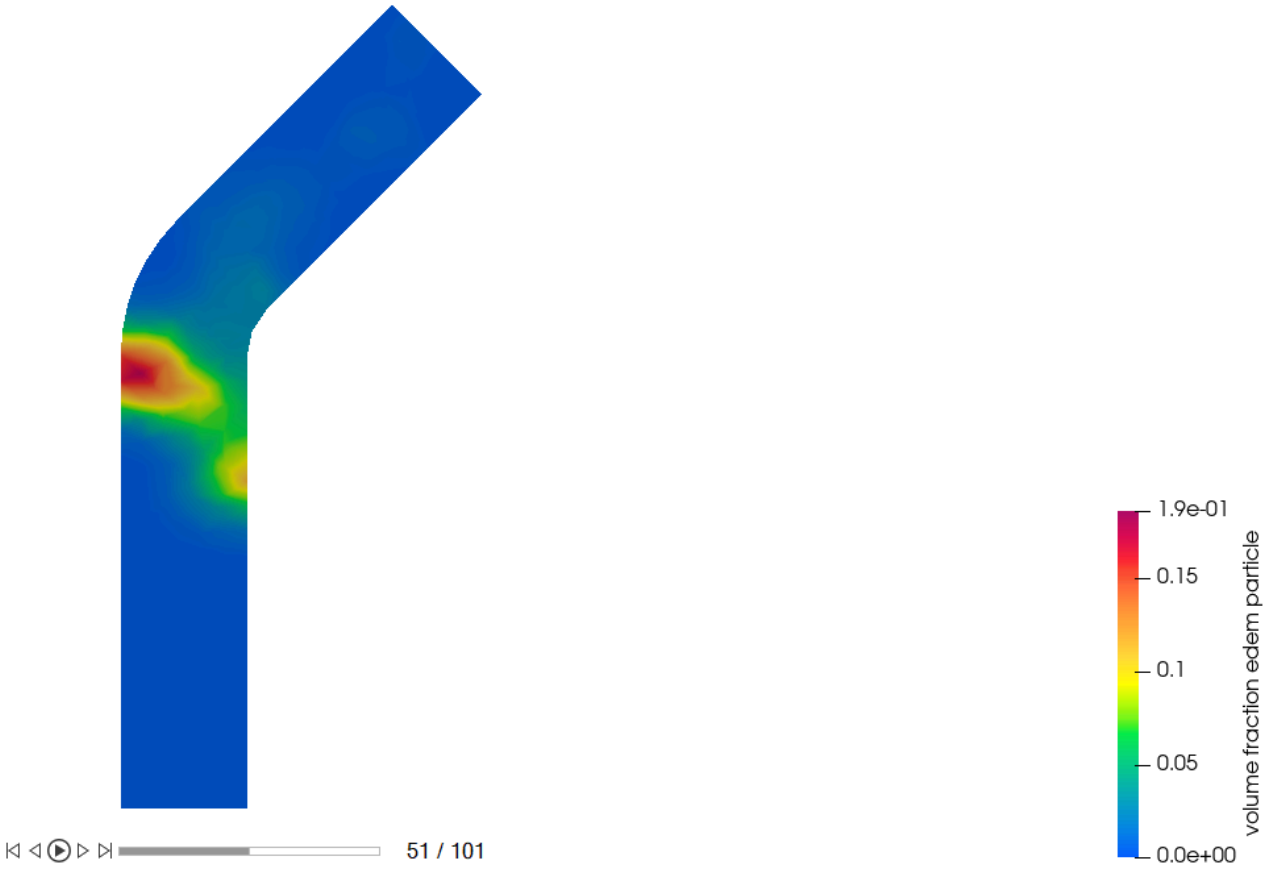


Figure 1388:

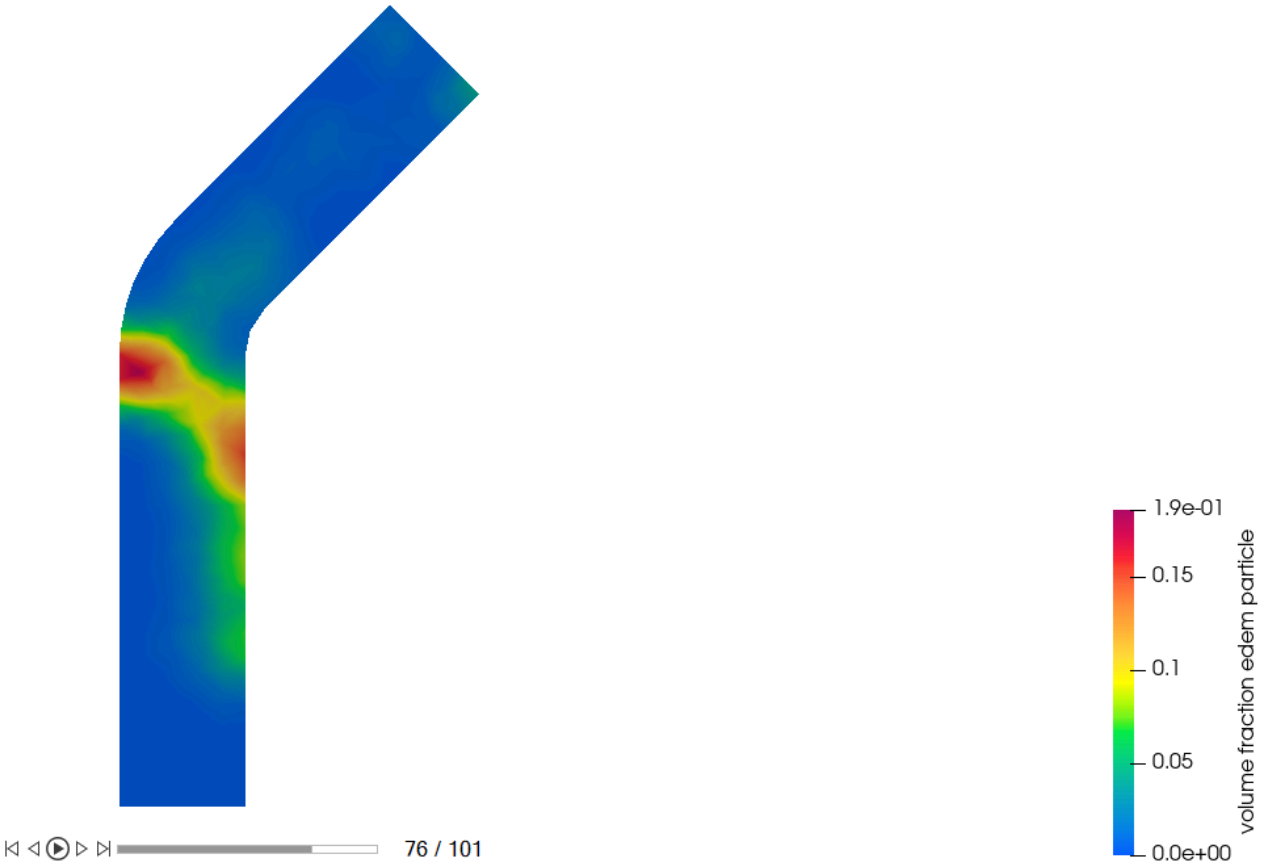


Figure 1389:

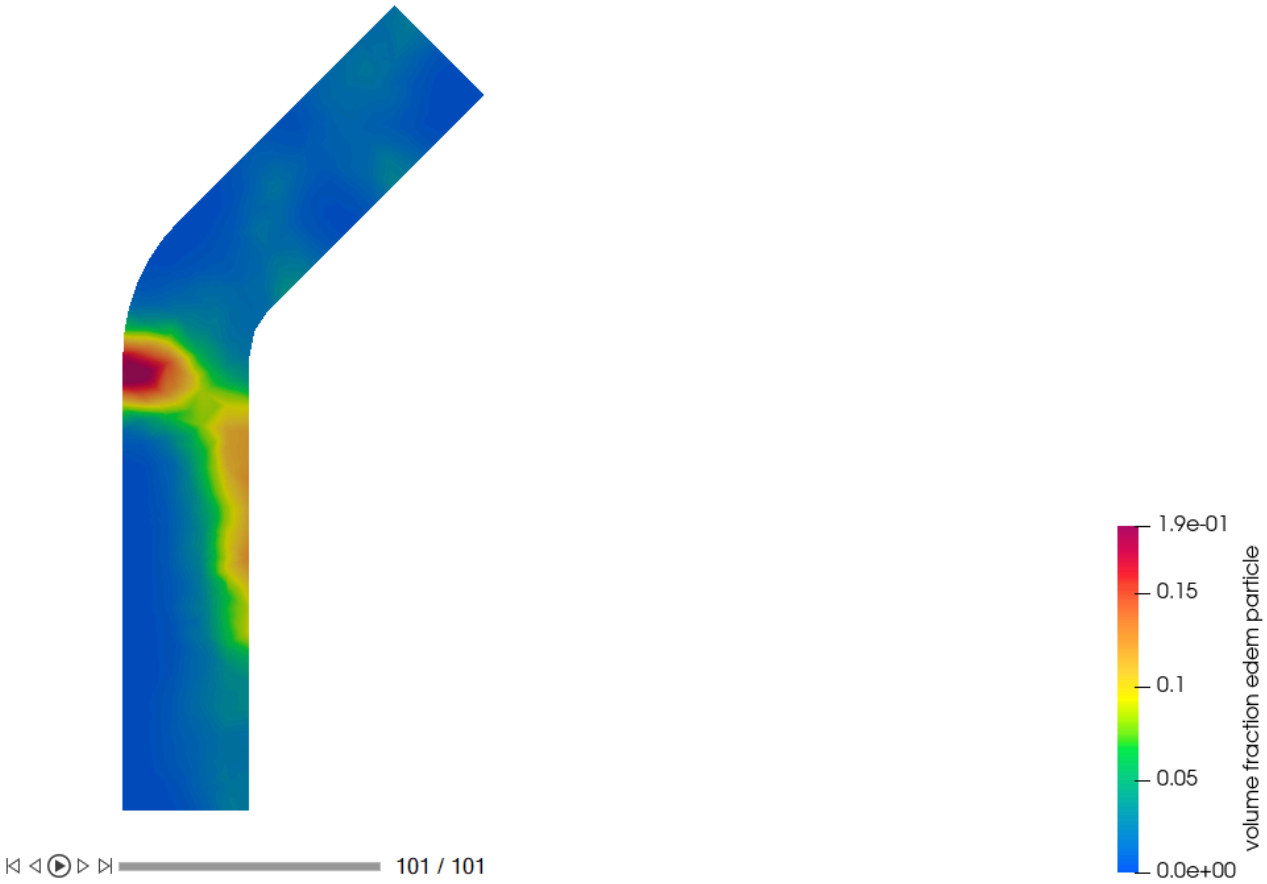


Figure 1390:

EDEM Post-Processing

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then click **Default Wall**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

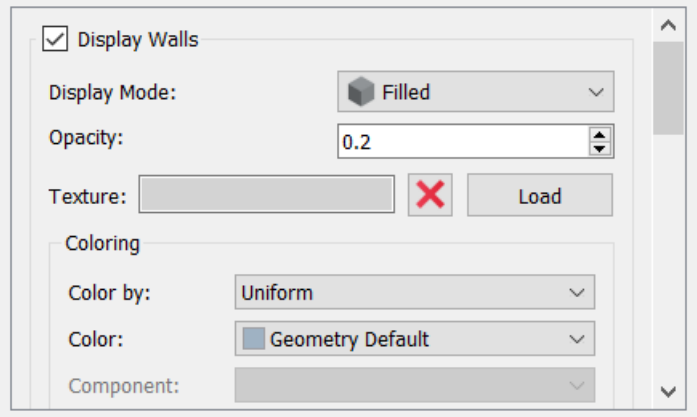


Figure 1391:

4. In the Analyst Tree, expand **Particles** and click **Heavy particle**.
5. Change the display color to **Magenta**.

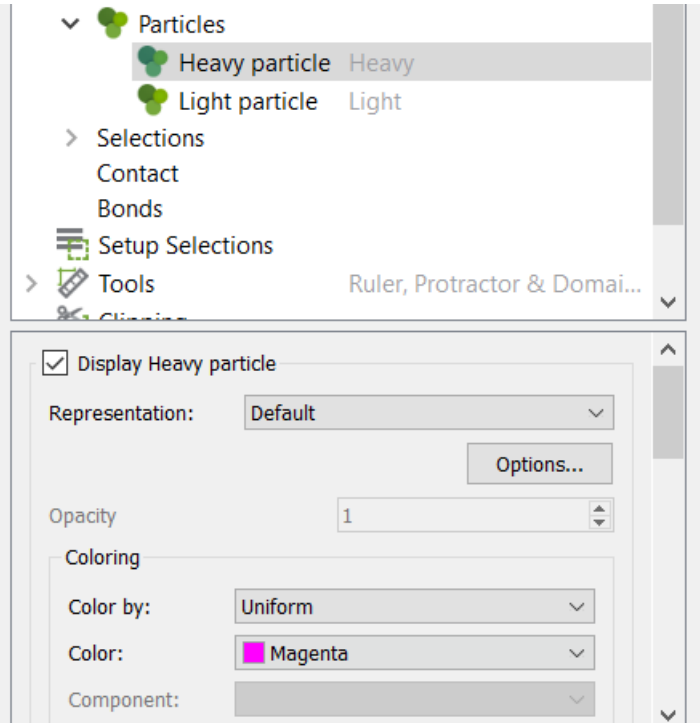


Figure 1392:

6. Click **Light particle** and set the display color to **Green**.

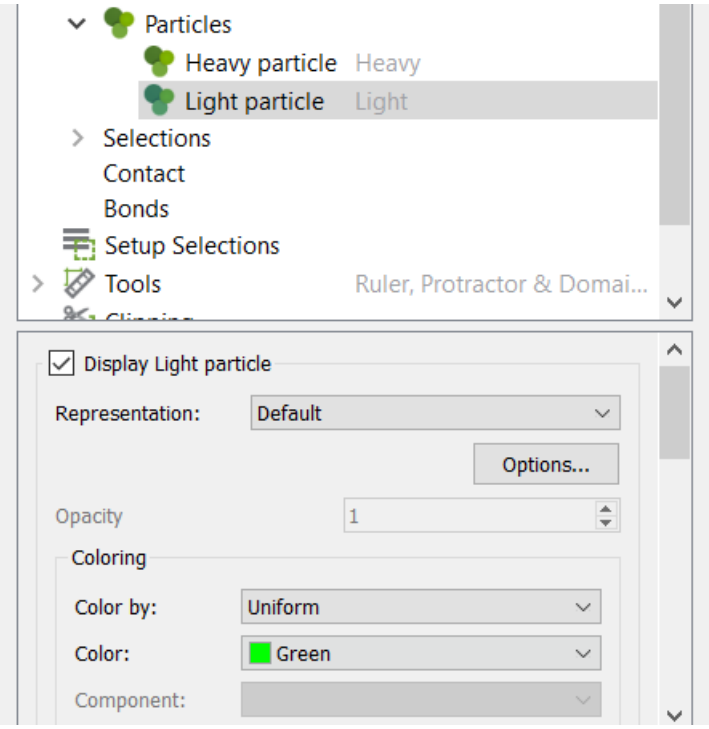


Figure 1393:

7. On the menu bar, set the time to `0` by clicking:

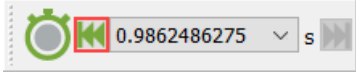


Figure 1394:

8. Set the View plane to **+ Y**.

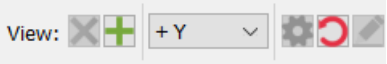


Figure 1395:

9. In the Viewer window, set the Playback Speed to `0.1x` and then click the play icon to play the particle flow animation.

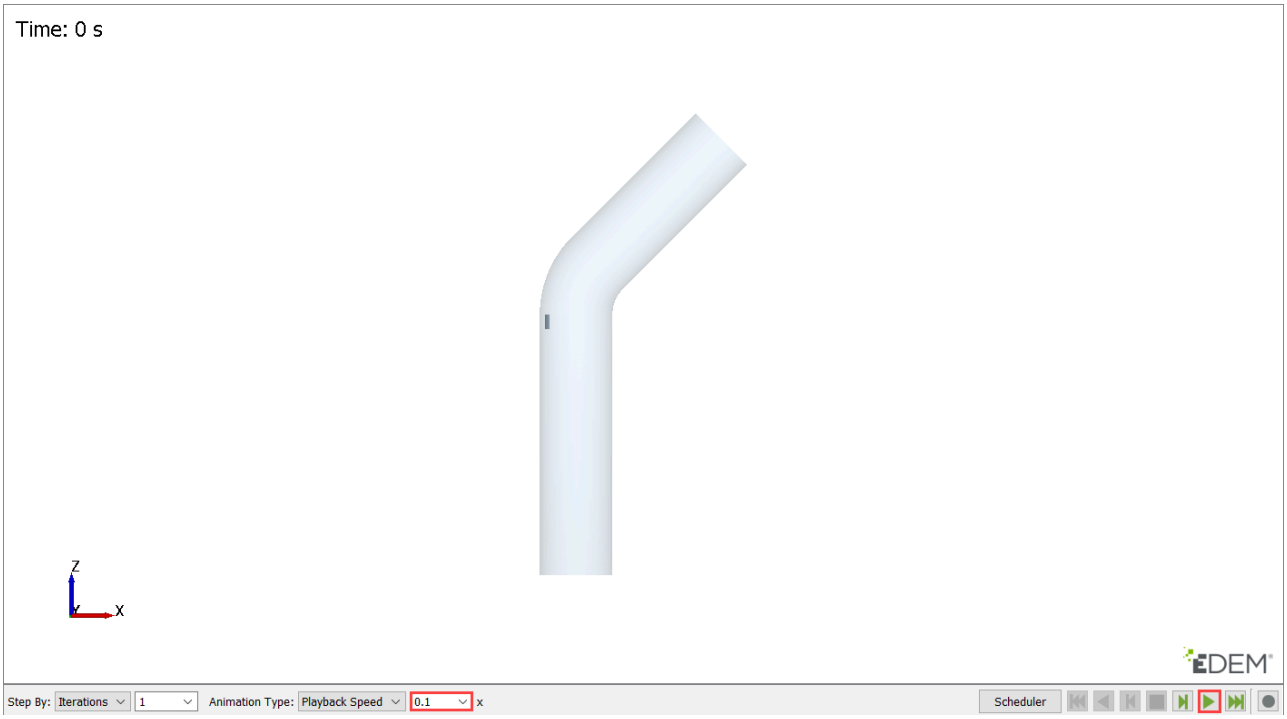


Figure 1396:

10. You can also plot the results at different timesteps by clicking the drop-down menu.

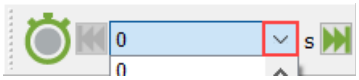


Figure 1397:

For the current case, the results are plotted at 0.25s, 0.5s, 0.75s, and 0.98s to see the particle distribution inside the fluid domain at both inlet and outlet.

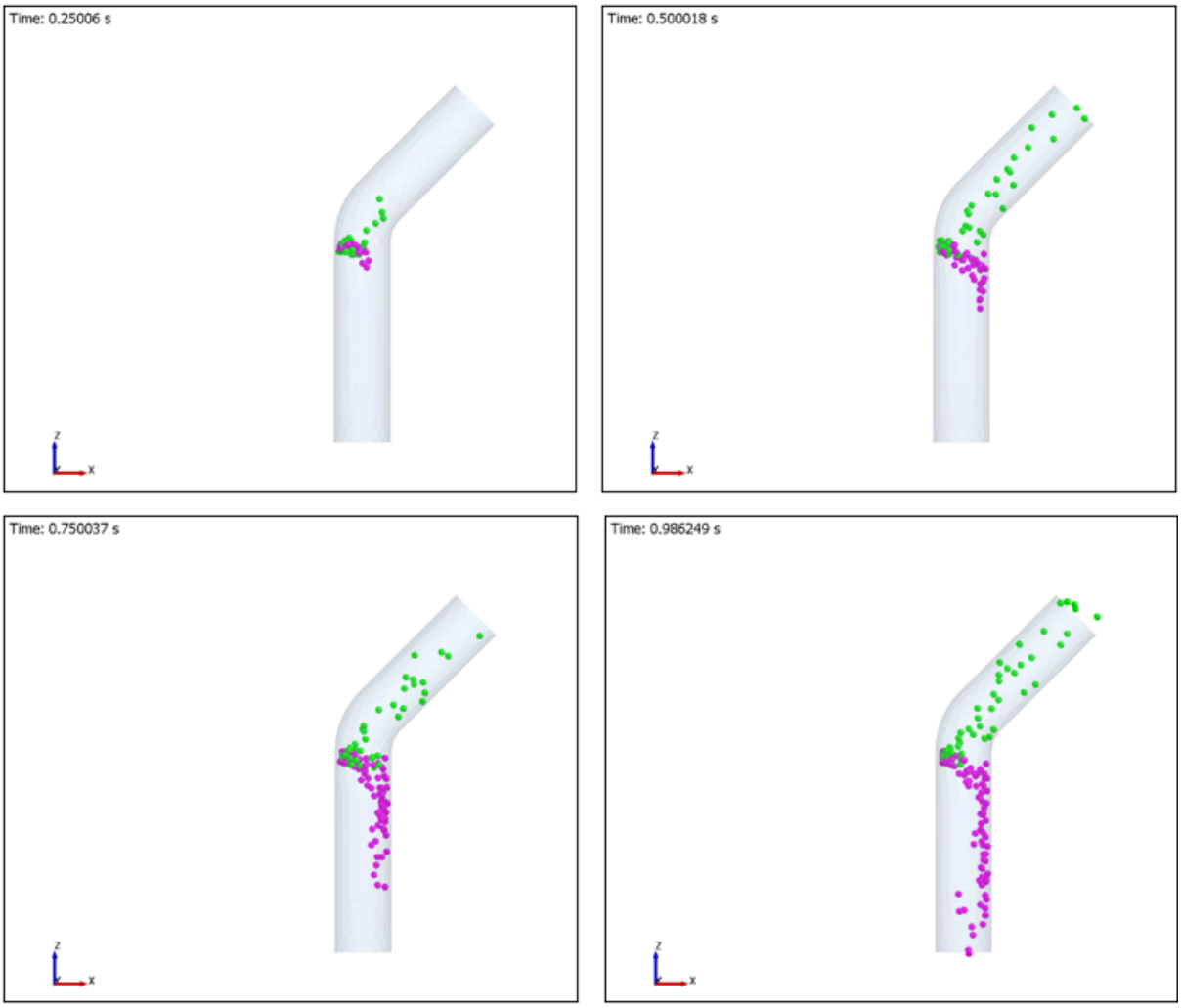


Figure 1398:

Observe that the lighter particles (green) get carried by the fluid and escape the domain through the outlet at the top and the heavier particles (magenta) stay inside the domain for a longer time while some of them fall through the bottom of the pipe.

11. On the menu bar, click the **Create Graph** icon .
12. In the Analyst Tree, change the plot type to Line by clicking Line.
13. Click on the **X-axis** tab and verify that the values are set as shown in the figure below.

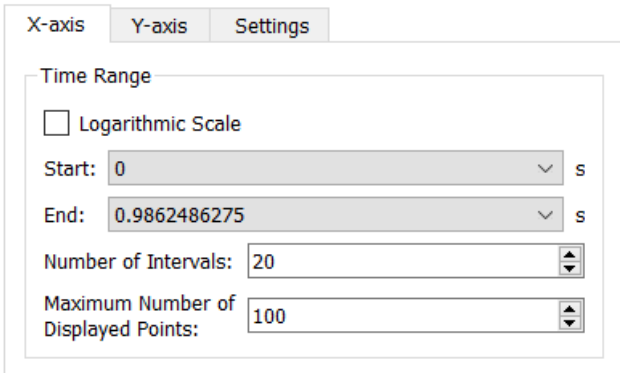


Figure 1399:

14. Click the **Y-axis** tab. Create a plot of average residence time of the heavy particle over the simulation time by setting the values as shown in the figure below.

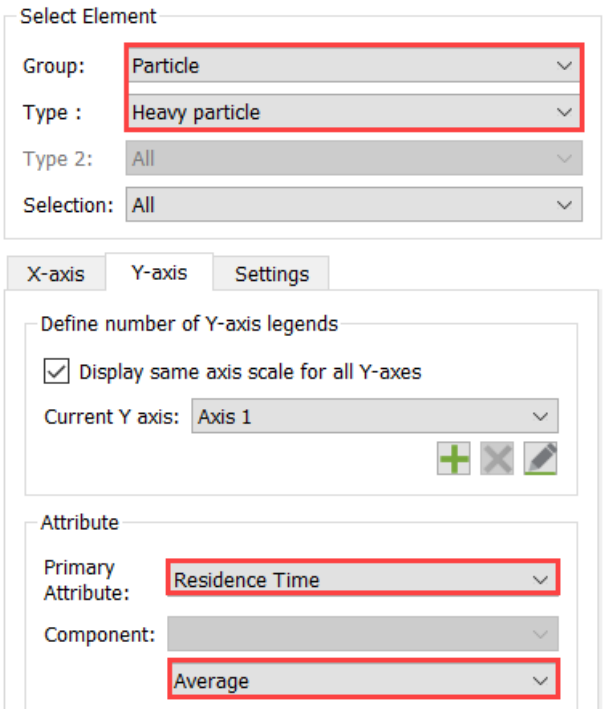


Figure 1400:

15. Click to add another Y-axis and set the Type to **Light particle**.

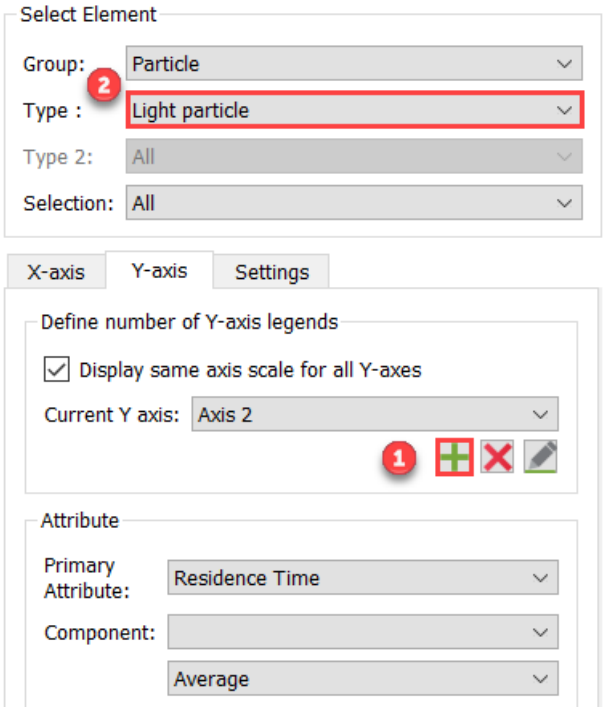


Figure 1401:

16. Leave all of the other options unchanged and then click **Create Graph** to create a plot of average residence time of both the particles.

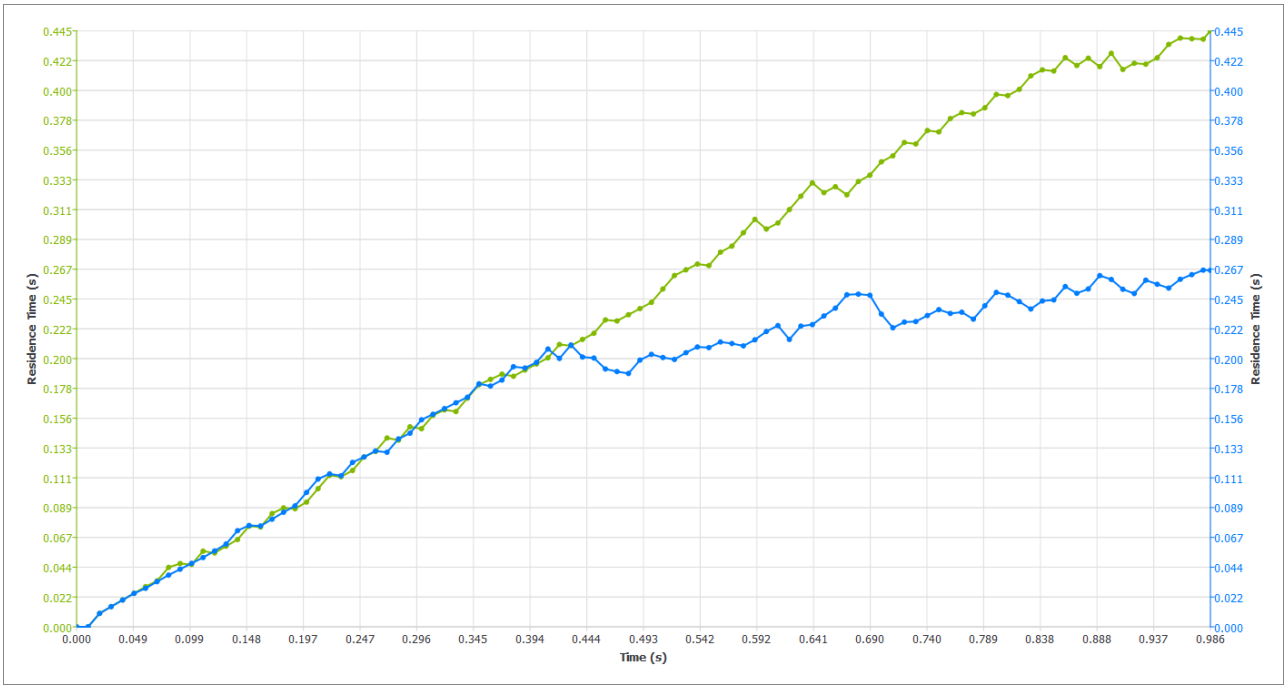


Figure 1402:

The plot in green is for the heavier particles and it can be observed that the average residence time is higher for the heavier particles compared to the lighter particles (blue).

Summary

In this tutorial, you learned how to setup and run a basic AcuSolve-EDEM unidirectional (one-way transient) coupling problem. In the first part, you set up the AcuSolve model in HyperMesh CFD and exported the geometry. Next, you imported the EDEM input files created by HyperMesh CFD, set up the EDEM model, and ran the coupled simulation. Once the coupled simulation was completed, you learned how to create animations and plots in both HyperMesh CFD Post and EDEM Analyst.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/8uoytie5r4

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6102 Particle Separation in a Windshifter using AcuSolve - EDEM Bidirectional Coupling

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up and running a basic bidirectional coupling (two-way) simulation using AcuSolve and EDEM. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6102_windshifter.hm`

Problem Description

The problem to be solved is shown schematically in Figure 1403. It is a windshifter model in which air enters the domain from the inlet at the bottom at a very high velocity (20 m/sec) and exits through the outlet. When particles are introduced into the domain through the particle inlet, the lighter particles get carried away with the air and the heavier particles exit through the opening at the bottom.

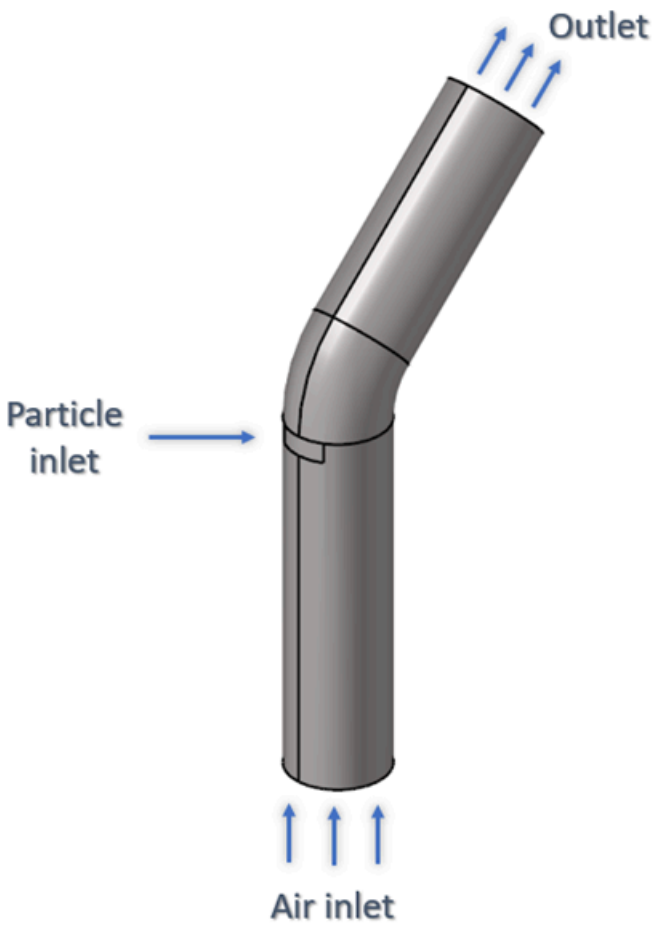


Figure 1403:

The model consists of a cylindrical pipe with a 45-degree bend. The radius of the pipe is 0.25 m and the particle inlet is located midway through the length of the pipe.

The workflow for an AcuSolve-EDEM bidirectional coupling simulation is shown below.

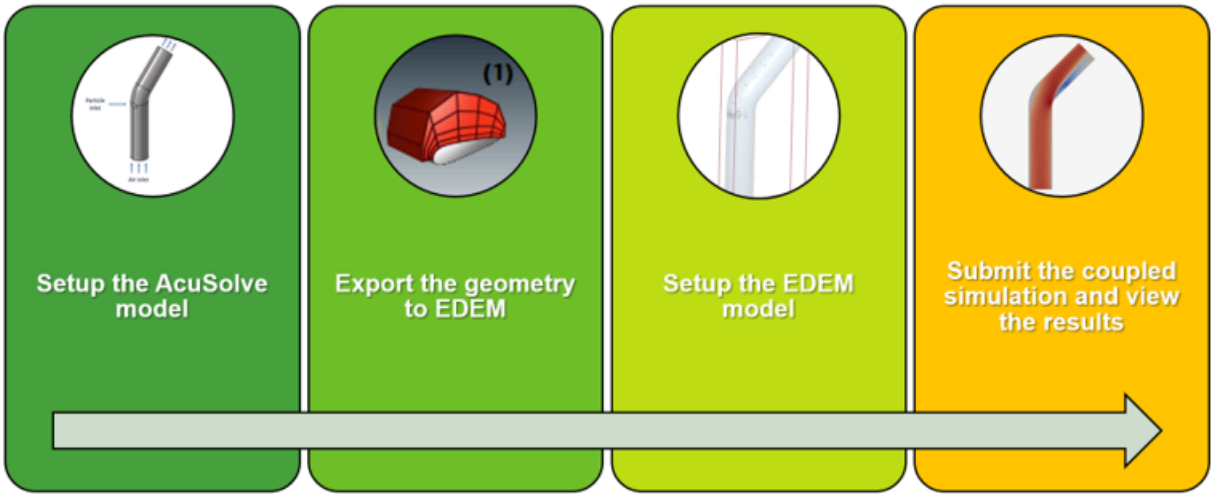


Figure 1404:

Accordingly, the tutorial consists of two parts:

1. AcuSolve setup and geometry export
2. EDEM setup and simulation

The AcuSolve model will be set up using HyperMesh CFD. Once the AcuSolve setup is complete, the EDEM deck with the geometry will be exported from HyperMesh CFD. This input deck will be opened in EDEM and will be used to complete the EDEM setup. Once the EDEM deck is set up, you will launch the coupled simulation.

Two different bulk materials used in the EDEM simulation and their properties are listed below:

Name	**Density (kg/m^3)**	**Particle radius (m)**	**Average weight of individual particle (kg)**	**Rate of generation (particles per sec)**
Heavy particle	900	0.03	0.1	100
Light particle	100	0.03	0.01	100

The fluid drag forces on the particles are calculated using the Ergun-Wen-Yu drag model. AcuSolve receives the particle position and velocity information from EDEM and calculates the drag forces on particles and updates the fluid field. This fluid forces information is sent back to EDEM, which will be used to update the location and velocities of the particles. This loop is repeated until the end of the simulation.

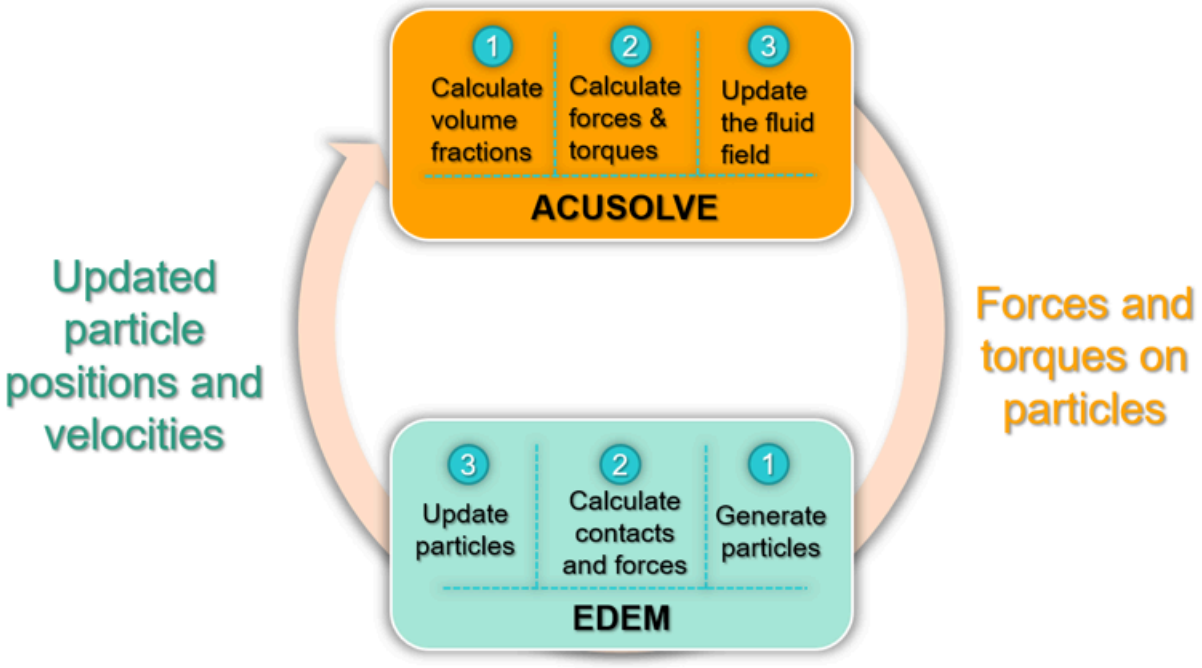


Figure 1405:

Part 1 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1406:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6102_windshifter.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Save the database as `windshifter_bidirectional` in the same directory as the other input files.

   This will be the working directory and all the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1407:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

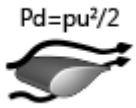
   

   Figure 1408:

   The **Setup** dialog opens.
2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the Material Library dialog, select **EDEM 2 Way Multiphase**, switch to the **My Material** tab, then click + to add a new material model.
6. In the microdialog, click **EDEM Bidirectional Material** in the top-left corner and change the name to `Air-Particle`.
7. Set the Carrier field to **Air** (if not set already).
8. Set the drag model and the drag coefficients as shown in the figure below.

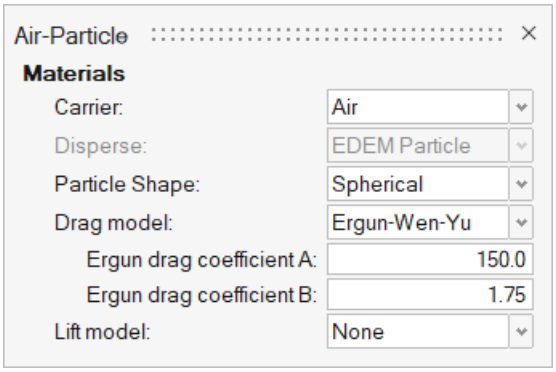


Figure 1409:

9. Close the material model microdialog and then close the Material Library dialog.
10. In the Setup dialog, set the Eulerian Material to **Air-Particle**.
11. Set Time step size and Final time to `0.001` and `1`, respectively. Select **Spalart-Allmaras** for the Turbulence model. Verify that Gravity is set to `0, 0, -9.81`, as shown below.

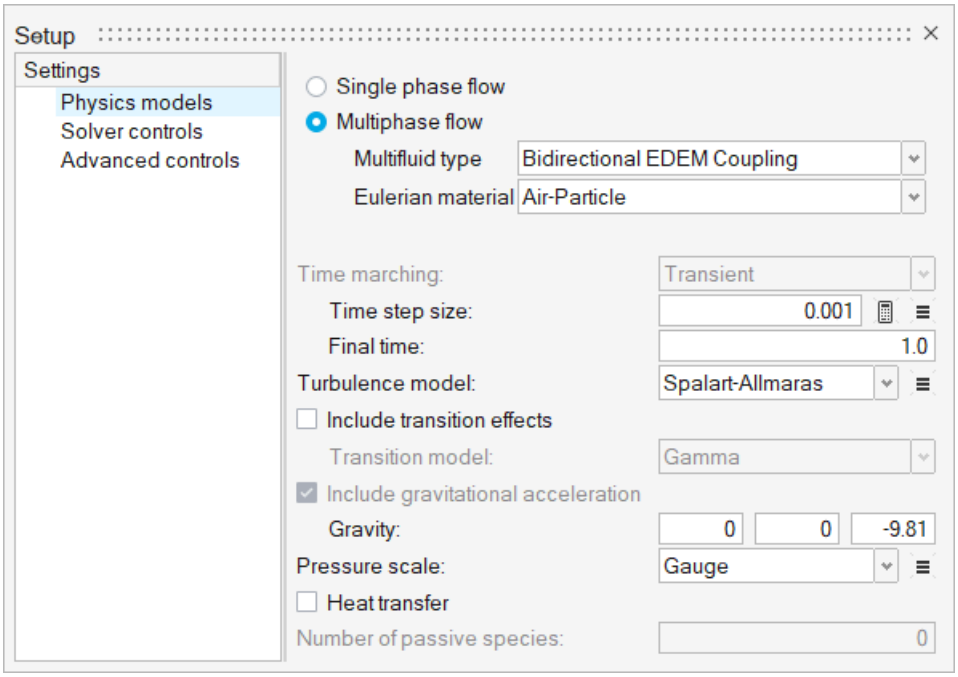


Figure 1410:

12. Click the **Solver controls** setting and set the Minimum and Maximum stagger iterations to `2` and `4`, respectively.

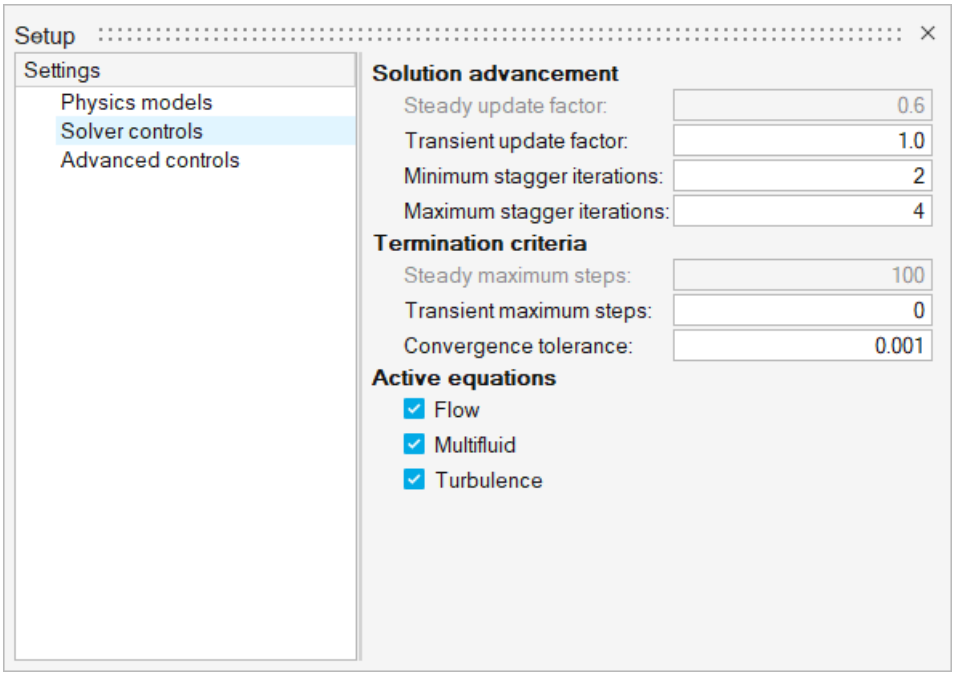


Figure 1411:

13. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1412:

2. Verify that Air-Particle has been assigned as the material.
3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1413:

2. Click on the inlet face highlighted in the figure below. In the microdialog, enter a value of `20` m/s for the Average velocity and set the Carrier fluid volume fraction to `1.0`.

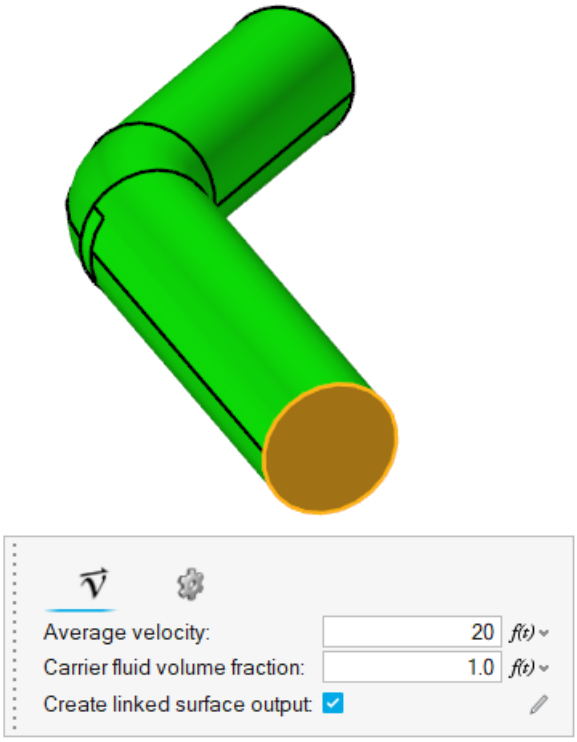


Figure 1414:

3. On the guide bar, click ✓ to execute the command and exit the tool.
4. Click the **Outlet** tool.

Figure 1415:

5. Select the face highlighted in the figure below and then click ✓ on the guide bar.

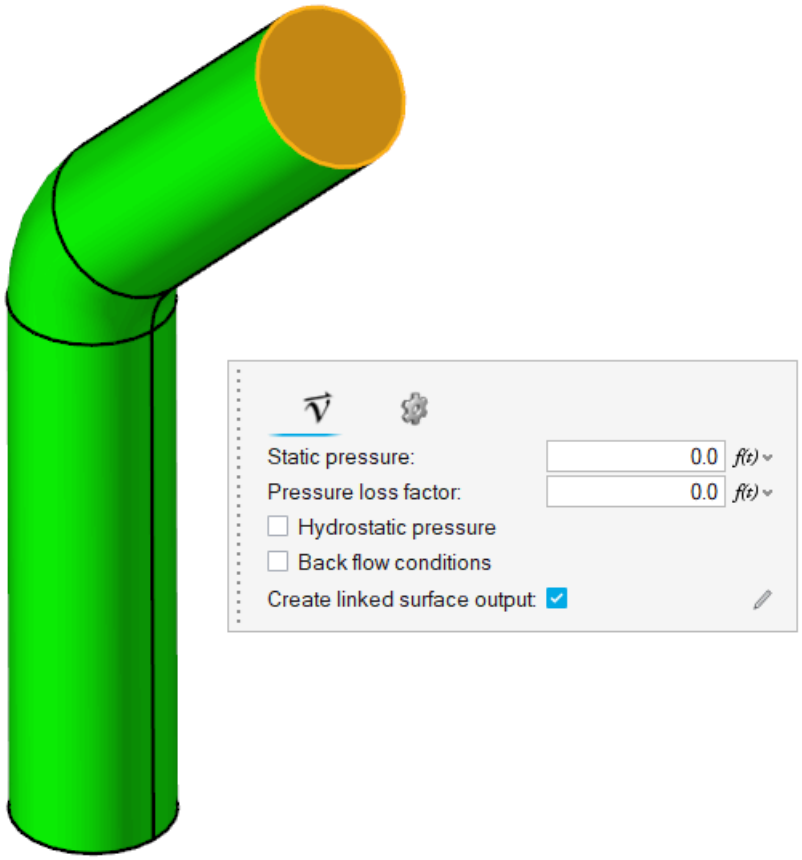


Figure 1416:

6. Click the **No Slip** tool.

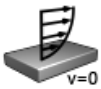


Figure 1417:

7. Select the face highlighted in the figure below.

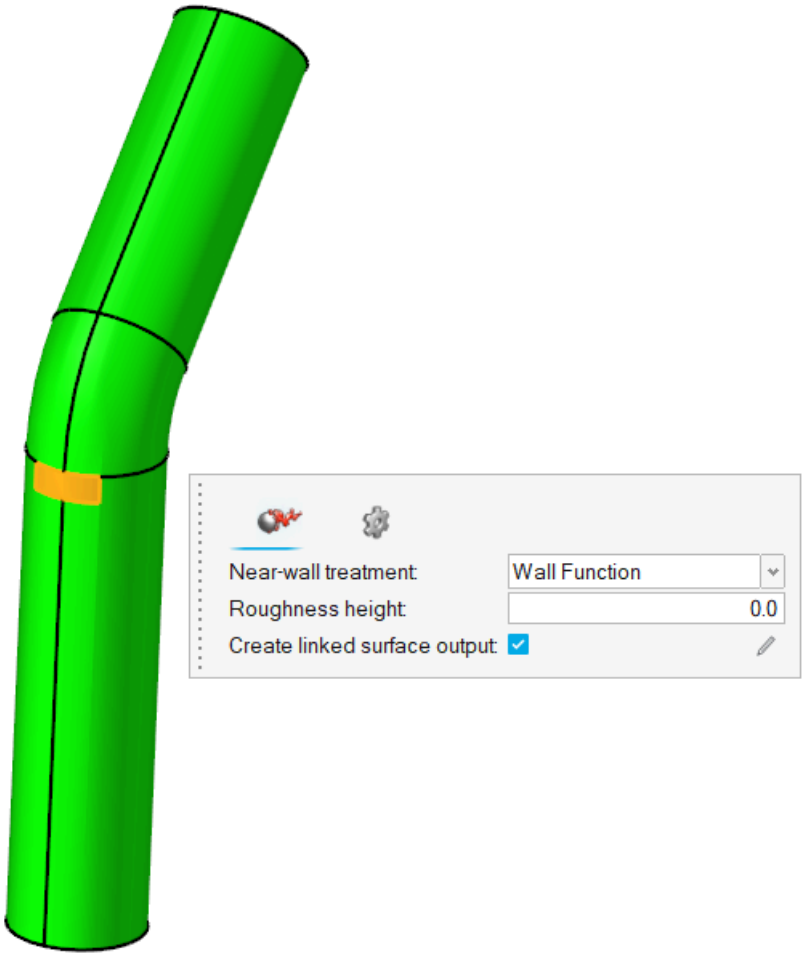


Figure 1418:

8. In the Boundaries legend, double-click **Wall** and rename it to `Particle_inlet`.
   You will use this surface as a reference geometry to create a particle factory in EDEM. Hence, you are placing this surface in a different geometry group.
9. Click ✓ on the guide bar.
10. Save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

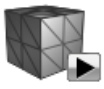


Figure 1419:

The **Meshing Operations** dialog opens.

2. Change the Average element size to `0.3`.

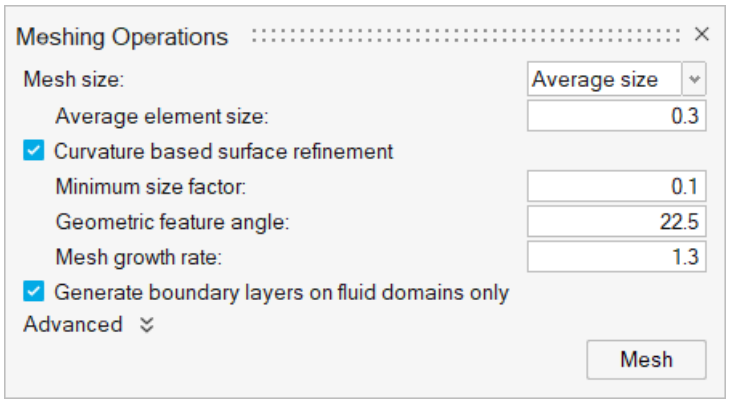


Figure 1420:

3. Click **Mesh**.
The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

> **Tip:** Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

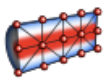


Figure 1421:

The **Field Output** dialog opens.

2. Check the box for **Write Initial Conditions**.
3. Set the Time step interval to `10`.

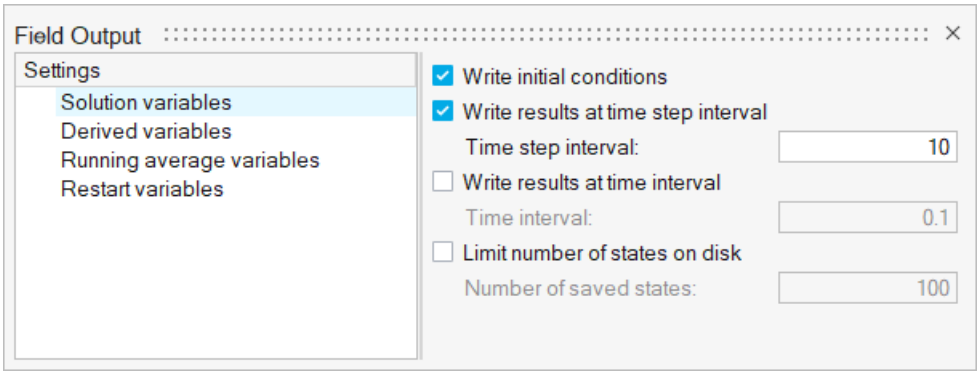


Figure 1422:

Export the Solver Deck

1. From the menu bar, go to **File** > **Export** > **Solver Deck**.
2. Name the file `windshifter_bidirectional` and make sure that **AcuSolve (*.inp)** is selected as the file type.
3. Click **Save**.

The AcuSolve input files and the EDEM input deck with the geometry sections are created. You will use this EDEM deck to set up the DEM simulation deck.

Part 2 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

As mentioned earlier, when the AcuSolve simulation was launched, HyperMesh CFD created a set of EDEM files in the problem directory. You will open that EDEM input deck and setup the DEM simulation.

1. In the Creator tab in EDEM, go to **File** > **Open**.
2. In the dialog, browse to the AcuSolve problem directory and open the `windshifter_bidirectional.dem` file located in the EDEM folder.
   The geometry is loaded.
3. Click the **Environment** tab under the Creator Tree, uncheck **Auto Update from Geometry**, and then check the box again to fit the geometry within the boundary.

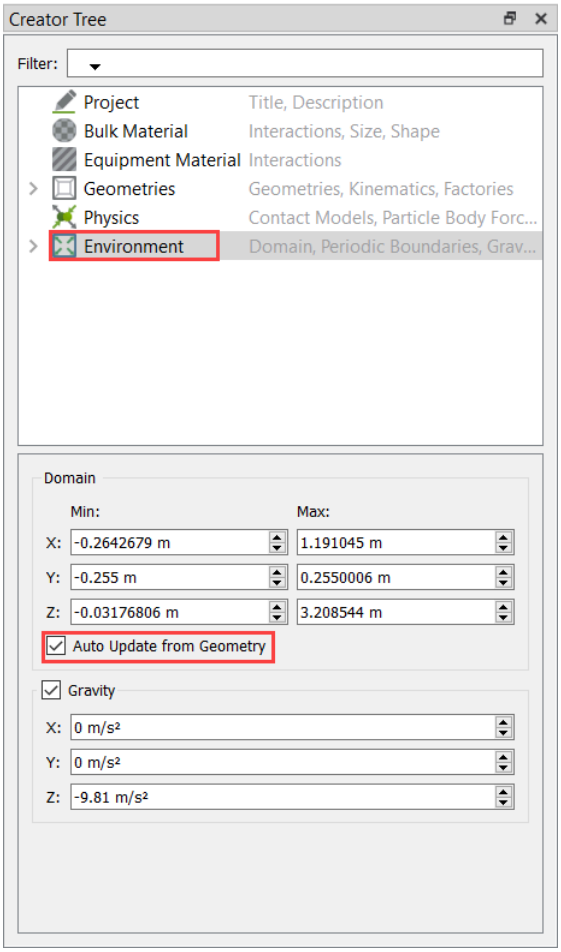

Creator Tree
Filter:
Project
Title, Description
Bulk Material
Interactions, Size, Shape
Equipment Material
Interactions
Geometries
Geometries, Kinematics, Factories
Physics
Contact Models, Particle Body Forc...
Environment
Domain, Periodic Boundaries, Grav...
Domain
Min:
Max:
X: -0.2642679 m
1.191045 m
Y: -0.255 m
0.2550006 m
Z: -0.03176806 m
3.208544 m
Auto Update from Geometry
Gravity
X: 0 m/s²
Y: 0 m/s²
Z: -9.81 m/s²

Figure 1423:

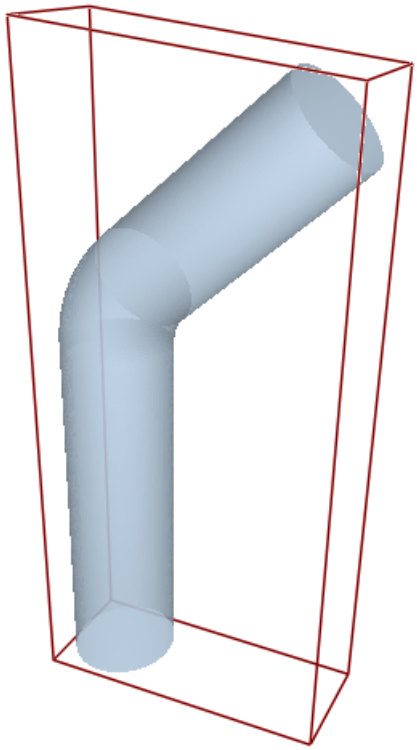


Figure 1424:

Define the Bulk Materials and Equipment Material

In this step, you will define the material models for the heavy and light bulk material and the equipment material.

1. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.
2. Rename the material to `Heavy`.
3. In the Creator Tree, set the Solids Density property to `900` kg/m^3.
   You will use the default values for other properties for this tutorial.

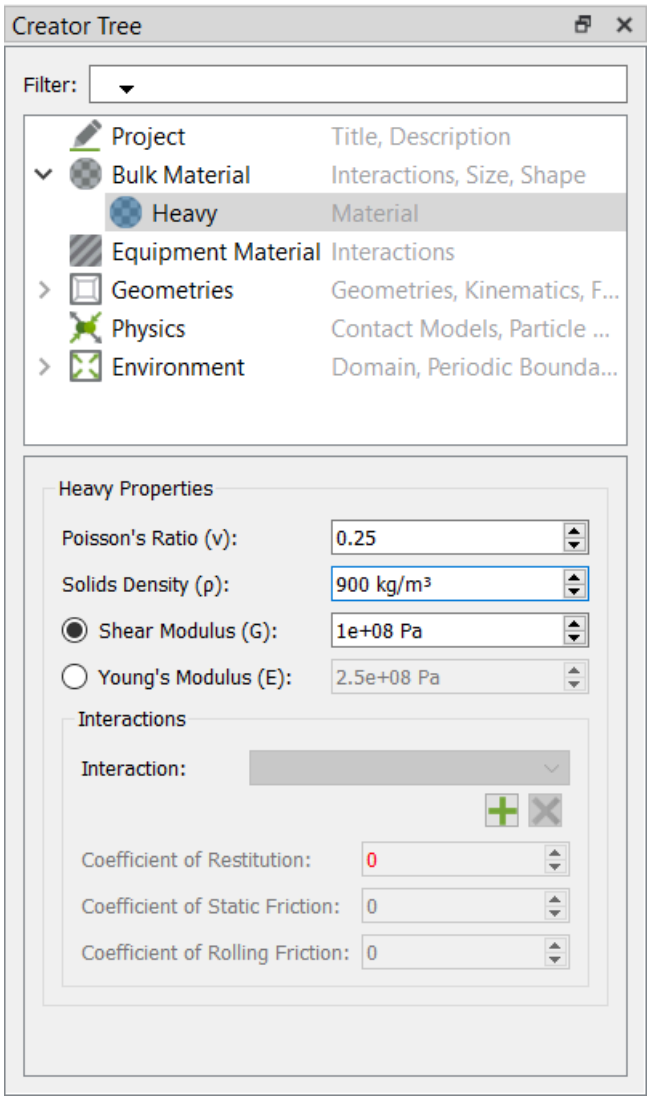


Figure 1425:

4. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, click **OK**.
5. In the Creator Tree, right click **Heavy** and select **Add Shape from Library** > **Dual Sphere Shape**.
6. Rename the particle to `Heavy particle`.
7. Under Heavy particle, click **Properties**.
8. In the Heavy particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

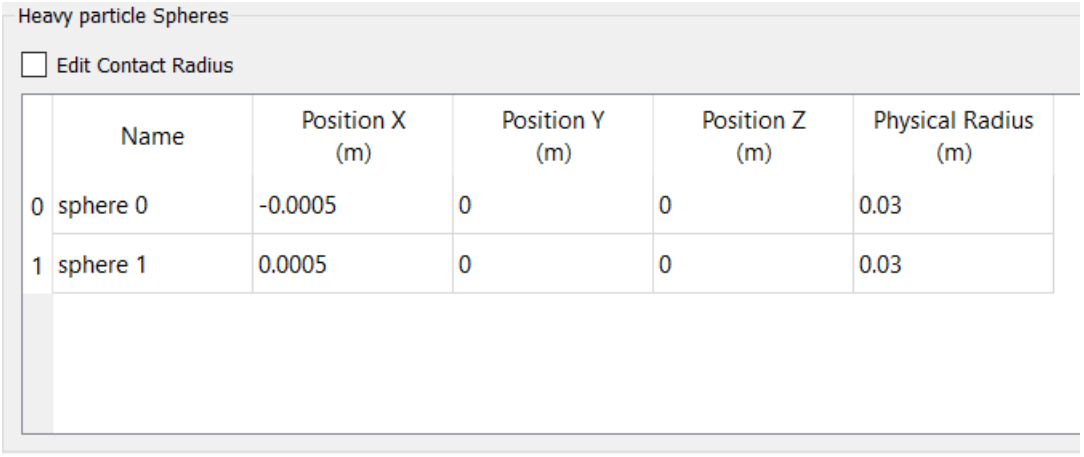


Figure 1426:

9. In the Creator Tree, click **Calculate Properties**.

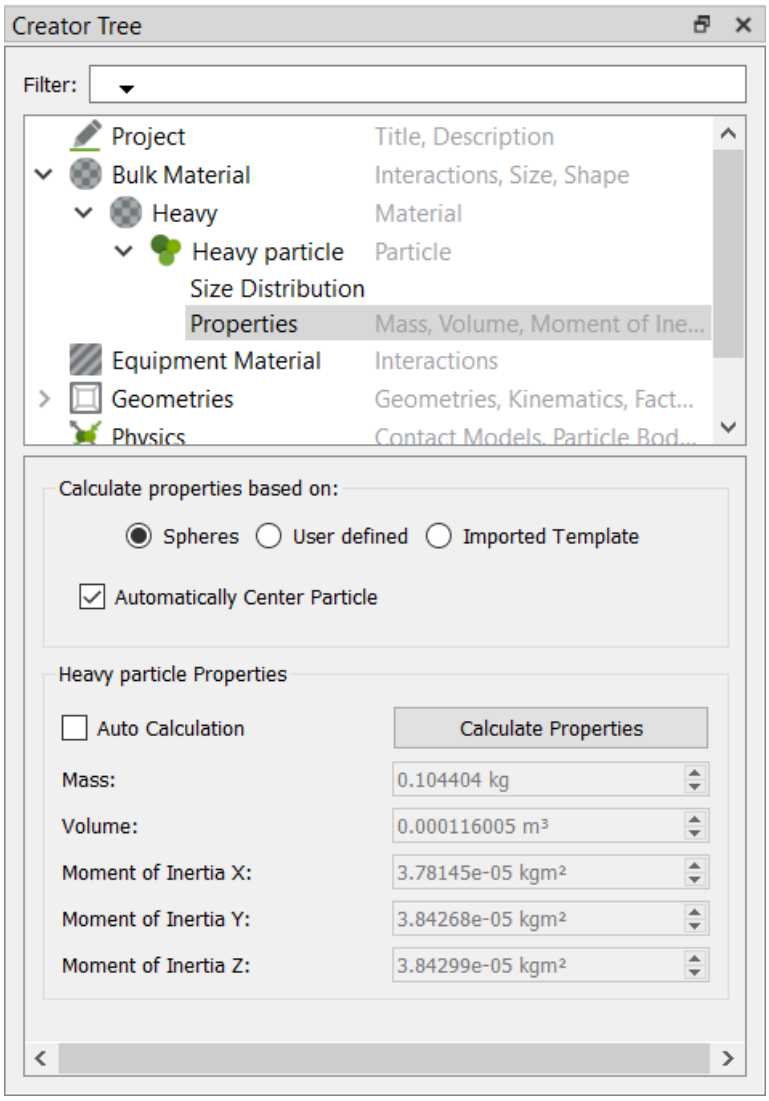


Figure 1427:

10. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.

11. Rename the material to `Light`.

12. In the Creator Tree, set the Solids Density property to `100` kg/m^3.

You will use the default values for other properties for this tutorial.

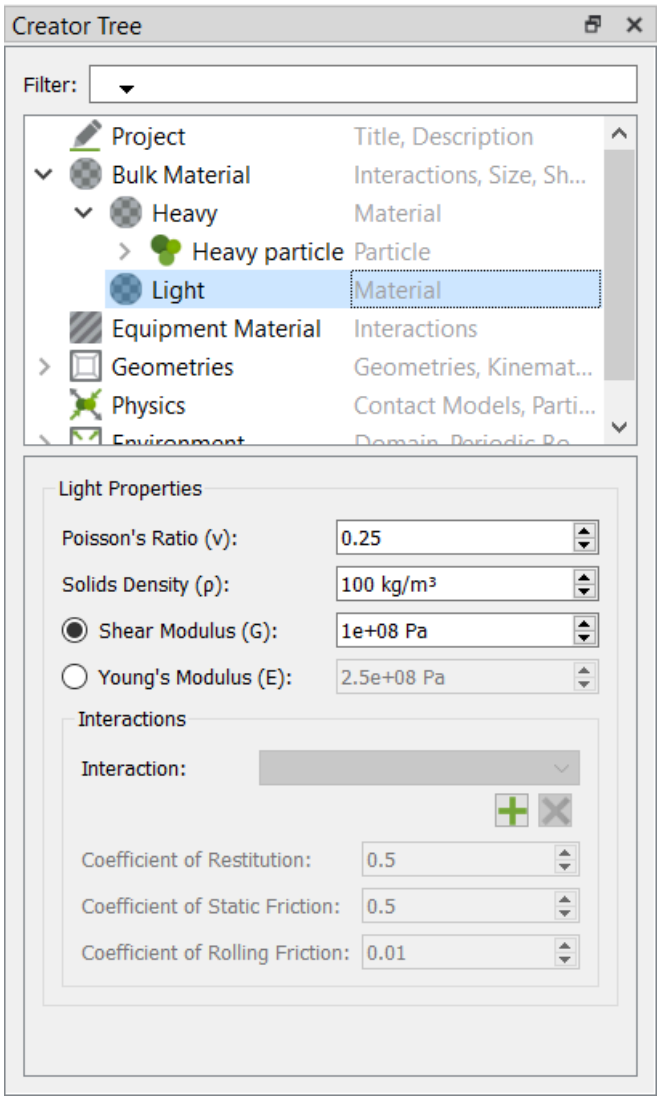


Figure 1428:

13. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.
14. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.
15. In the Creator Tree, right click on **Light** and select **Add Shape from Library** > **Dual Sphere Shape**.
16. Rename the particle to `Light particle`.
17. Under Light particle, click **Properties**.
18. In the Light particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

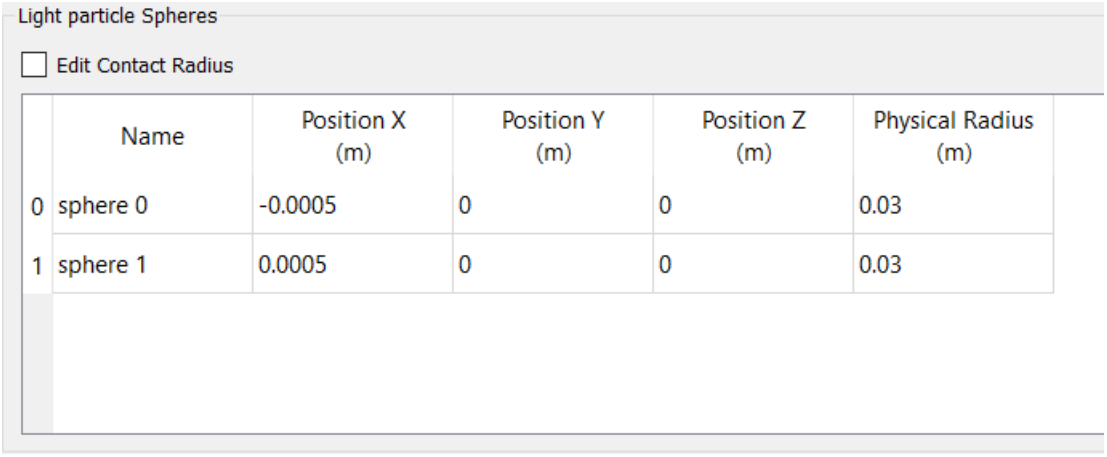


Figure 1429:

19. In the Creator Tree, click **Calculate Properties**.

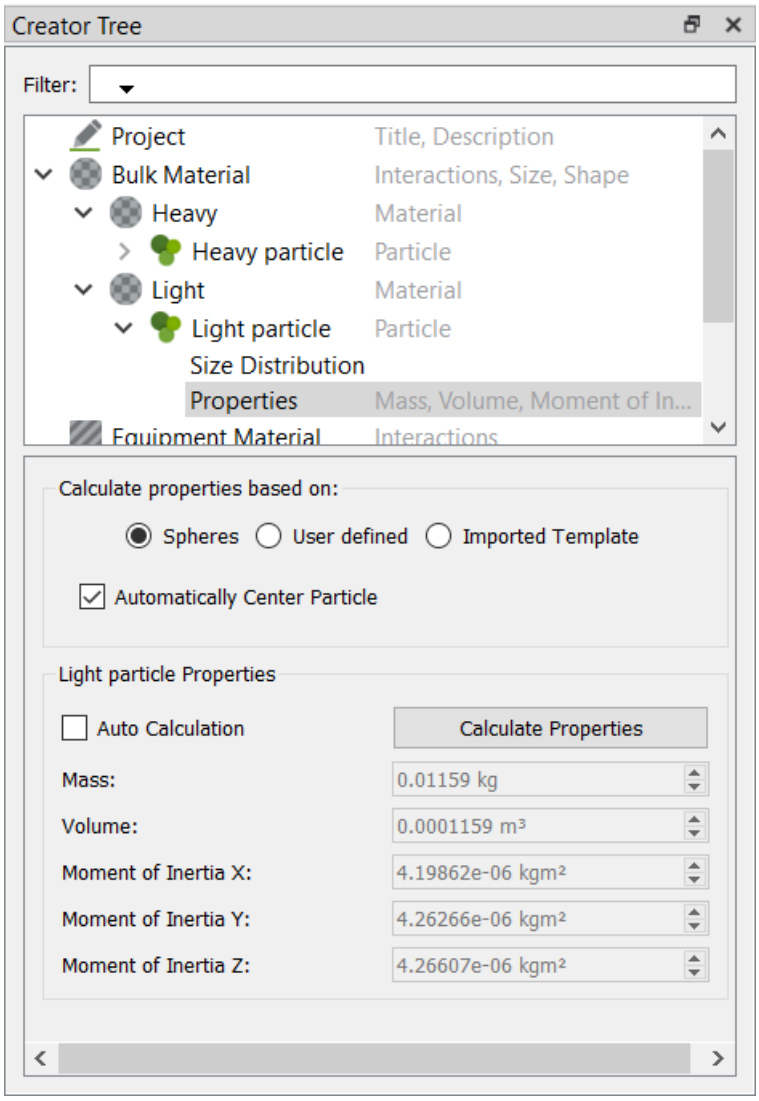


Figure 1430:

20. In the Creator tree, right-click **Equipment Material** and select **Add Equipment Material**. Rename it to `Steel`.

21. Set the Density to `7800` kg/m^3.

22. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.

23. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.

24. Save the model.

Create the Particle Factory

1. Expand **Geometries** under the Creator Tree tab. Next, right-click the **Particle inlet** surface group and select **Copy Geometry** > **Single Copy**.
2. Rename the new geometry section to `Particle_factory`.
3. Right-click the **Default Wall** geometry section in the Creator Tree and select **Merge Geometry(s)**.
4. In the **Merge Geometry** dialog, select the **Particle_inlet** and then click **OK**.
5. Click the **Default Wall** geometry section, set the Type to **Physical**, and the Material to **Steel** (if not set already).

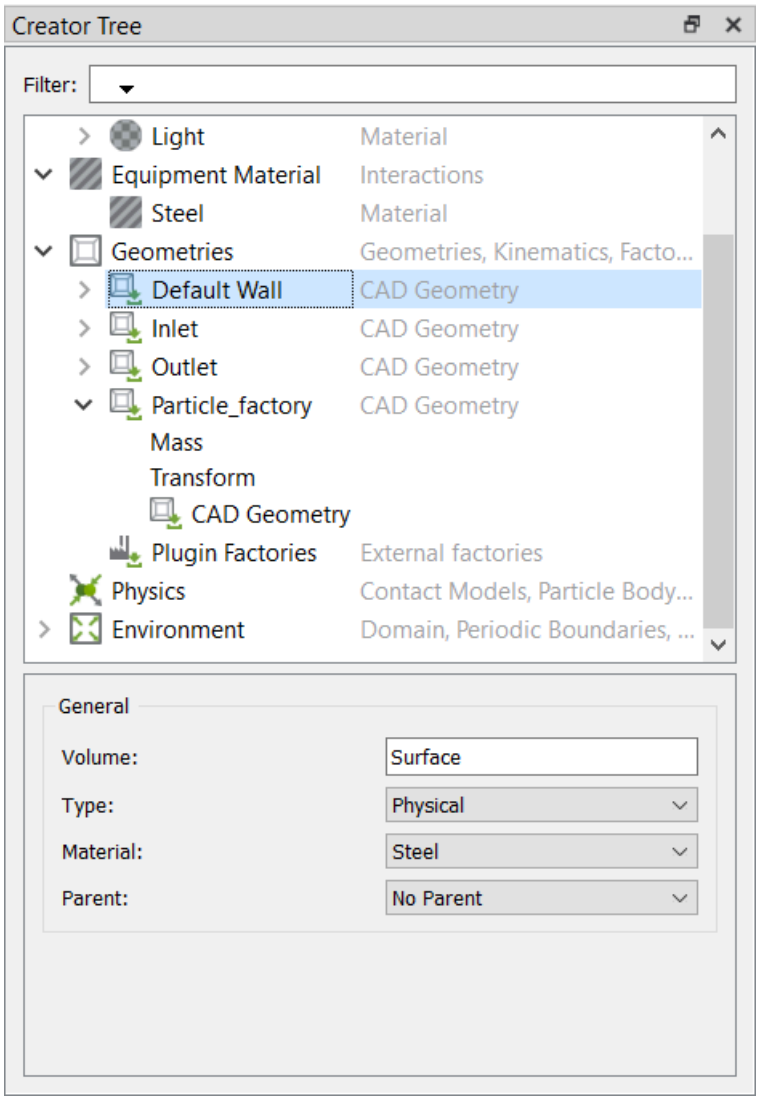


Figure 1431:

6. In the Creator Tree, click the **Inlet** section and change the Type to **Virtual**.
7. Similarly, change the Type to **Virtual** for the Outlet and Particle_factory sections.
8. Under Particle_factory, click **Transform**. Set the X-Position to `0.035` m

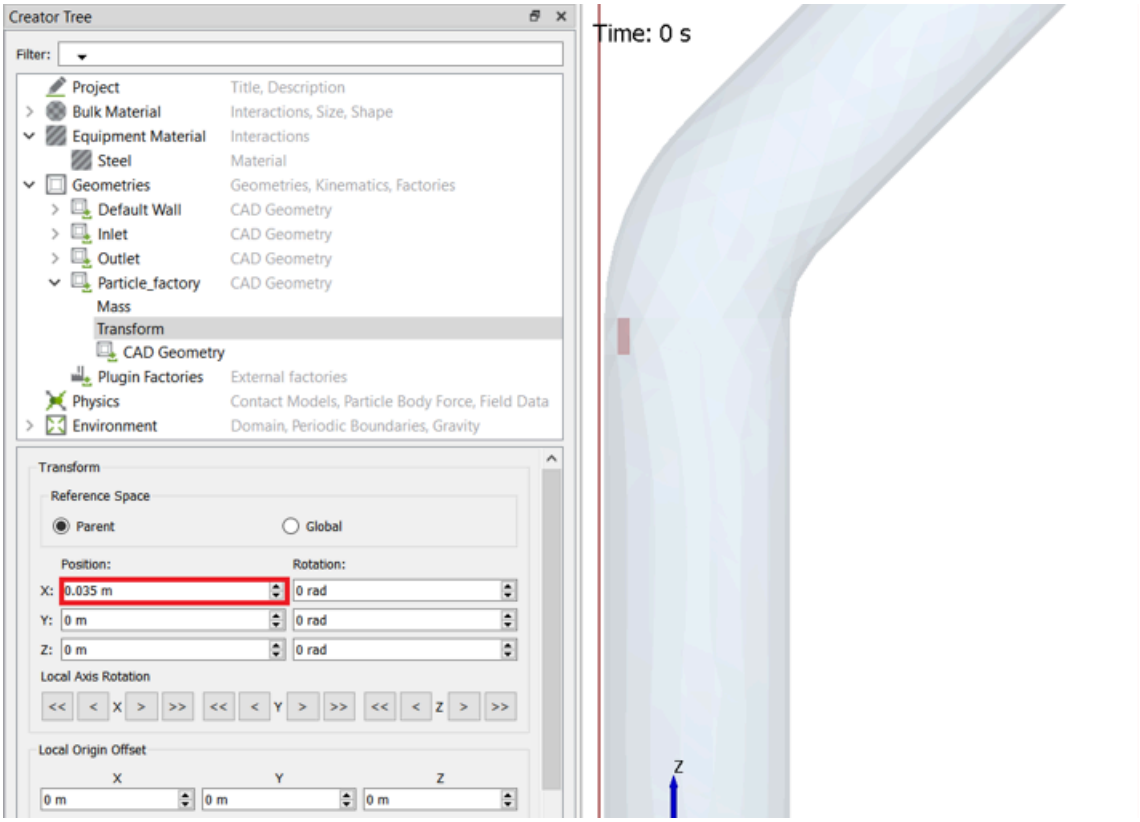


Figure 1432:

> **Note:** Set the Opacity value to 0.2 to see the transformed surface location inside the pipe geometry.
>
> This is done to make sure that the particles are generated inside the fluid domain.

Define the Particle Factory

Now that the bulk material, geometry sections, and equipment materials are defined, you need to create a particle factory to generate the particles. You will create one factory for each bulk material.

1. In the Creator Tree, right-click **Particle_factory** and select **Add Factory**.
2. Rename the new factory to `Heavy factory`.
3. Set the particle generation parameters as shown in the figure below.

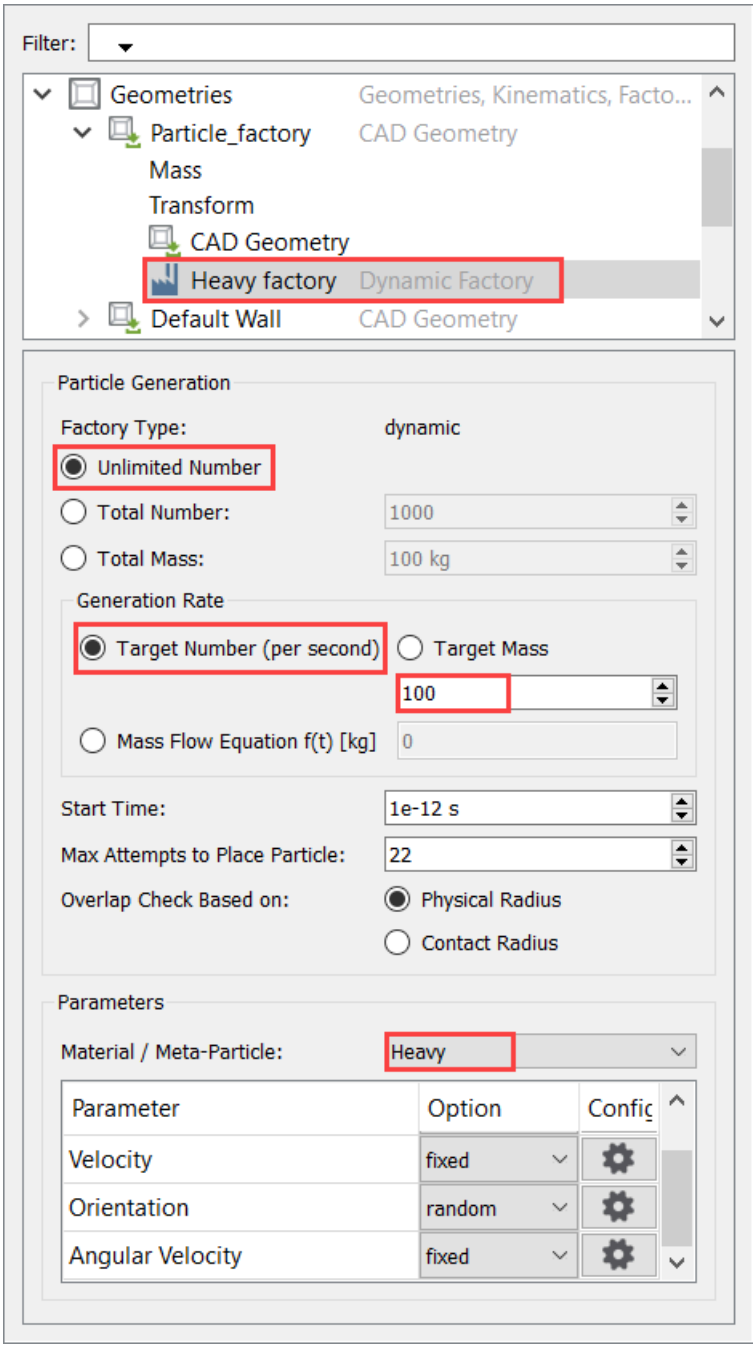


Figure 1433:

4. Click ⚙ besides Velocity, set the X-velocity to `1` m/s, and then click **OK**.
5. Repeat steps 1-4 to create another factory named `Light factory` using the same parameters but with **Light** as the Material.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).
   When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.
3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and deactivate the **Auto Time Step** checkbox.

   > **Note:** The Auto time step should never be used for AS-EDEM bidirectional coupled simulations.

3. Enter a value of `2e-5` for Fixed Time Step.

   When setting the time step for the DEM model, make sure that the CFD time step is an integer multiple of DEM time step. In this case, the CFD time step is 1e-3, which is 50 times the DEM time step. This ensures that the physical time in both the solvers match at the end of each data exchange interval. Also, the recommended range for the CFD-to-DEM time step ratio is 50-200 for most cases.
4. Set the Total Time to `1` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `4` R min.

   Generally, a value in the range of 3-6 R_{min} is recommended as the optimum cell size. The cell size in EDEM does not have any impact on the accuracy of the simulation and affects only the run time.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

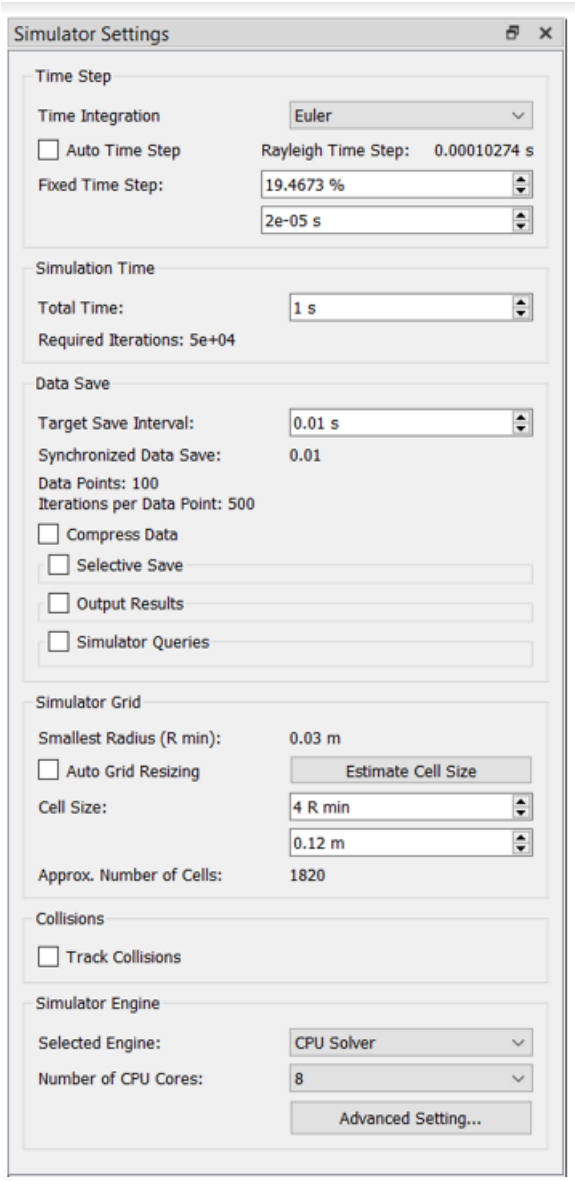


Figure 1434:

7. Once the simulation settings have been defined, save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1435:

Once the Coupling server is activated, the icon changes.

Figure 1436:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1437:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Optional: Set the number of processors to **4** or **8** based on availability.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
7. Click **Run** to launch AcuSolve.

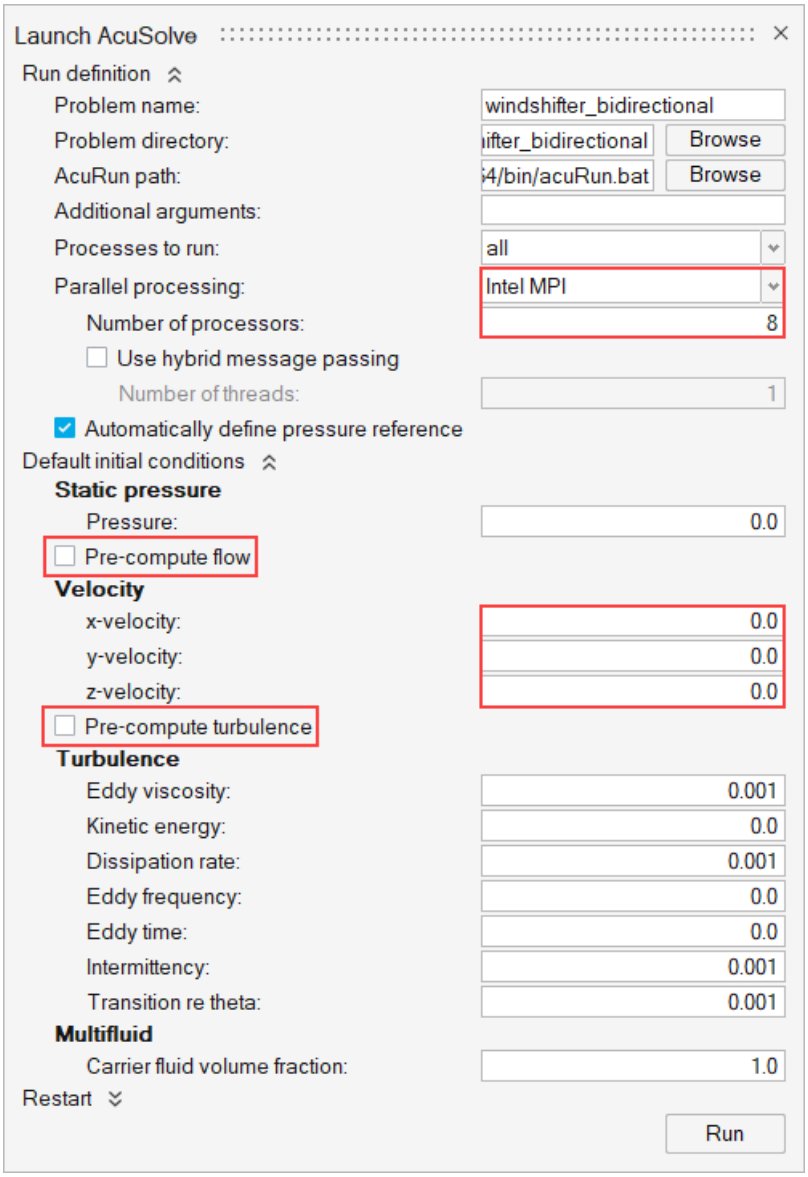


Figure 1438:

If the coupling between AcuSolve and EDEM is successful, a message will be printed in the AcuSolve Log file before the first time step.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 3.719000e+000 6.761500e+001 Sec
acuSolve: Input Memory Usage        = 7.000000e+001 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Interface thickness is 1.449828e-001
acuSolve: Init. CPU/Elapse time     = 5.000000e-001 7.100000e-002 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 1.000000e-003 ; time= 0.000000e+000
```

Figure 1439:

Analyze the Results

AcuSolve Post-Processing

1. Once the solution is completed, navigate to the **Post** ribbon.
2. From the menu bar, click **File** > **Open** > **Results**.
3. Select the AcuSolve `.log` file in your problem directory to load the results for post-processing. The solid and all the surfaces are loaded in the Post Browser.
4. Select the **Left** face on the view cube to align the model to the x-z plane.

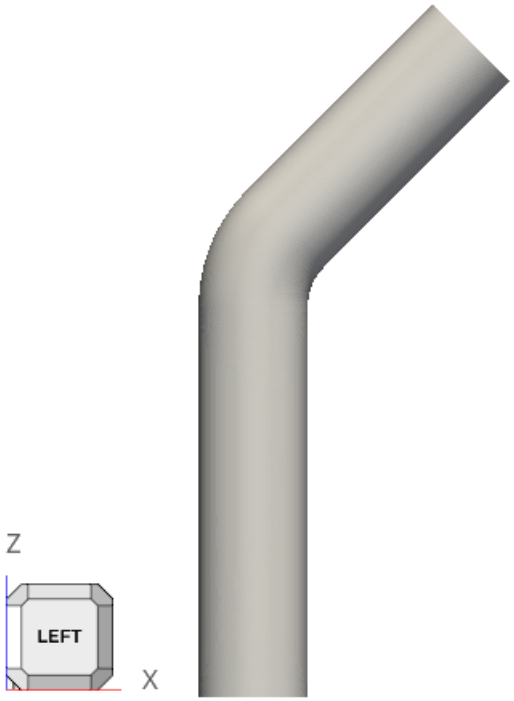


Figure 1440:

5. In the Post Browser, turn off the display of the boundary surfaces by clicking on the icon next to Flow Boundaries.

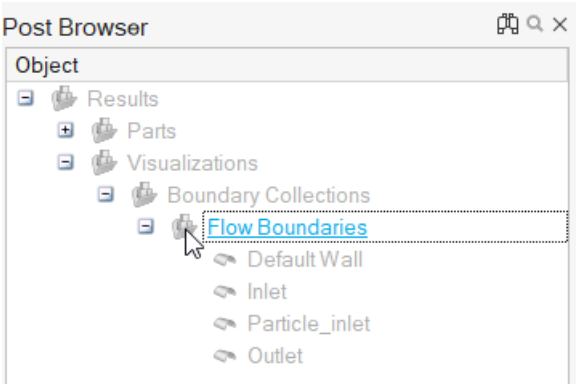


Figure 1441:

6. Click the **Slice Planes** tool.

Figure 1442:

7. In the modeling window, click on the plane that is parallel to the screen (the x-z plane).
8. In the slice plane microdialog, click to create the slice plane.
9. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
10. Change the upper bound of the legend to `36`.
11. Click and set the Colormap name to **Rainbow Uniform**.

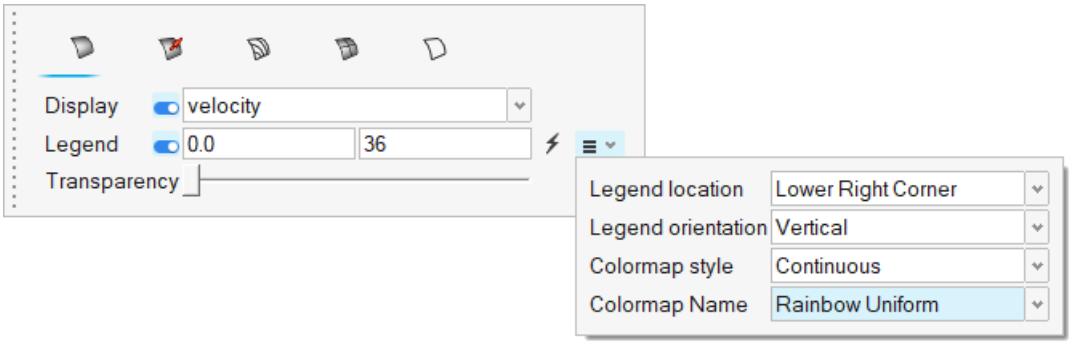


Figure 1443:

12. Click on the guide bar.
    The velocity contour should look like the one shown below at the beginning of the simulation.

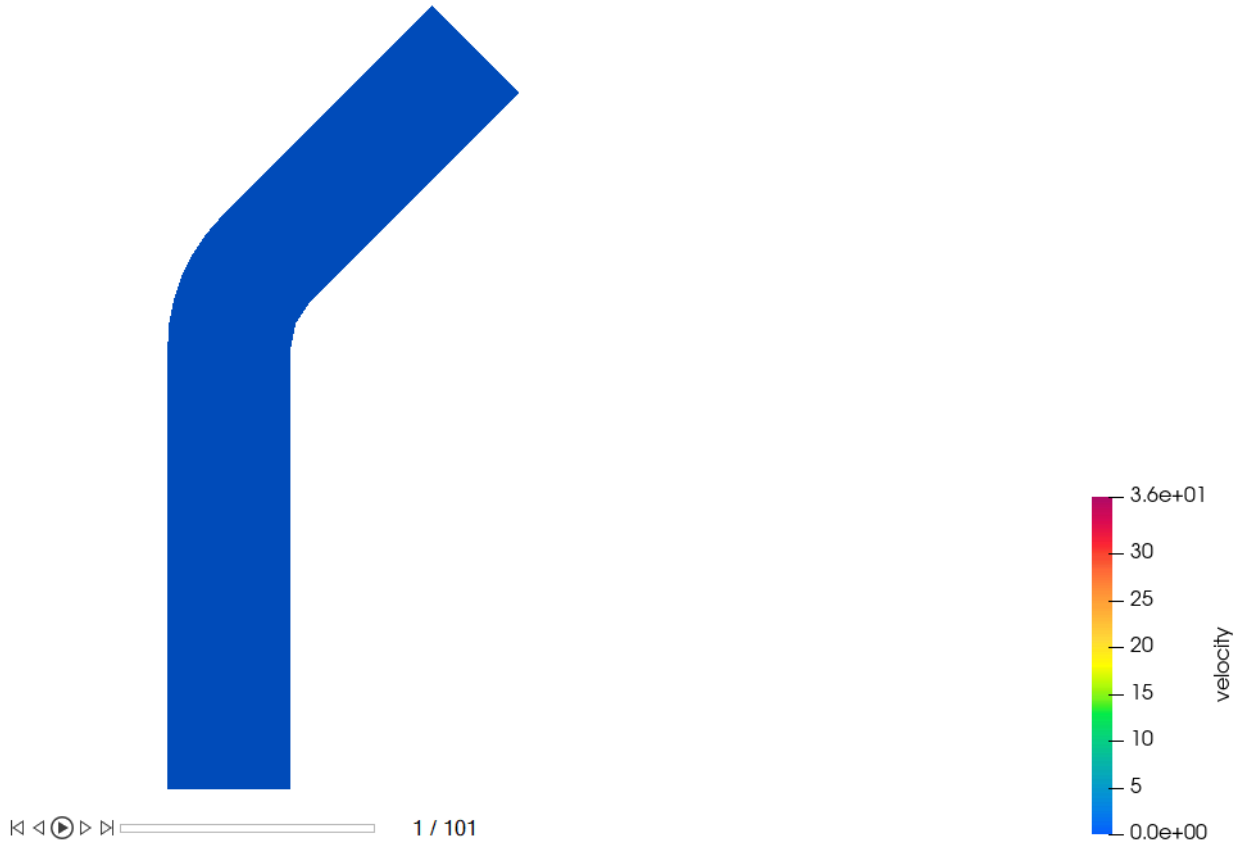


Figure 1444:

13. Plot the contours at 0.25s, 0.5s, 0.75s, and 1.0s by dragging the slider on the bottom to the 26th, 51th, 76th, and 101st frames.

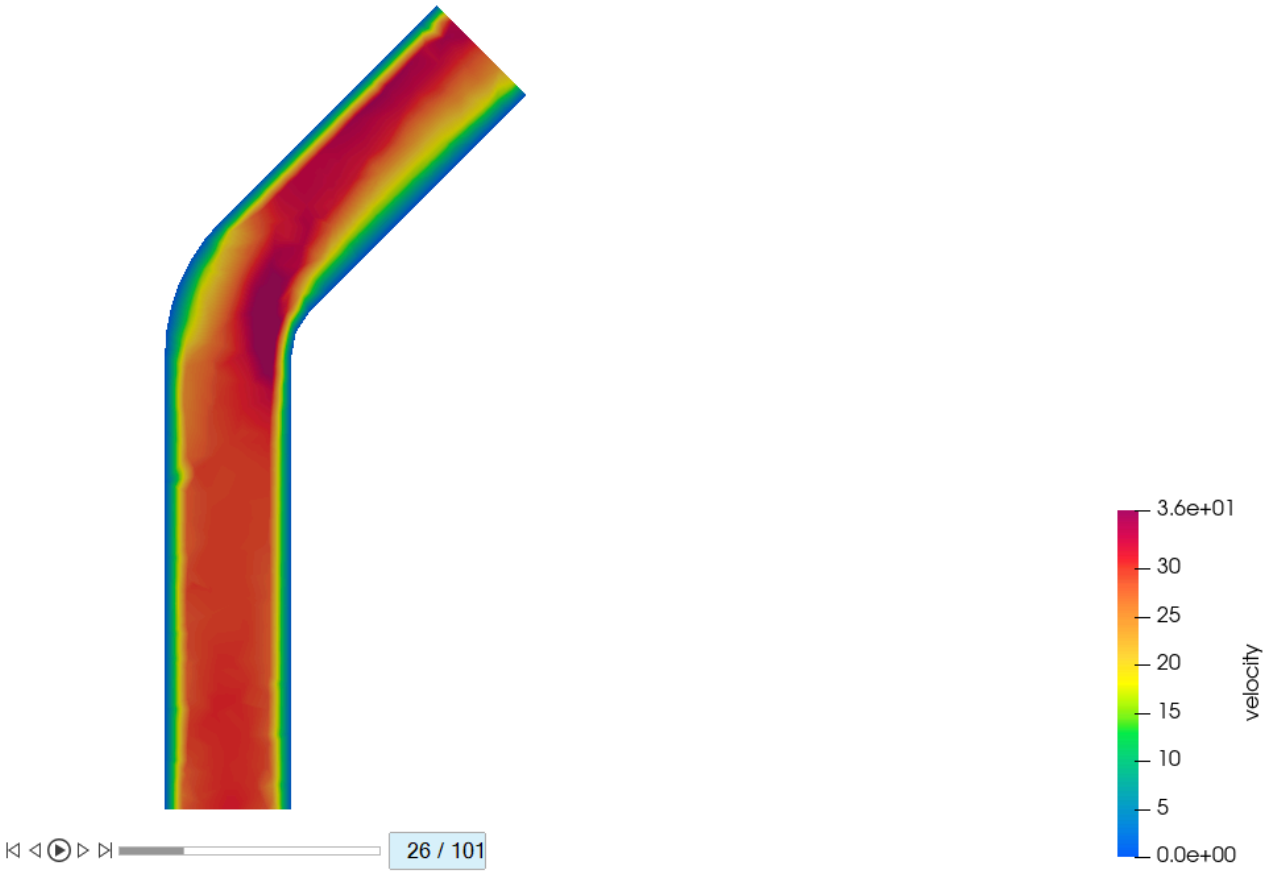


Figure 1445:

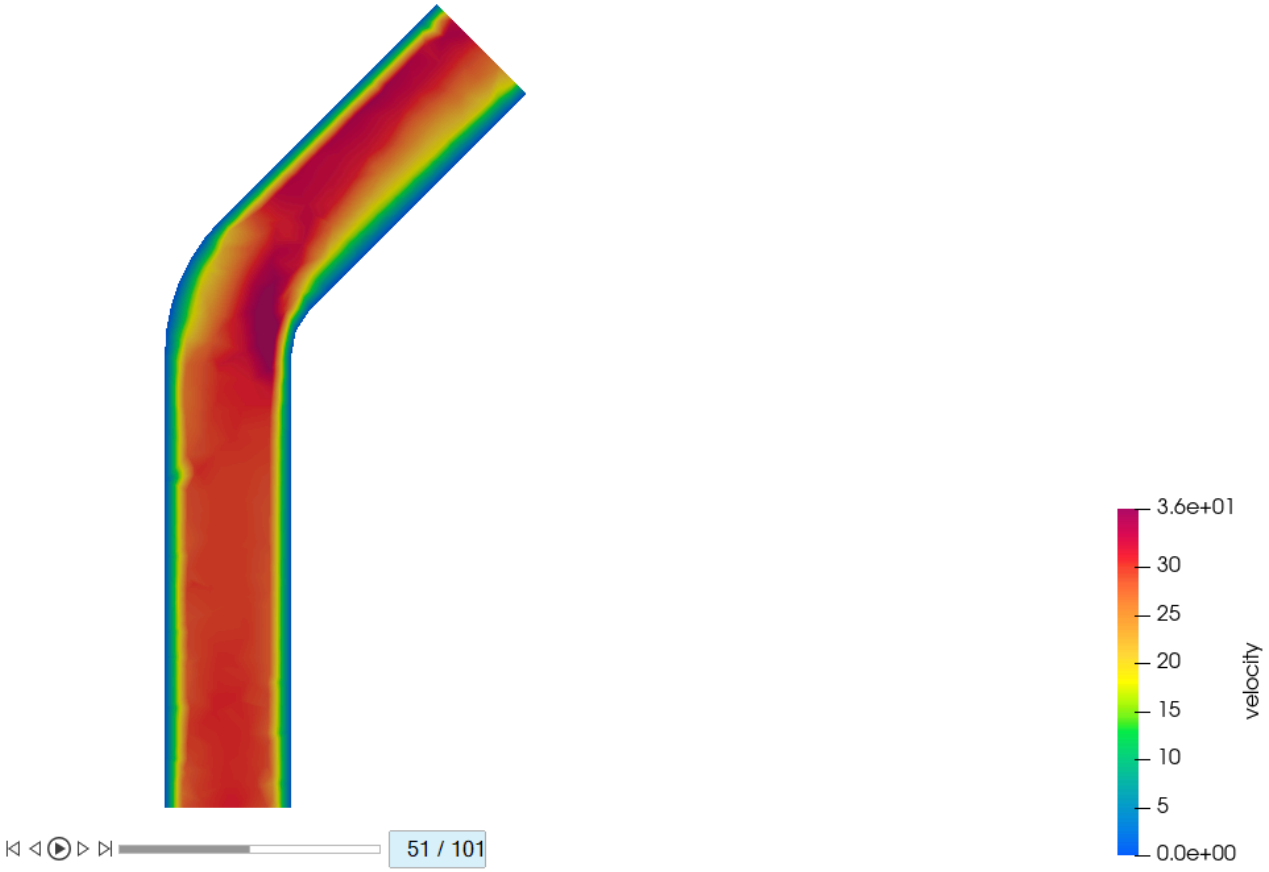


Figure 1446:

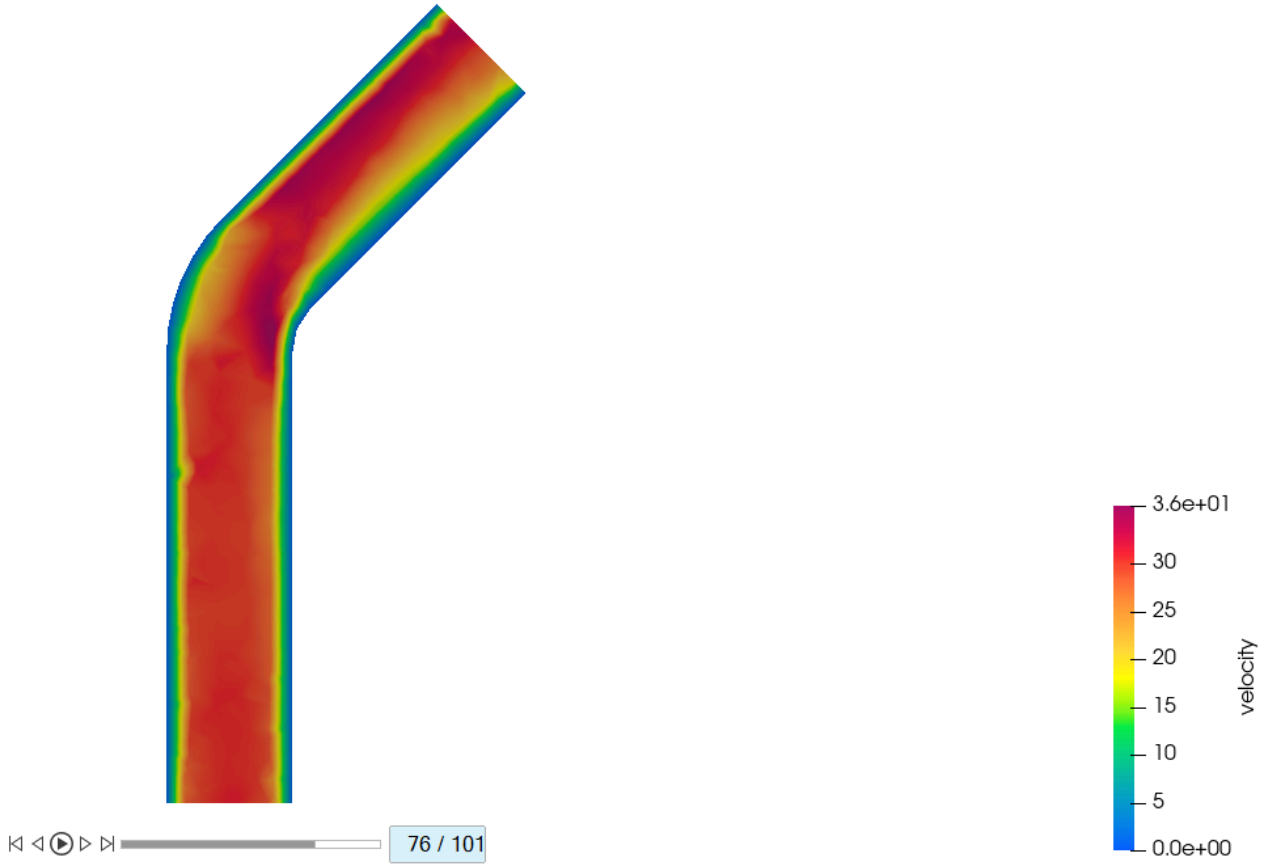


Figure 1447:

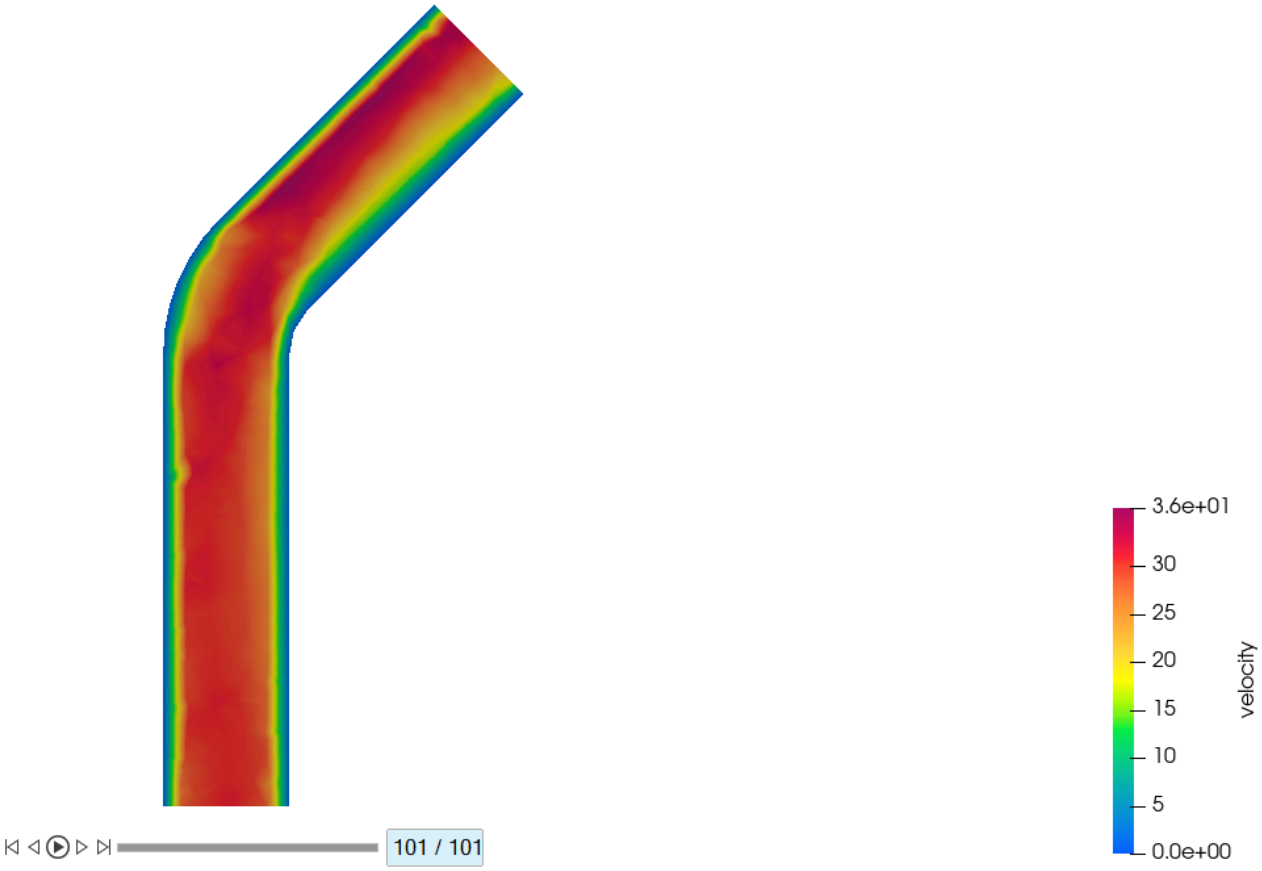


Figure 1448:

14. Right-click **Slice Plane 1** in the Post Browser and select **Edit**.
15. In the microdialog, change the Display variable from velocity to **volume fraction edem particle** and set the upper bound of the legend to `0.186`.
16. Click ✓ on the guide bar.

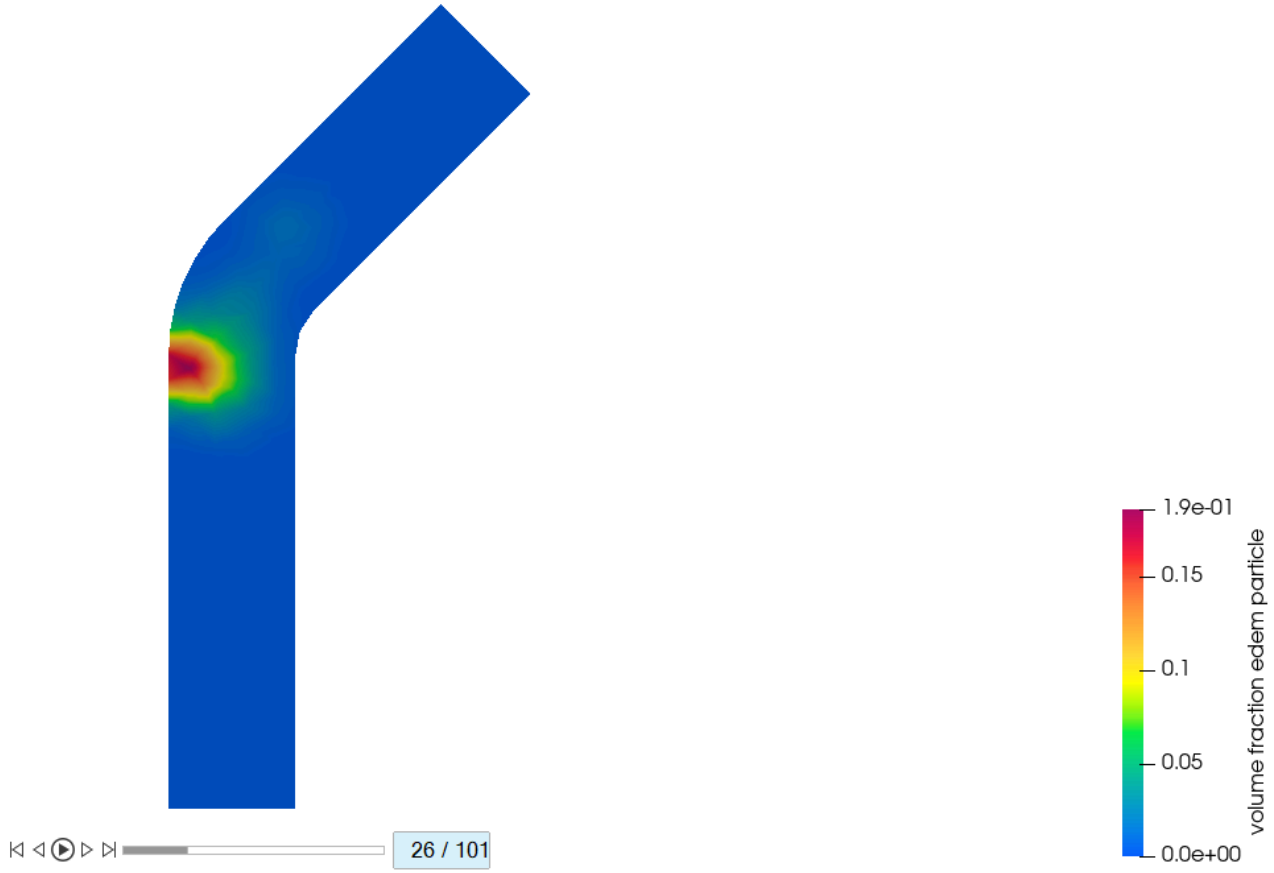


Figure 1449:

17. Plot the contours at 0.5s (step=51/101), 0.75s (step=76/101), and 1.0s (step = 101/101) to see the volume fraction edem particle distribution inside the fluid domain.

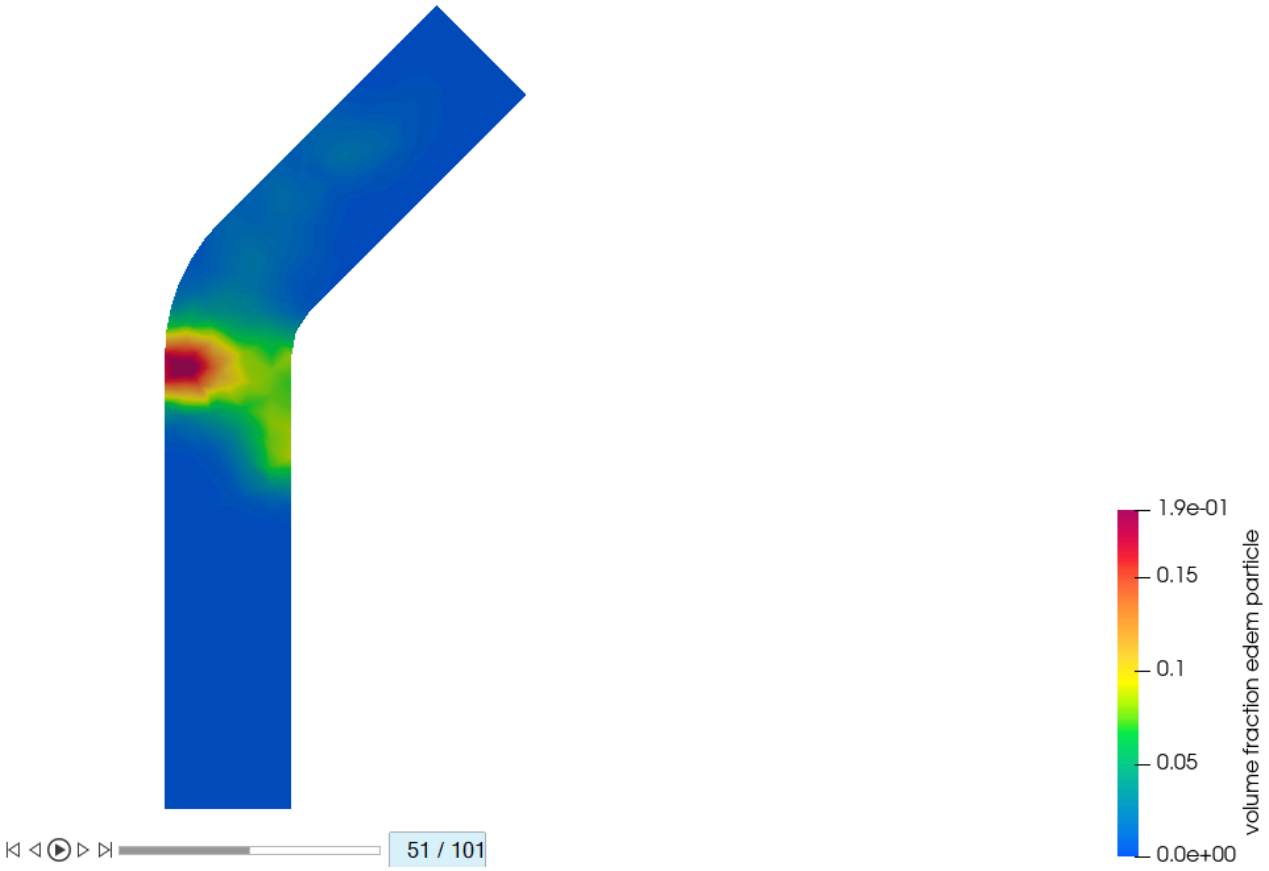


Figure 1450:

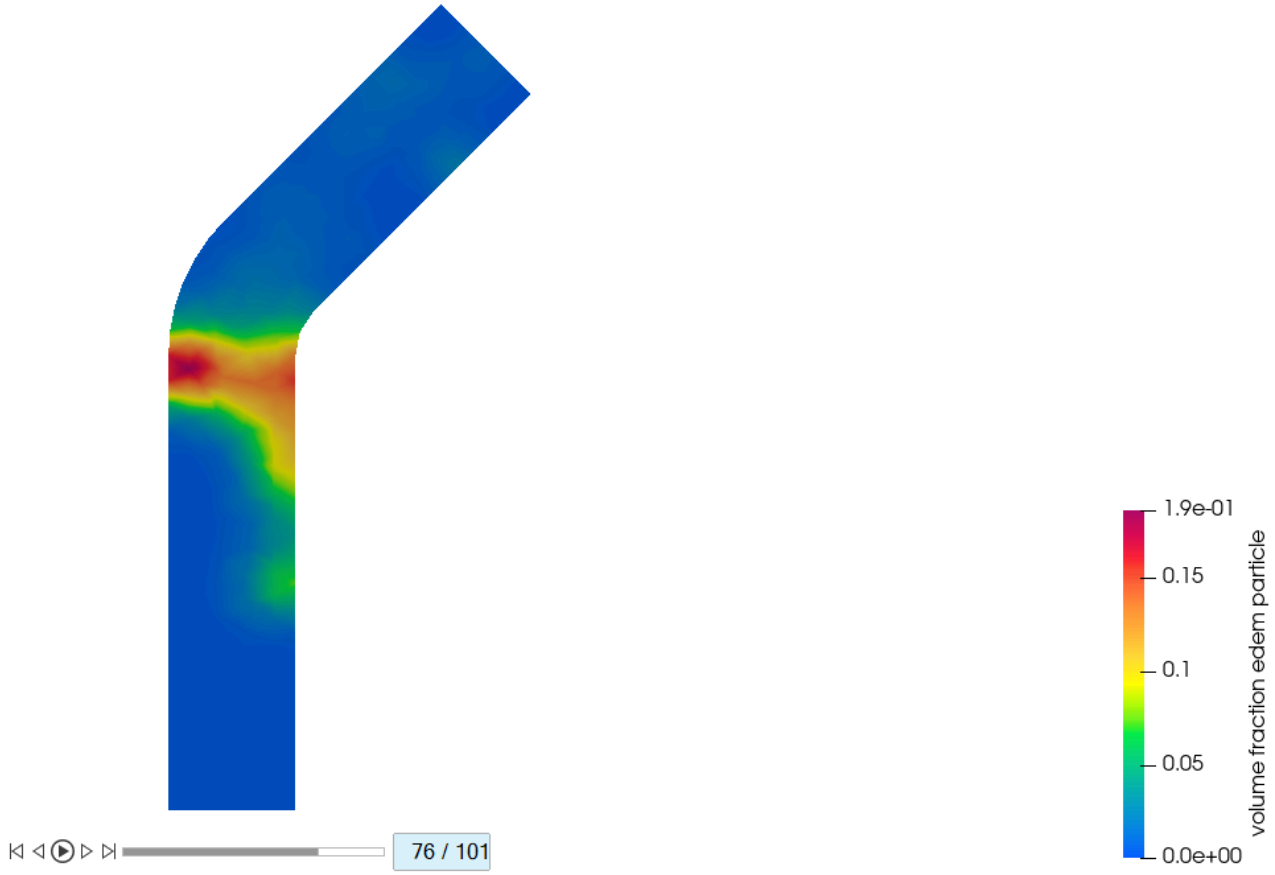


Figure 1451:

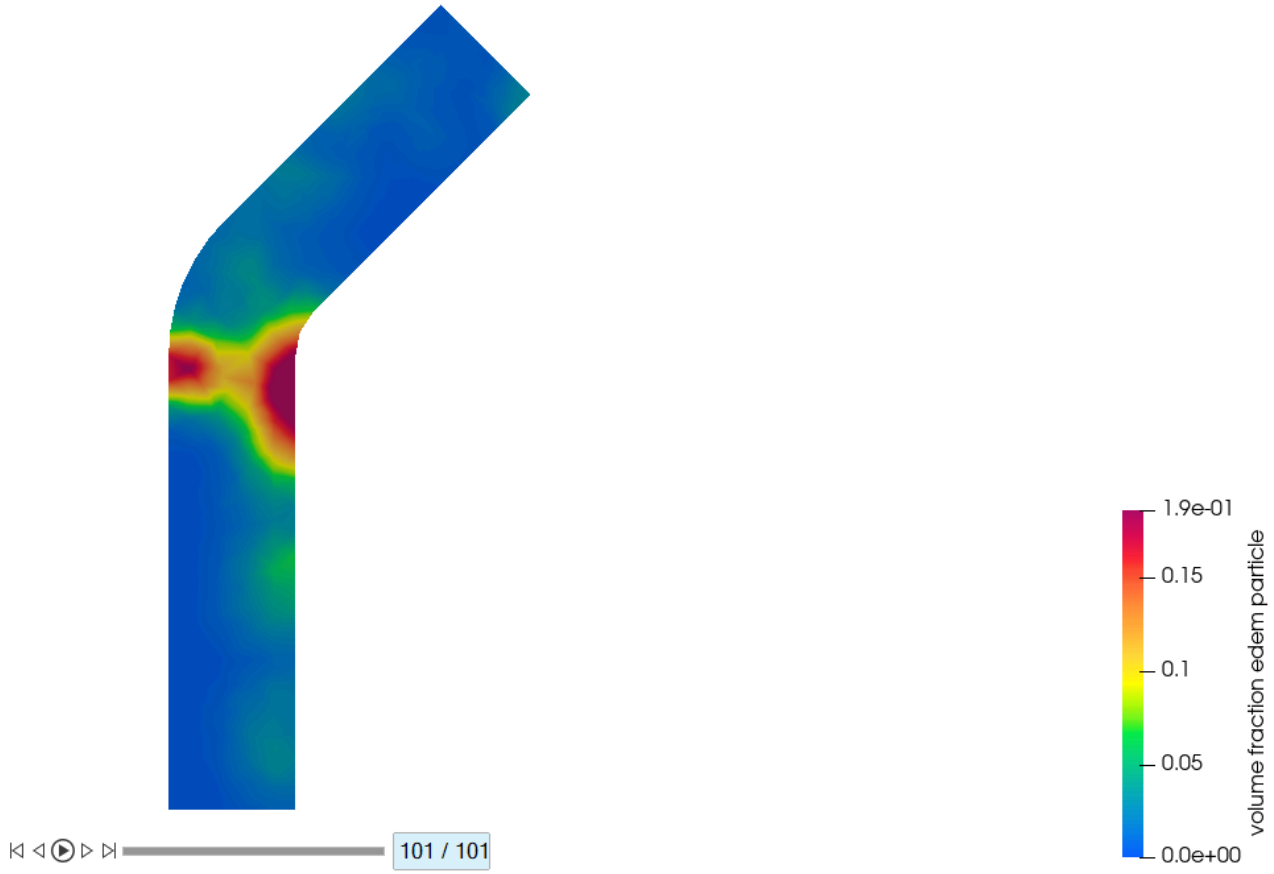


Figure 1452:

EDEM Post-Processing

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then click **Default Wall**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

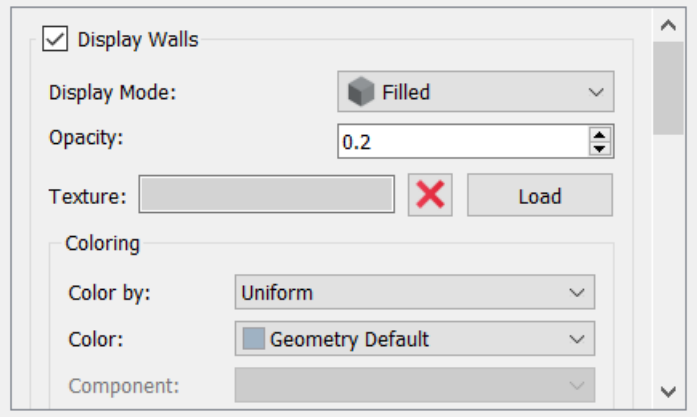


Figure 1453:

4. In the Analyst Tree, expand **Particles** and click **Heavy particle**.
5. Change the display color to **Magenta**.

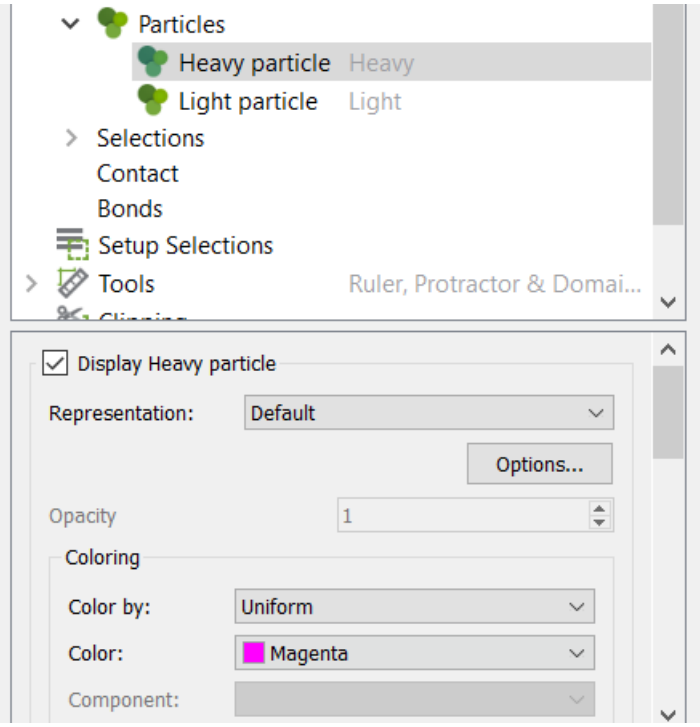


Figure 1454:

6. Click on **Light particle** and set the display color to **Green**.

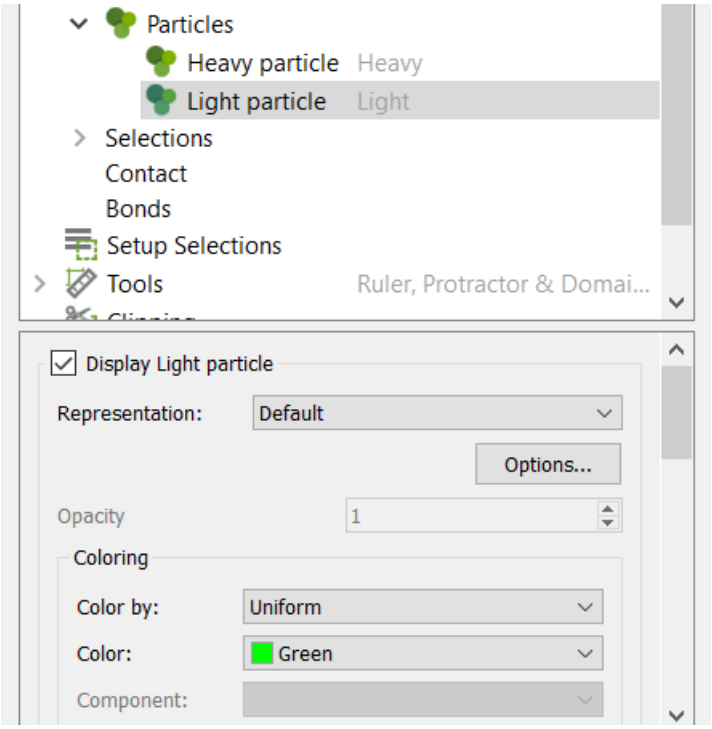


Figure 1455:

7. On the menu bar, set the time to `0` by clicking:

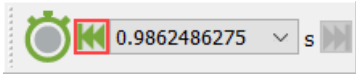


Figure 1456:

8. Set the View plane to **+ Y**.

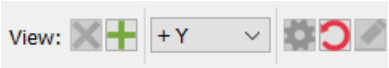


Figure 1457:

9. In the Viewer window, set the Playback Speed to `0.1x` then click on the play icon to play the particle flow animation.

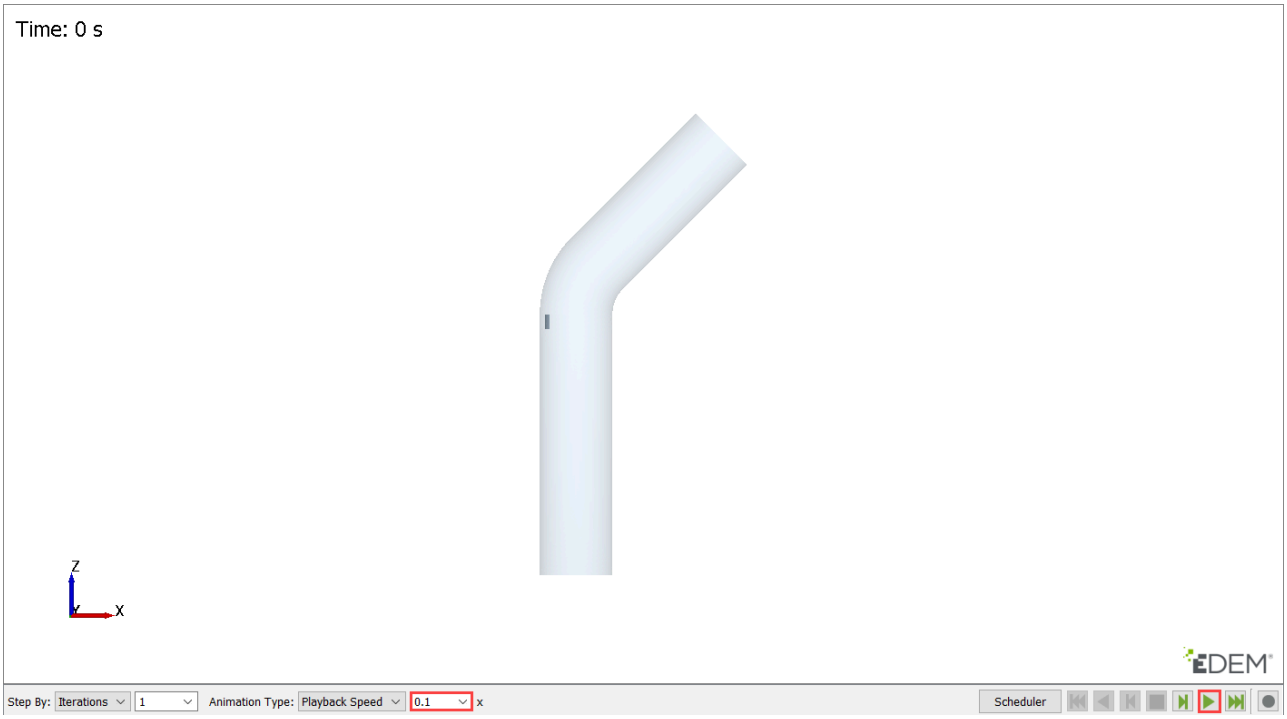


Figure 1458:

10. You can also plot the results at different timesteps by clicking the drop-down menu.

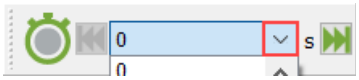


Figure 1459:

For the current case, the results are plotted at 0.25s, 0.5s, 0.75s, and 0.98s to see the particle distribution inside the fluid domain at both inlet and outlet.

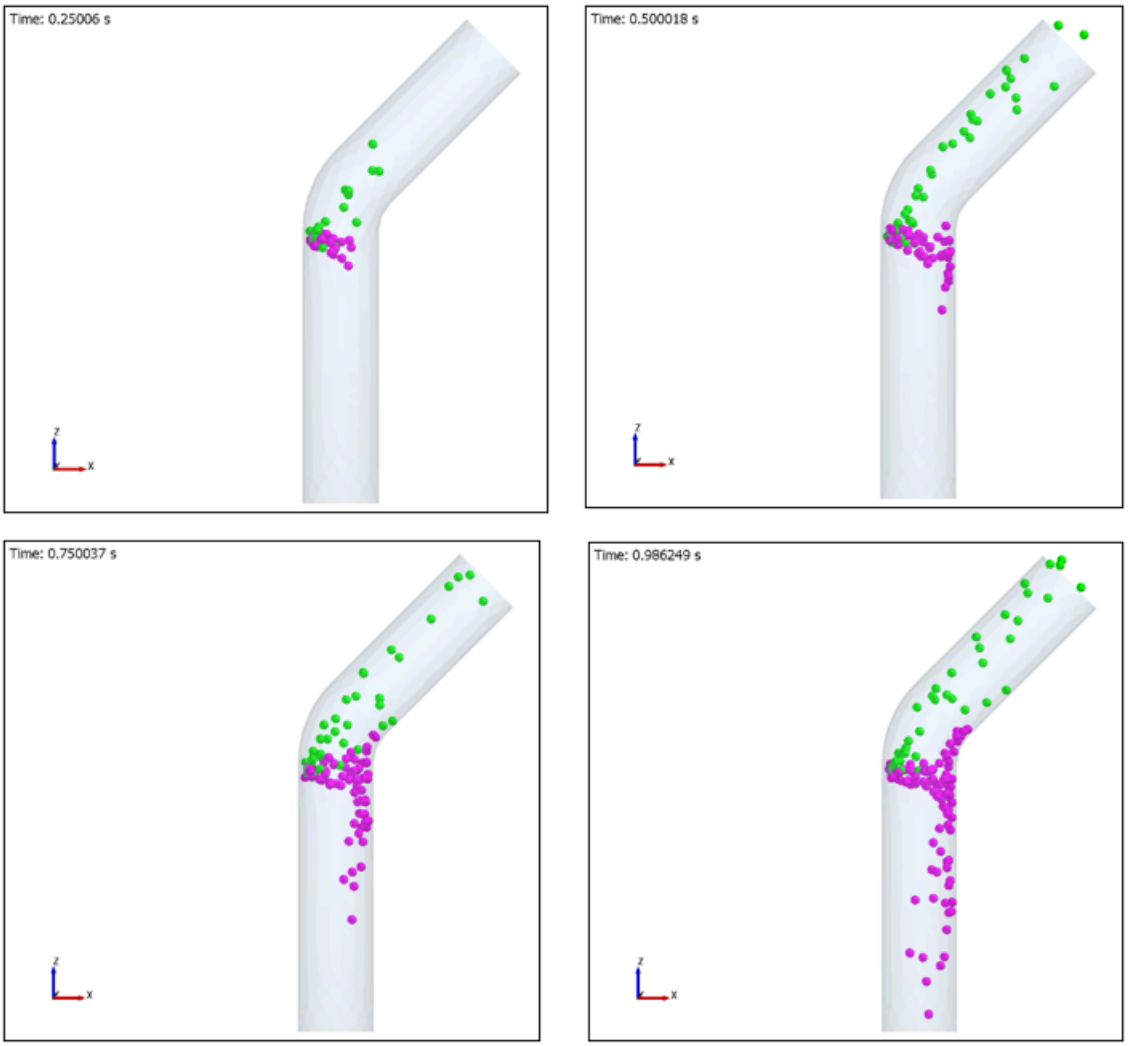


Figure 1460:

Observe that the lighter particles (green) get carried by the fluid and escape the domain through the outlet at the top and the heavier particles (magenta) stay inside the domain for a longer time while some of them fall through the bottom of the pipe.

11. On the menu bar, click the **Create Graph** icon .
12. In the Analyst Tree, change the plot type to Line by clicking Line.
13. Click the **X-axis** tab and verify that the values are set as shown in the figure below.

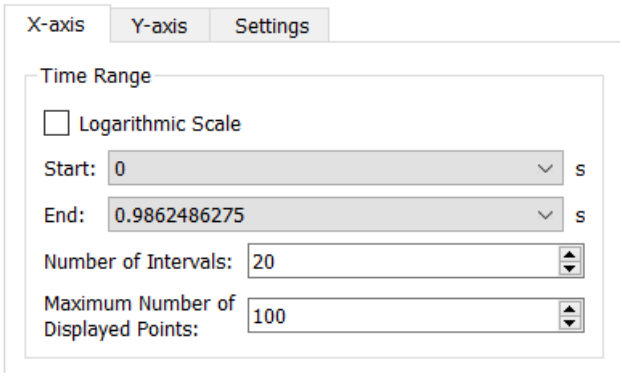


Figure 1461:

14. Click the **Y-axis** tab. Create a plot of average residence time of the heavy particle over the simulation time by setting the values as shown in the figure below.

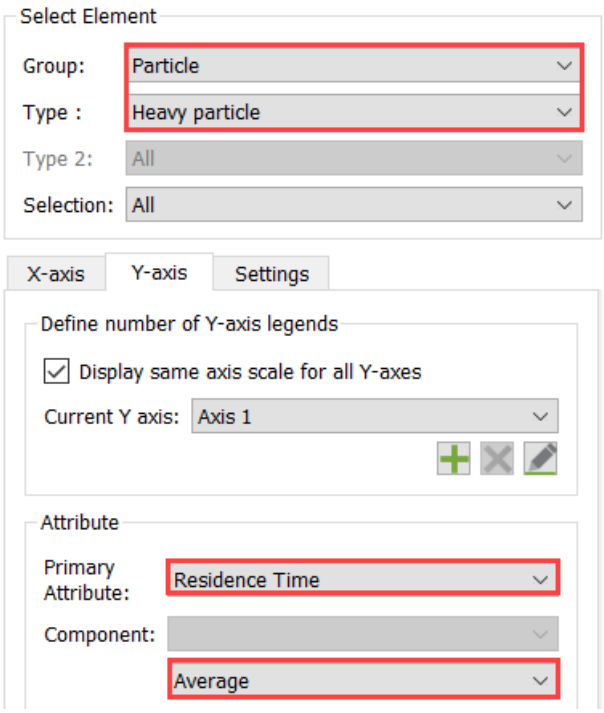


Figure 1462:

15. Click to add another Y-axis and set the Type to **Light particle**.

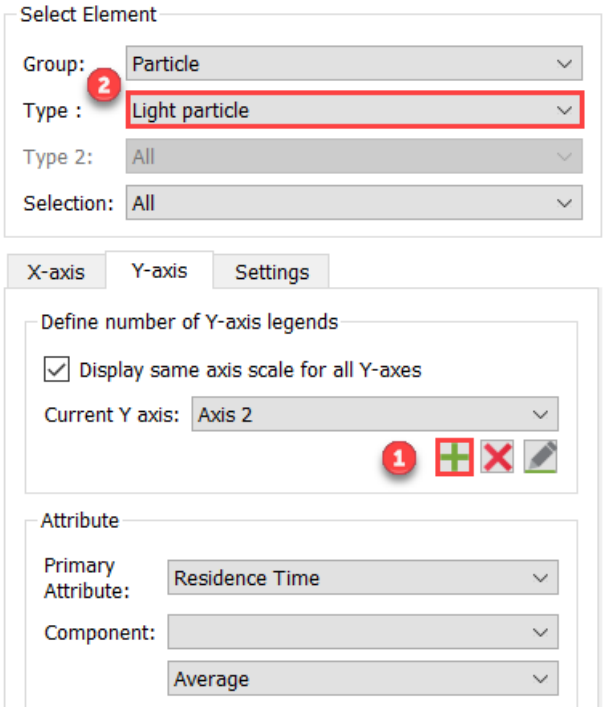


Figure 1463:

16. Leave all the other options unchanged and then click **Create Graph** to create a plot of average residence time of both the particles.

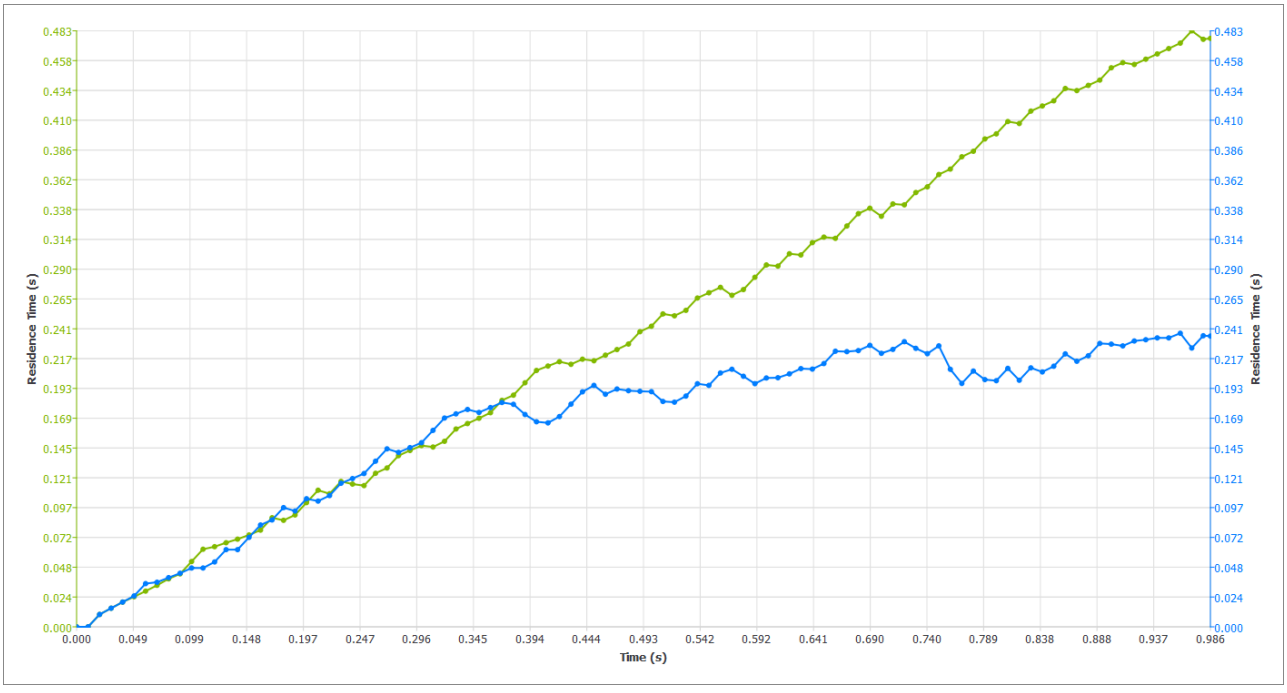


Figure 1464:

The plot in green is for the heavier particles and it can be observed that the average residence time is higher for the heavier particles compared to the lighter particles (blue).

Summary

In this tutorial, you learned how to setup and run a basic AcuSolve-EDEM bidirectional (two-way) coupling problem. In the first part, you set up the AcuSolve model in HyperMesh CFD and exported the geometry. Next, you imported the EDEM input files created by HyperMesh CFD, set up the EDEM model, and ran the coupled simulation. Once the coupled simulation was completed, you have learned how to create animations and plots in both HyperMesh CFD Post and EDEM Analyst.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/4ekcc9g3xs

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6103 AcuSolve - EDEM Bidirectional Coupling with Heat Transfer

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

This tutorial introduces you to the workflow for setting up and running a basic bidirectional coupling (two-way) simulation with heat transfer using AcuSolve and EDEM. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6103_windshifter_heat.hm`

Problem Description

The problem to be solved is shown schematically in Figure 1465. It is a windshifter model in which air enters the domain from the inlet at the bottom at a velocity of 20 m/sec and exits through the outlet. The temperature of the incoming fluid is 293 K. Particles at a temperature of 363 K are introduced into the fluid domain. As the particles move through the fluid domain, they exchange energy with the surrounding fluid and as a result their temperature drops down before they exit the fluid domain. The walls of the pipe are maintained at a constant temperature of 293 K.

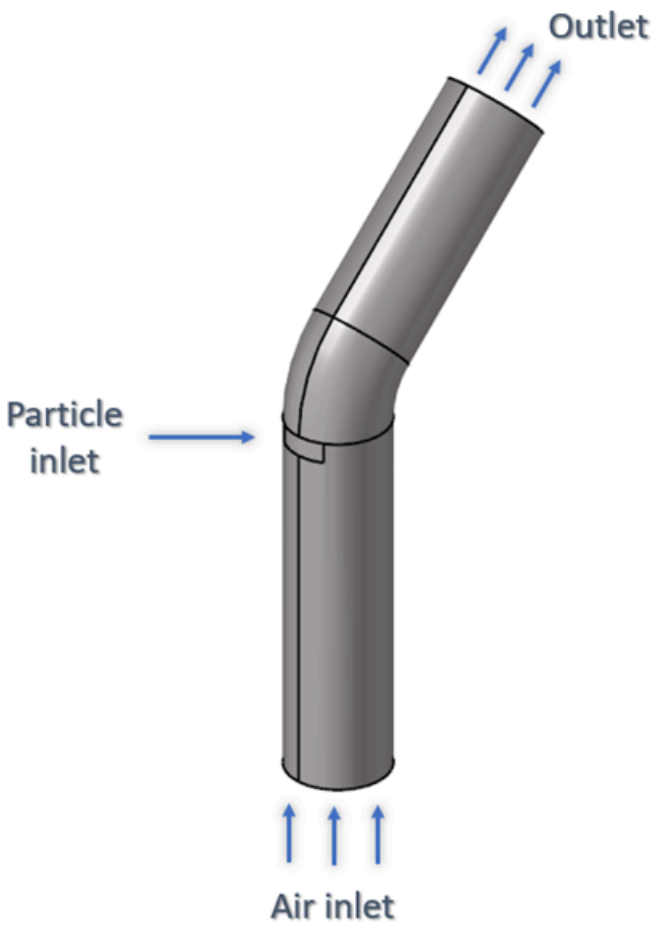


Figure 1465:

The model consists of a cylindrical pipe with a 45-degree bend. The radius of the pipe is 0.25 m and the particle inlet is located midway through the length of the pipe.

The workflow for an AcuSolve-EDEM bidirectional coupling simulation is shown below.

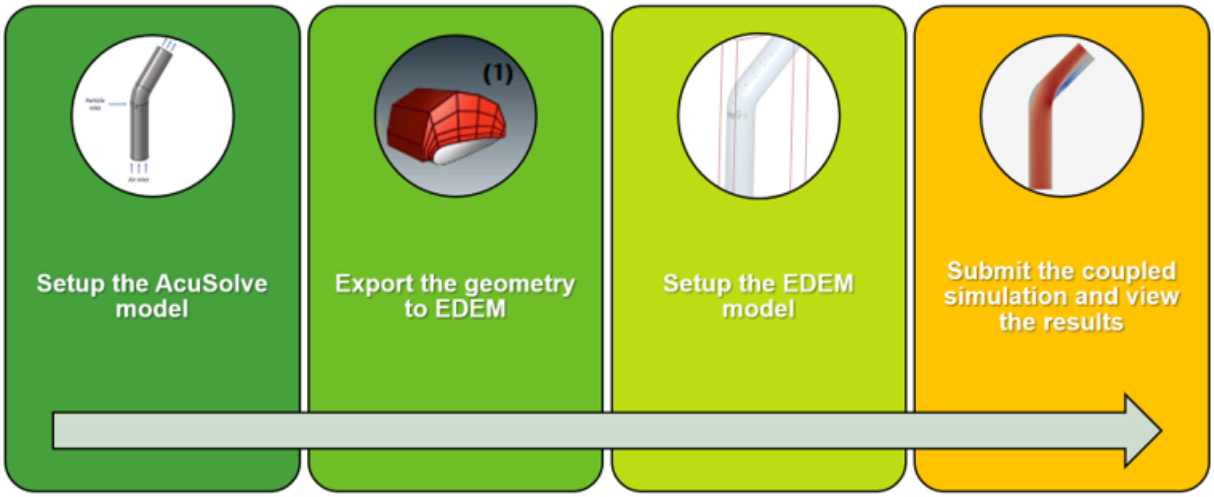


Figure 1466:

Accordingly, the tutorial consists of two parts:

1. AcuSolve setup and geometry export
2. EDEM setup and simulation

The AcuSolve model will be set up using HyperMesh CFD. Once the AcuSolve setup is complete, the EDEM deck with the geometry will be exported from HyperMesh CFD. This input deck will be opened in EDEM and will be used to complete the EDEM setup. Once the EDEM deck is set up, you will launch the coupled simulation.

Two different bulk materials used in the EDEM simulation and their properties are listed below:

Name	**Density (kg/m^3)**	**Particle radius (m)**	**Average mass of individual particle (kg)**	**Rate of generation (particles per sec)**
Heavy particle	900	0.03	0.1	100
Light particle	100	0.03	0.01	100

The thermal material properties used for the particles are listed in the table below:

Name	**Thermal conductivity (W/m-k)**	**Specific heat capacity (J/kg-K)**	**Temperature (K)**
Particles	1.4	840	363 (initial)
Walls	30	-	293

Part 1 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1467:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6103_windshifter_heat.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Save the database as `windshifter_heat` in the same directory as the other input files.

   This will be the working directory and all the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1468:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

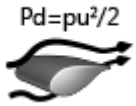


Figure 1469:

The **Setup** dialog opens.

2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the Material Library dialog, select **EDEM 2 Way Multiphase**, switch to the **My Material** tab, then click + to add a new material model.
6. In the microdialog, click **EDEM Bidirectional Material** in the top-left corner and change the name to `Air-Particle`.
7. Set the Carrier field to **Air** (if not set already).
8. Set the drag model and the drag coefficients as shown in the figure below.

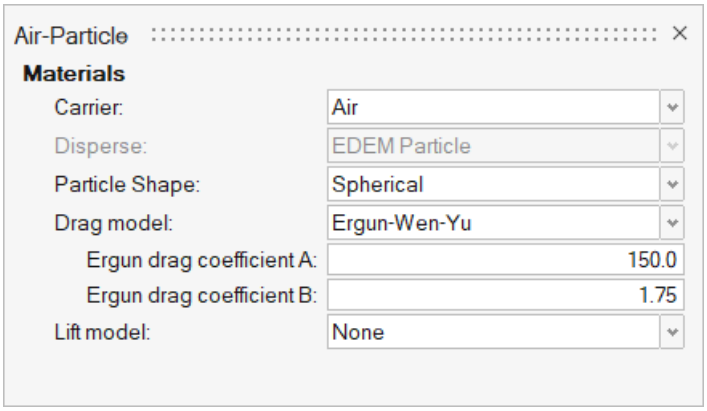


Figure 1470:

9. Close the material model microdialog and then close the Material Library dialog.
10. In the Setup dialog, set the Eulerian Material to **Air-Particle**.
11. Set Time step size and Final time to `0.001` and `1`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
12. Verify that Gravity is set to `0, 0, -9.81`.
13. Activate the **Heat transfer** checkbox.

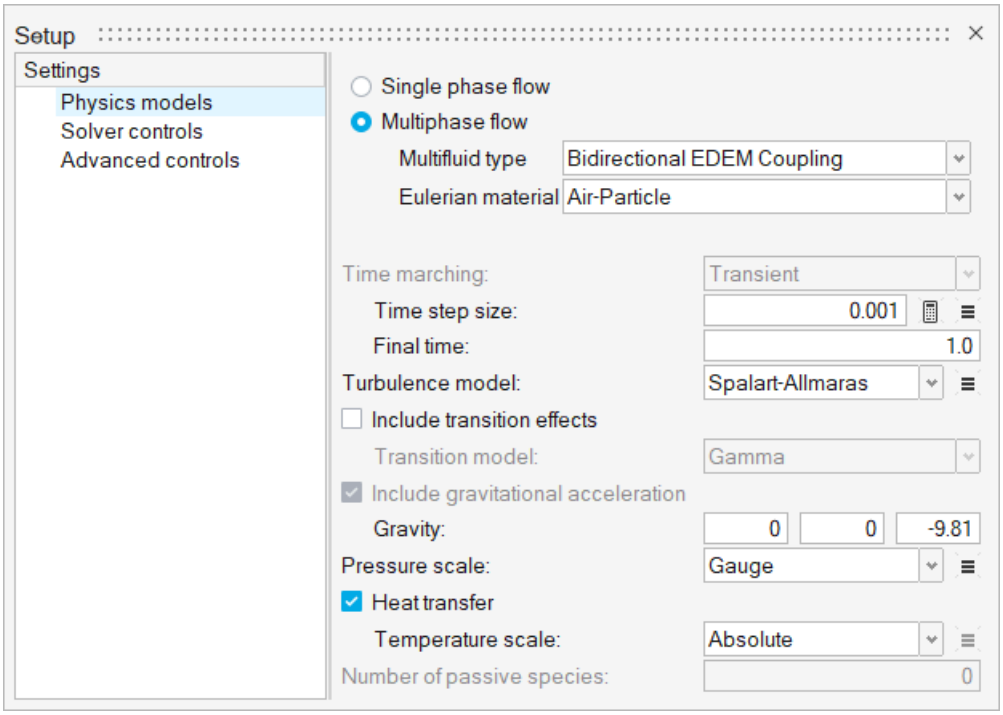


Figure 1471:

14. Click the **Solver controls** setting and set the Minimum and Maximum stagger iterations to `2` and `4`, respectively.

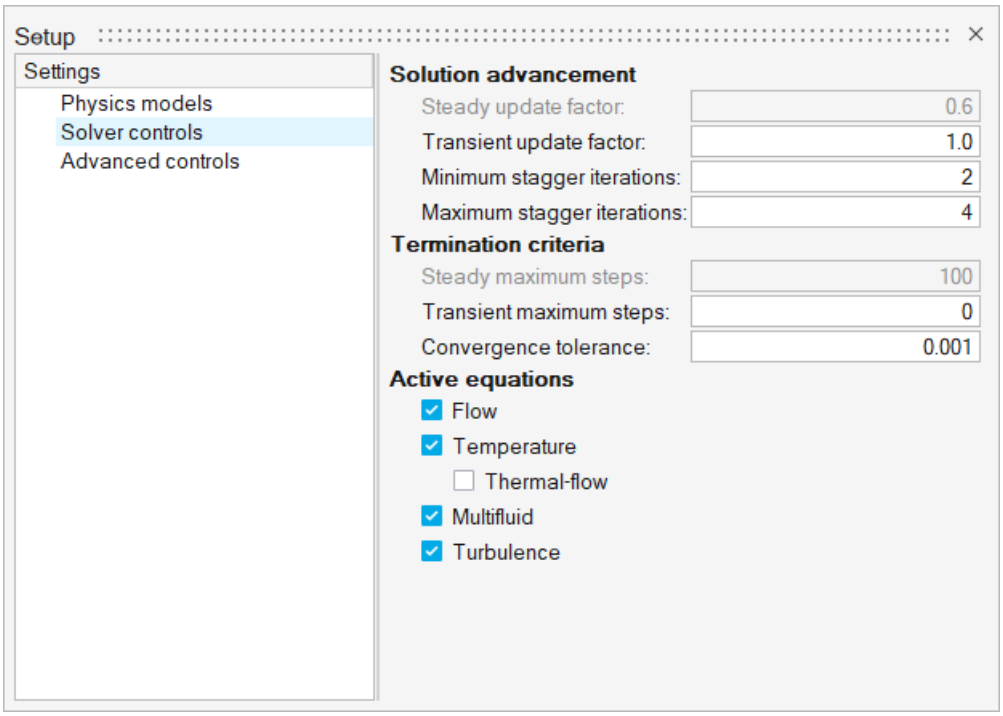


Figure 1472:

15. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1473:

2. Verify that Air-Particle has been assigned as the material.

3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1474:

2. Click on the inlet face highlighted in the figure below. In the microdialog, enter a value of `20` m/s for the Average velocity, `293` K for Temperature, and `1.0` for the Carrier fluid volume fraction.

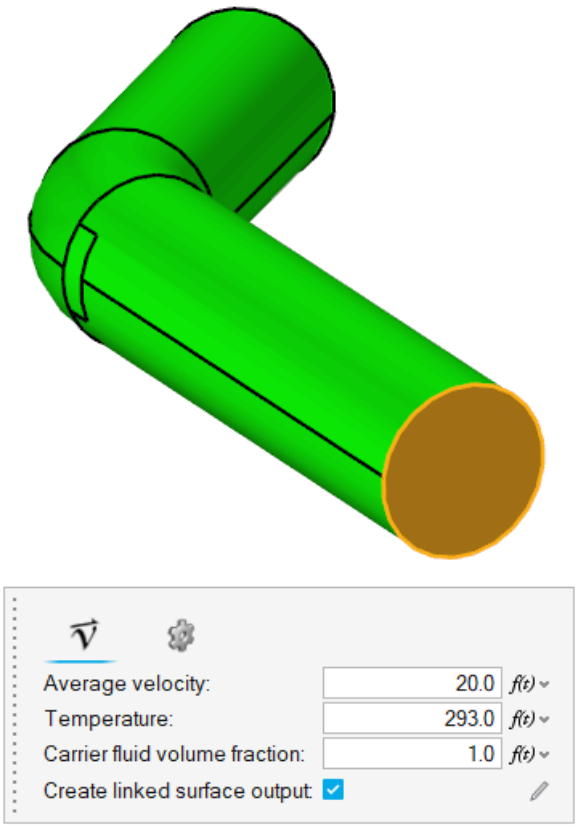


Figure 1475:

3. On the guide bar, click ✓ to execute the command and exit the tool.
4. Click the **Outlet** tool.

Figure 1476:

5. Select the face highlighted in the figure below then click ✓ on the guide bar.

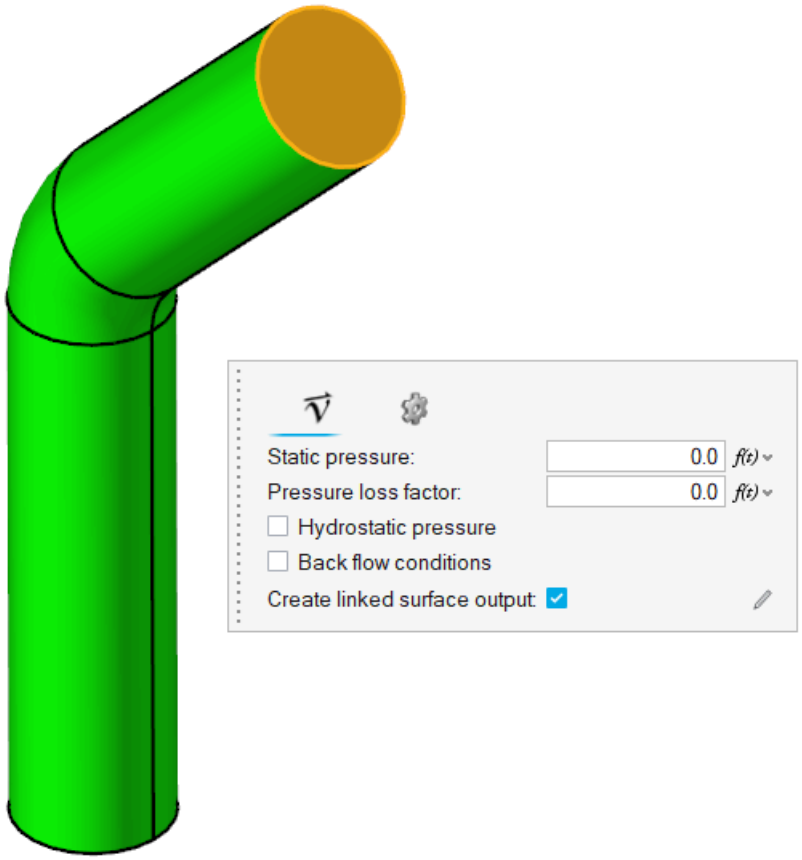


Figure 1477:

6. Click the **No Slip** tool.

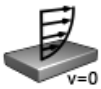


Figure 1478:

7. Select all the faces except the inlet and outlet faces.
8. In the microdialog, click the **Temperature** tab and set the parameters as shown in the figure below.

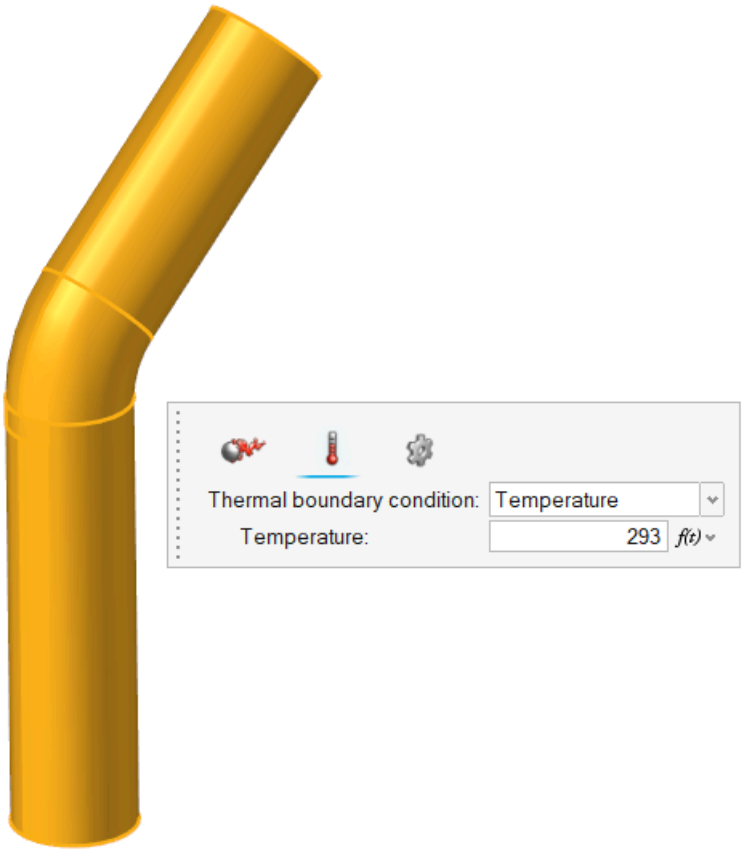


Figure 1479:

9. On the guide bar, click ▸ to execute the command and remain in the tool.

10. Select the face highlighted in the figure below. In the microdialog, click the **Temperature** tab and set the parameters as shown.

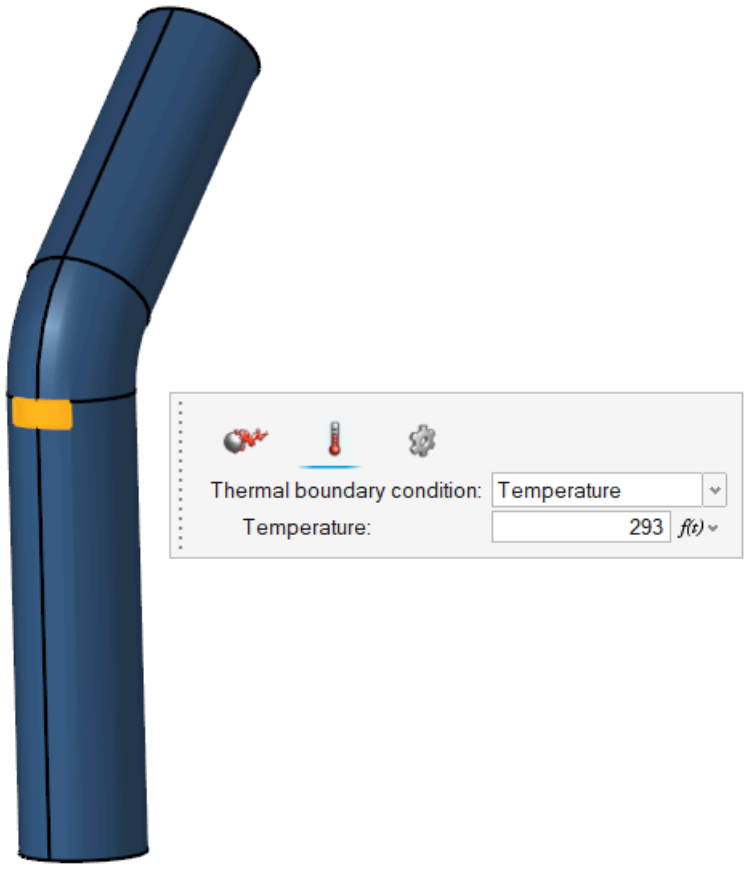


Figure 1480:

11. Click ▸ on the guide bar.
12. In the Boundaries legend, right-click on **Wall 1** and rename it to `Particle Inlet`.

    You will use this surface as a reference geometry to create a particle factory in EDEM. Hence, you are placing this surface in a different geometry group.
13. Click ✓ on the guide bar.
14. Save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

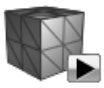


Figure 1481:

The **Meshing Operations** dialog opens.

2. Change the Average element size to `0.3`.

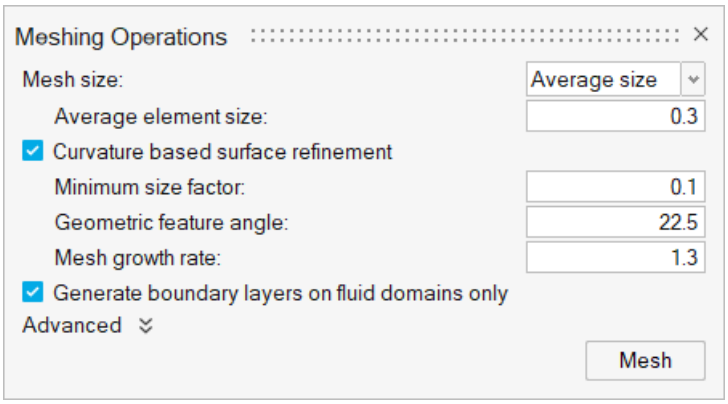


Figure 1482:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

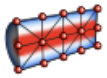


Figure 1483:

   The **Field Output** dialog opens.
2. Check the box for **Write Initial Conditions**.
3. Set the Time step interval to `10`.

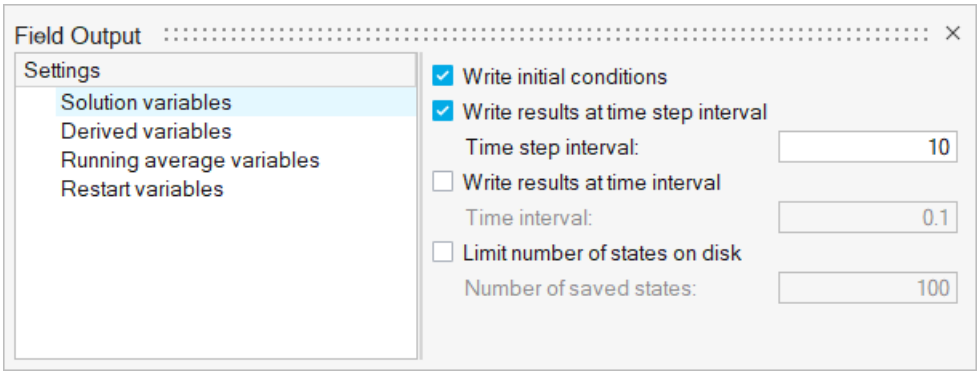


Figure 1484:

Export the Solver Deck

1. From the menu bar, go to **File** > **Export** > **Solver Deck**.
2. Name the file `windshifter_heat` and make sure that **AcuSolve (*.inp)** is selected as the file type.
3. Click **Save**.

The AcuSolve input files and the EDEM input deck with the geometry sections are created. You will use this EDEM deck to set up the DEM simulation deck.

Part 2 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

As mentioned earlier, when the AcuSolve simulation was launched, HyperMesh CFD created a set of EDEM files in the problem directory. You will open that EDEM input deck and setup the DEM simulation.

1. In the Creator tab in EDEM, go to **File** > **Open**.
2. In the dialog, browse to the AcuSolve problem directory and open the `windshifter_heat.dem` file located in the EDEM folder.
   The geometry is loaded.
3. Click the **Environment** tab under the Creator Tree, uncheck **Auto Update from Geometry**, and then check the box again to fit the geometry within the boundary.

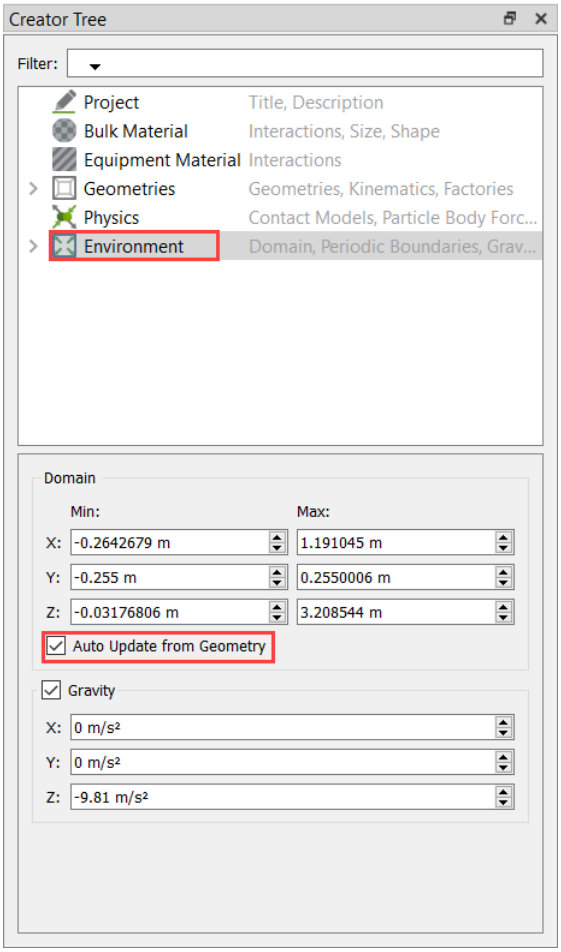

Creator Tree
Filter:
Project
Title, Description
Bulk Material
Interactions, Size, Shape
Equipment Material
Interactions
Geometries
Geometries, Kinematics, Factories
Physics
Contact Models, Particle Body Forc...
Environment
Domain, Periodic Boundaries, Grav...
Domain
Min:
Max:
X: -0.2642679 m
1.191045 m
Y: -0.255 m
0.2550006 m
Z: -0.03176806 m
3.208544 m
Auto Update from Geometry
Gravity
X: 0 m/s²
Y: 0 m/s²
Z: -9.81 m/s²

Figure 1485:

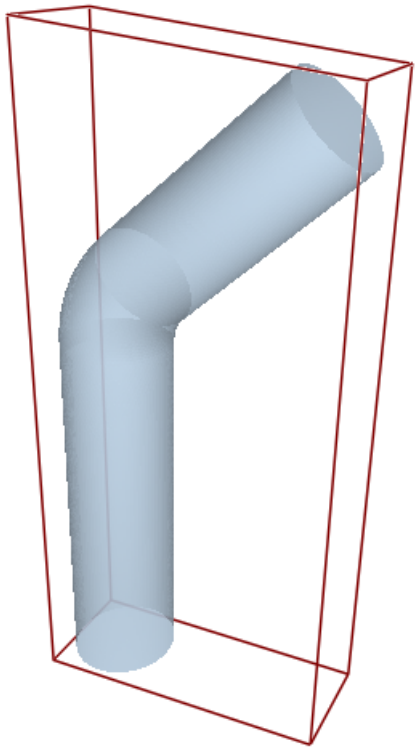


Figure 1486:

Define the Bulk Materials and Equipment Material

In this step, you will define the material models for the heavy and light bulk material and the equipment material.

1. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.
2. Rename the material to `Heavy`.
3. In the Creator Tree, set the Solids Density property to `900` kg/m^3.
   You will use the default values for other properties for this tutorial.

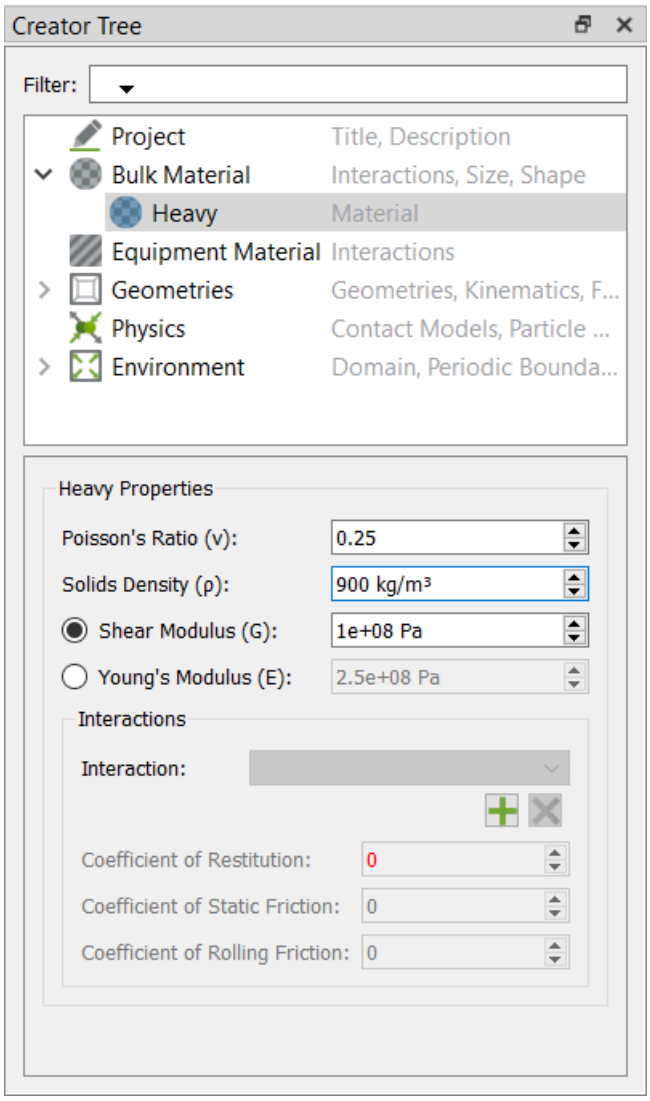


Figure 1487:

4. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, click **OK**.
5. In the Creator Tree, right click **Heavy** and select **Add Shape from Library** > **Dual Sphere Shape**.
6. Rename the particle to `Heavy particle`.
7. Under Heavy particle, click **Properties**.
8. In the Heavy particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

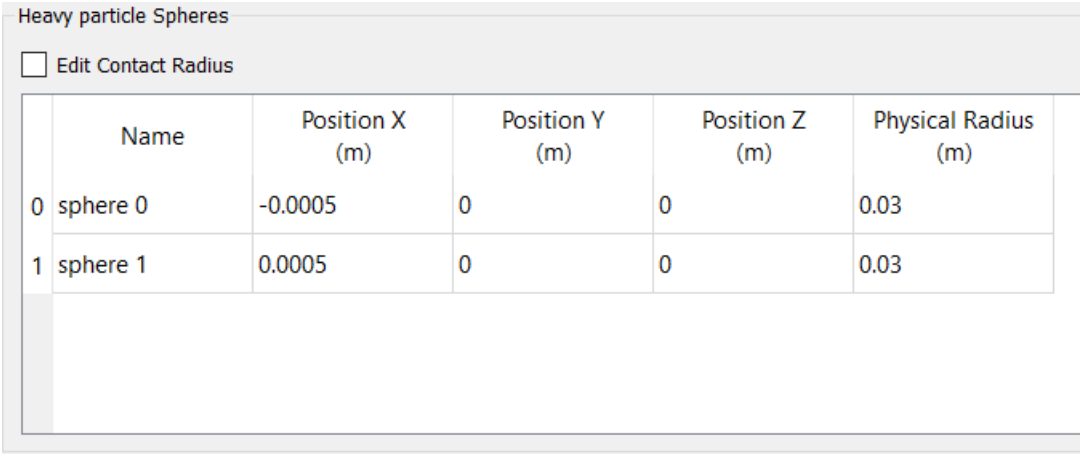


Figure 1488:

9. In the Creator Tree, click **Calculate Properties**.

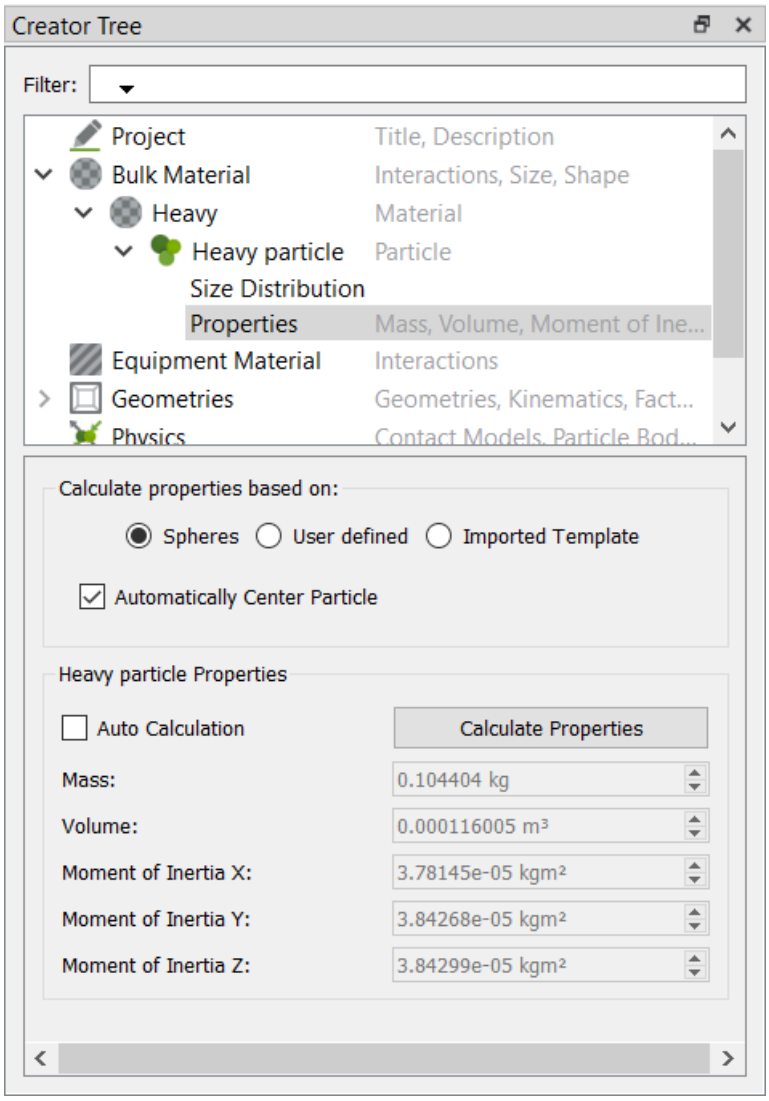


Figure 1489:

10. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.

11. Rename the material to `Light`.

12. In the Creator Tree, set the Solids Density property to `100` kg/m^3.

You will use the default values for other properties for this tutorial.

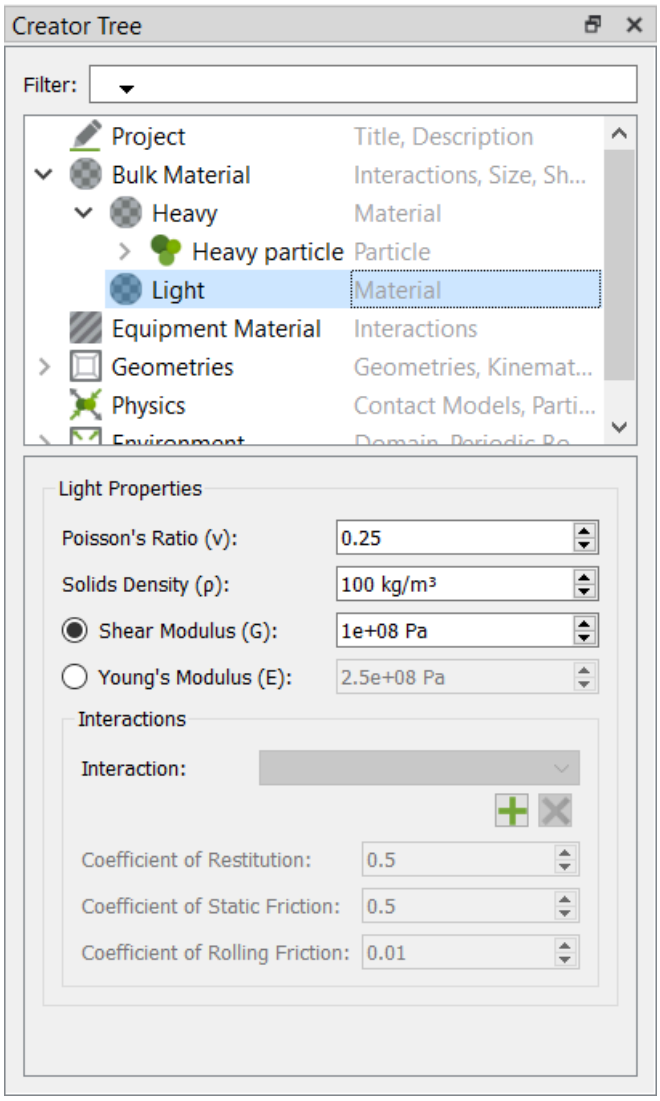


Figure 1490:

13. Click + below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.
14. Click + again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.
15. In the Creator Tree, right click on **Light** and select **Add Shape from Library** > **Dual Sphere Shape**.
16. Rename the particle to `Light particle`.
17. Under Light particle, click **Properties**.
18. In the Light particle Spheres panel, set the Physical Radius of both the spheres to `0.03` m and press **Enter**.

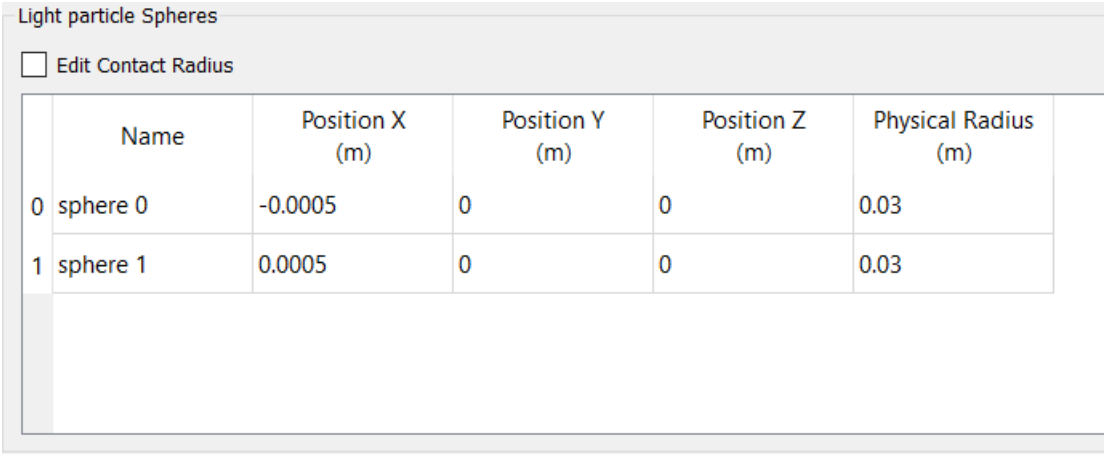


Figure 1491:

19. In the Creator Tree, click **Calculate Properties**.

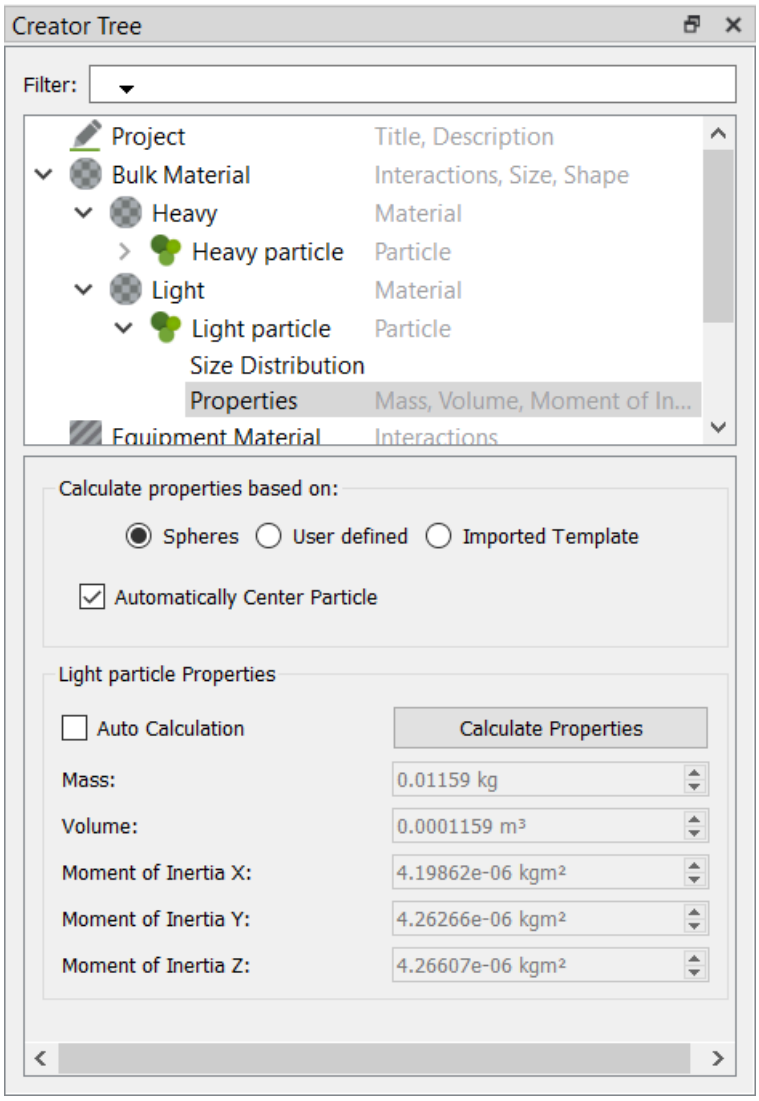


Figure 1492:

20. In the Creator tree, right-click **Equipment Material** and select **Add Equipment Material**. Rename it to `Steel`.

21. Set the Density to `7800` kg/m^3.

22. Click below Interaction to define the interaction properties for collisions among the heavy particles. In the dialog, select **Heavy** and then click **OK**.

23. Click again to define the interaction properties for collisions among the light particles. In the dialog, select **Light** and then click **OK**.

24. Save the model.

Create the Particle Factory

1. Expand **Geometries** under the Creator Tree tab. Next, right-click on the **Particle Inlet** surface group and select **Copy Geometry** > **Single Copy**.
2. Rename the new geometry section to `Particle_factory`.
3. Right-click on the **Wall** geometry section in the Creator Tree and select **Merge Geometry(s)**.
4. In the Merge Geometry dialog, select the **Particle Inlet** then click **OK**.
5. Click on the **Wall** geometry section, set the Type to **Physical**, and the Material to **Steel** (if not set already).

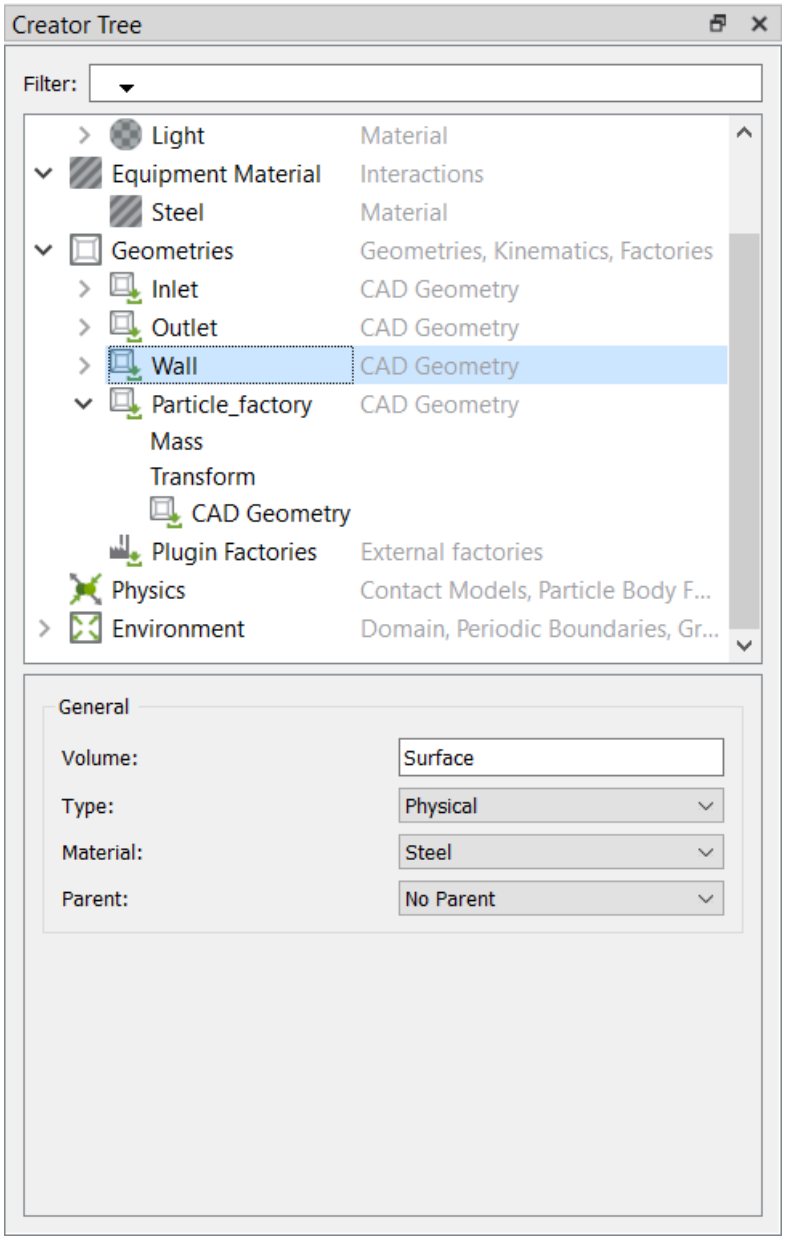


Figure 1493:

6. In the Creator Tree, click the **Inlet** section and change the Type to **Virtual**.

7. Similarly, change the Type to **Virtual** for the Outlet and Particle_factory sections.
8. Under Particle_factory, click **Transform**. Set the X-Position to `0.035` m

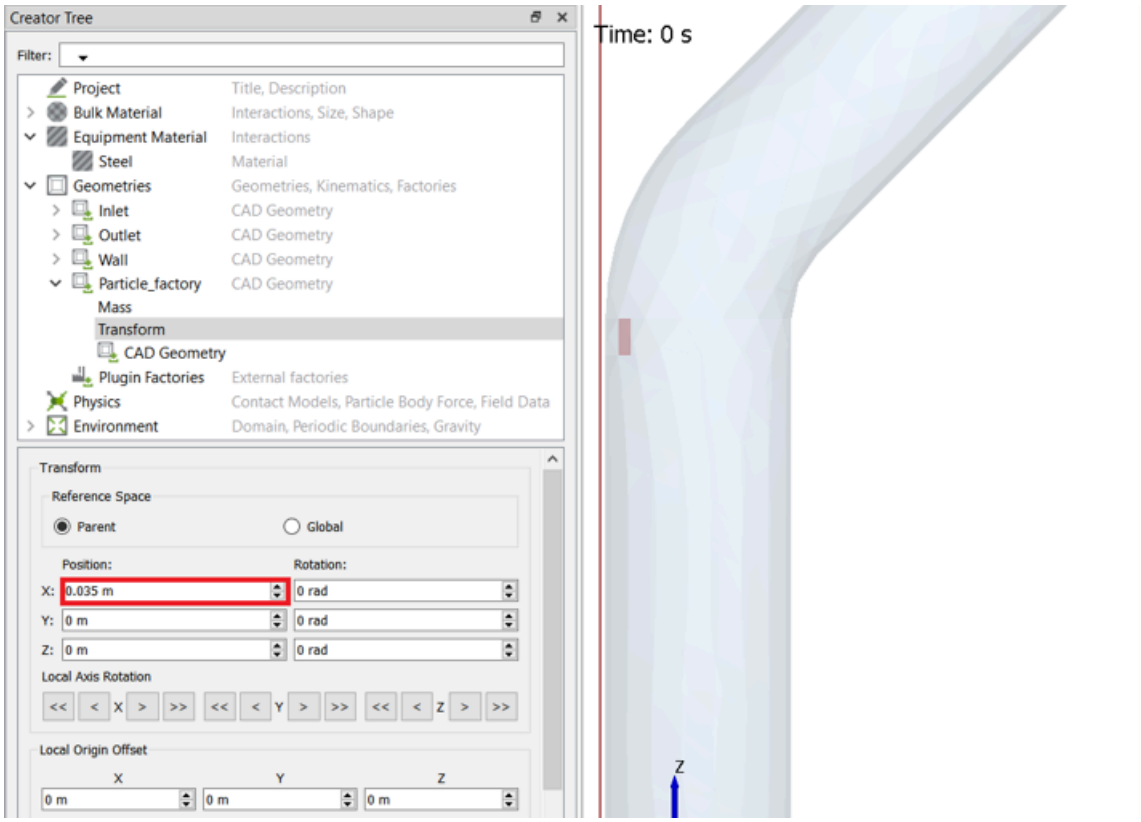


Figure 1494:

> **Note:** Set the Opacity value to 0.2 to see the transformed surface location inside the pipe geometry.
>
> This is done to make sure that the particles are generated inside the fluid domain.

Define the Heat Transfer Paramaters

1. In the Creator Tree, click on **Physics**.
2. Set the Interaction drop down to **Particle to Particle** and click **Edit Contact Chain**.
3. In the dialog, activate **Heat Conduction** and click **OK**.

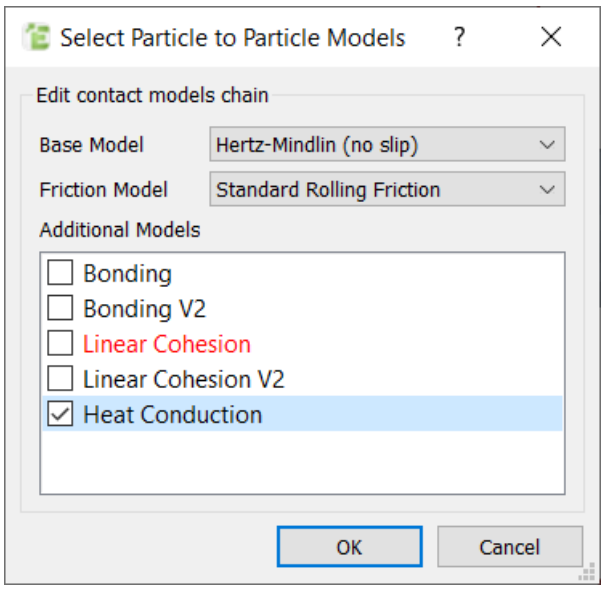


Figure 1495:

4. In the Physics tab, click **Heat Conduction** then click ⚙.

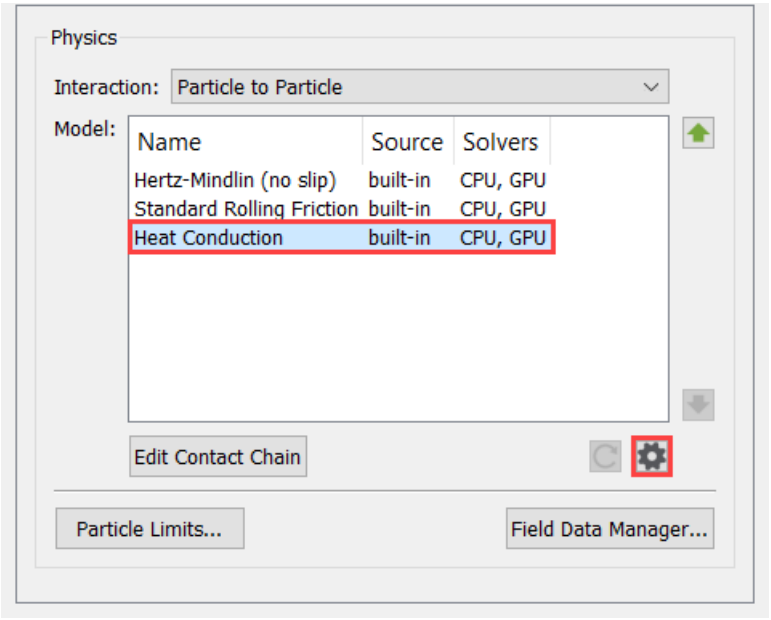


Figure 1496:

5. In the dialog, set the Thermal conductivity of both Heavy and Light particles to `1.4` W/mK then click **OK**.

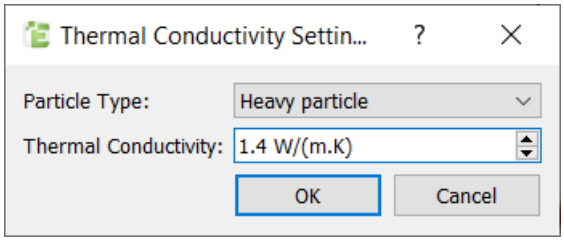


Figure 1497:

6. In the Physics tab, change the Interaction to **Particle to Geometry** and click **Edit Contact Chain**.
7. In the dialog, activate **Heat Conduction** and click **OK**.
8. In the Physics tab, click **Heat Conduction** then click .
9. In the dialog, set the thermal conductivity of the particles to `1.4` W/mK and the conductivity and temperature of the wall to `30` W/mK and `293` K, respectively. Click **OK**.

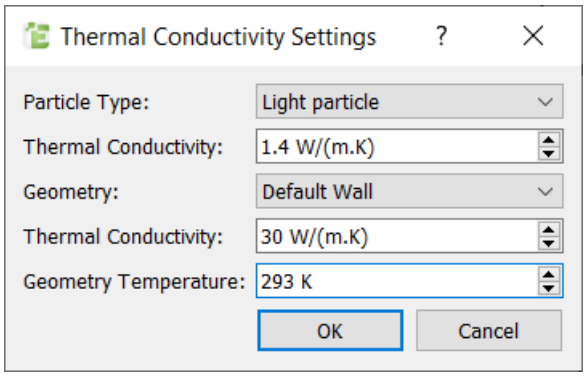


Figure 1498:

10. In the Physics tab, change the interaction to **Particle Body Force**.
11. Click **Edit Contact Chain**.
12. In the dialog, activate **Temperature Update** then click **OK**.
13. In the Physics tab, click **Temperature Update** then click .
14. In the dialog, set the specific heat capacity for both the particle types to `840` J/kgK then click **OK**.

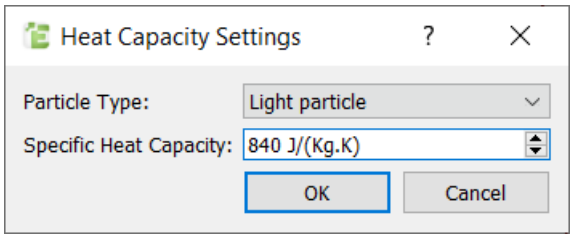


Figure 1499:

15. Save the model.

Define the Particle Factory

Now that the bulk material, geometry sections, and equipment materials are defined, you need to create a particle factory to generate the particles. You will create one factory for each bulk material.

1. In the Creator Tree, right-click on **Particle_factory** and select **Add Factory**.
2. Rename the new factory to `Heavy factory`.
3. Set the particle generation parameters as shown in the figure below.

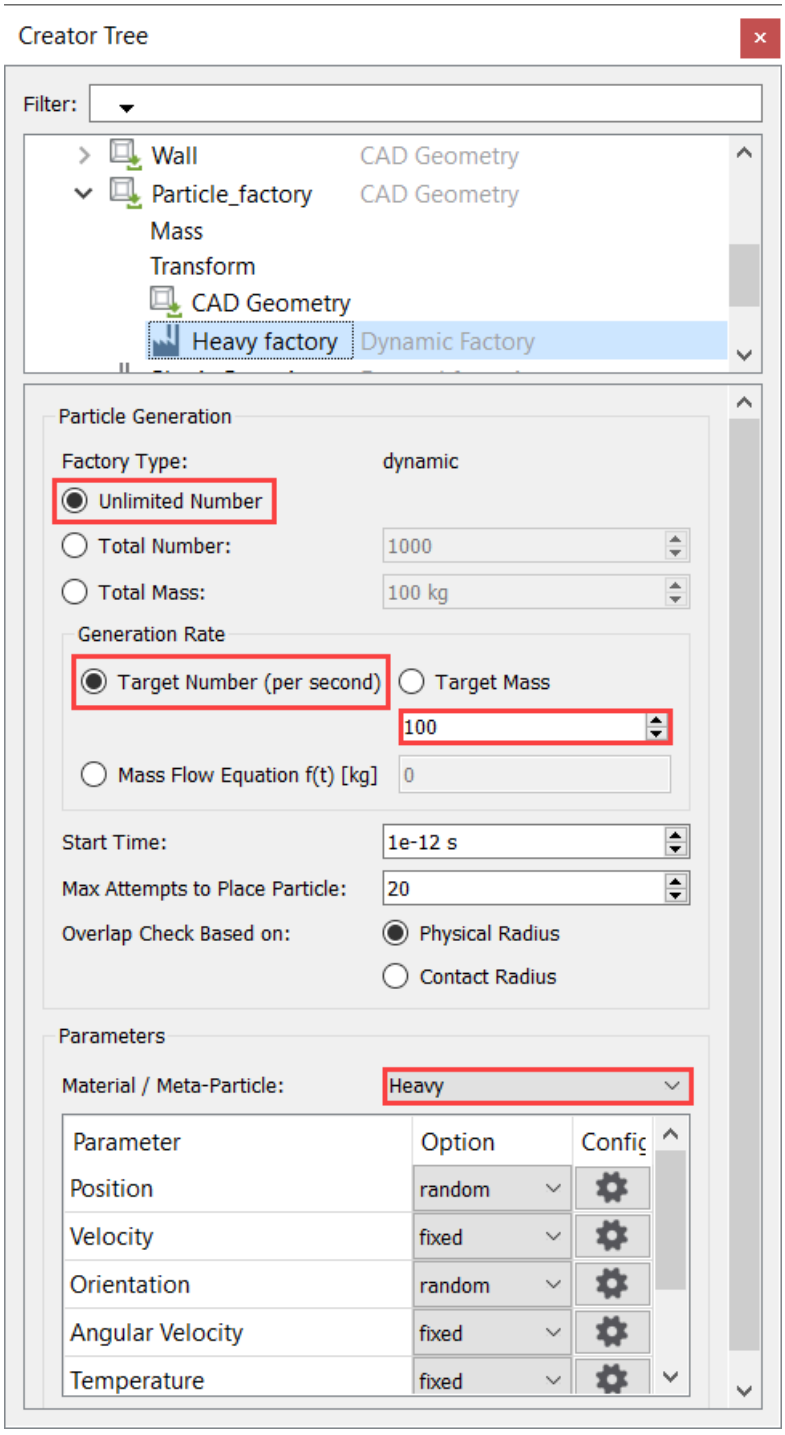


Figure 1500:

4. Click ⚙ besides Velocity, set the X-velocity to `1` m/s, then click **OK**.
5. Click ⚙ besides Temperature, set the value to `363` K, then click **OK**.

> **Note:** The fields for Temperature and Heat Flux will not be visible unless heat transfer properties are defined.

6. Repeat steps 1-5 to create another factory named `Light factory` using the same parameters but with **Light** as the Material.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).
   When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.
3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and de-activate the **Auto Time Step** checkbox.
3. Set the Fixed Time Step to `2.5e-5` s.

> **Note:** Generally, a value of 20-40% of the Rayleigh Time Step is recommended as the time step size to ensure stability of the DEM simulation.

4. Set the Total Time to `1` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `4` R min.
   Generally, a value in the range of 3-6 R_{min} is recommended as the optimum cell size.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

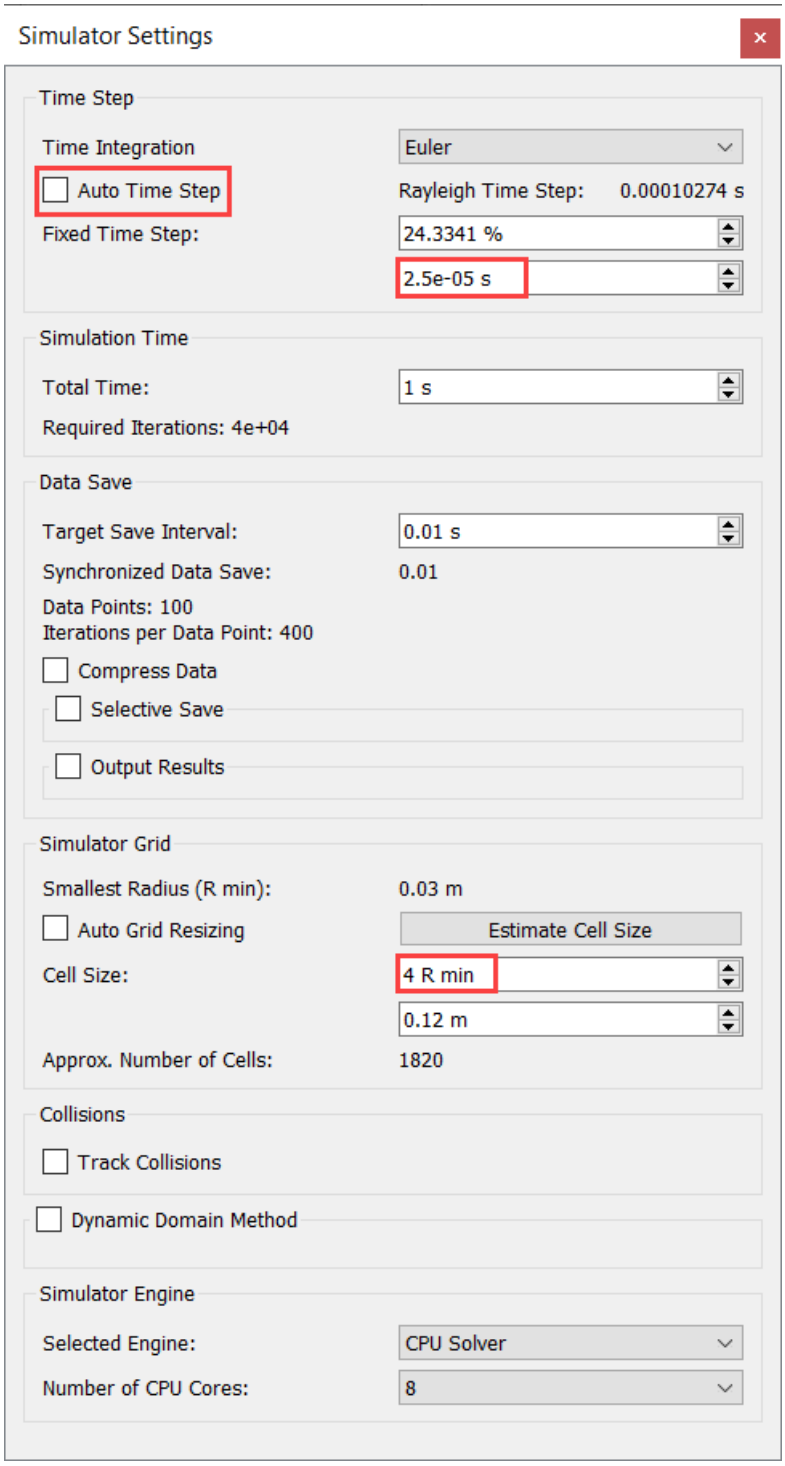


Figure 1501:

7. Once the simulation settings have been defined, save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1502:

Once the Coupling server is activated, the icon changes.

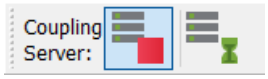


Figure 1503:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1504:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Optional: Set the number of processors to **4** or **8** based on availability.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
7. Set the Temperature to `293` K.
8. Click **Run** to launch AcuSolve.

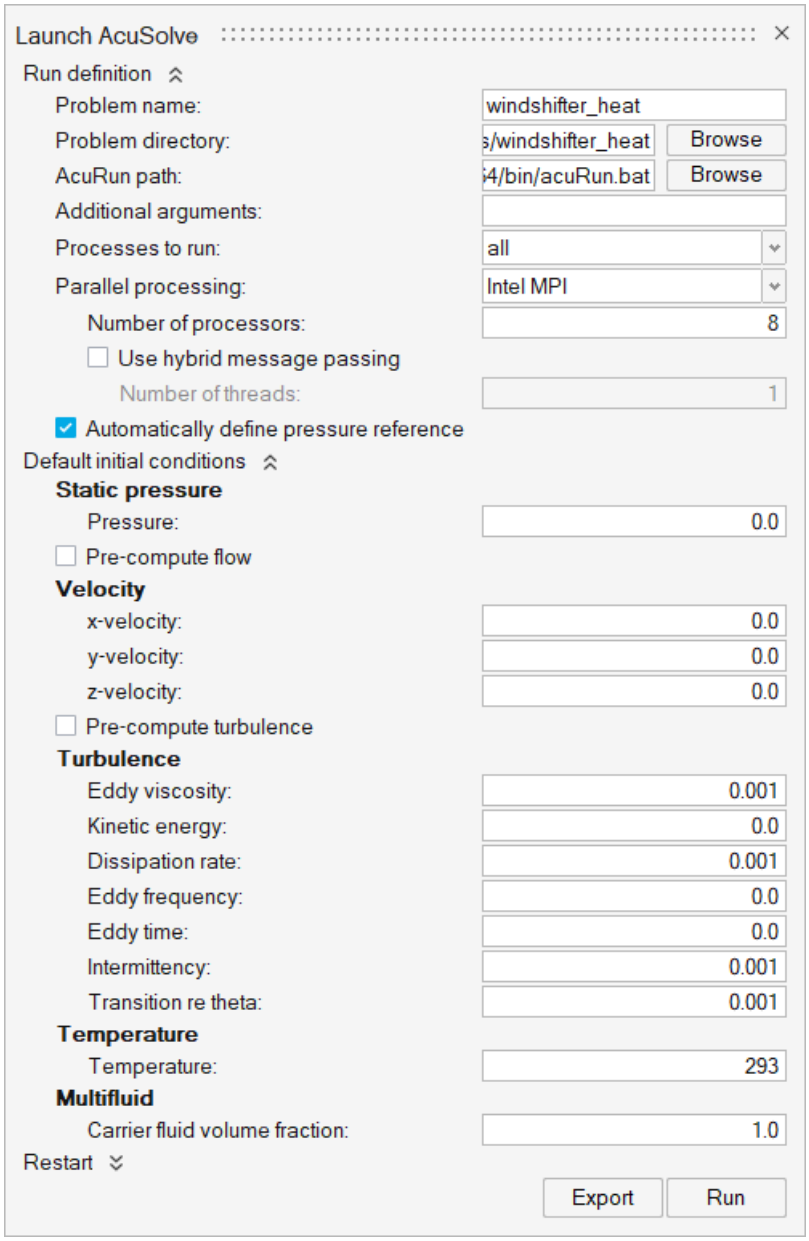


Figure 1505:

Once the AcuSolve run is launched, the **Run Status** dialog opens.

9. In the dialog, right-click on the AcuSolve run and select **View log file**.
   If the coupling with EDEM is successful, that information is printed in the log file.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 7.190000e-001 2.360240e+002 Sec
acuSolve: Input Memory Usage        = 3.200000e+001 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Temperature in coupling client registered
acuSolve: Interface thickness is 1.449514e-001
acuSolve: Init. CPU/Elapse time     = 4.700000e-002 5.800000e-002 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 1.000000e-003 ; time= 0.000000e+000
```

Figure 1506:

Once the simulation is complete, the summary of the run time is printed at the end of the log file.

```
acuSolve: ---------- End Time Step ----------
acuSolve: ---------- Offset Values ----------
acuSolve:         Pressure Offset = 1.013250e+005
acuSolve:       Temperature Offset = 0.000000e+000
acuSolve: ------------------------------------------------------------
acuSolve: Input      CPU/Elapse time= 7.660000e-001 2.360820e+002 Sec
acuSolve: Output     CPU/Elapse time= 2.729670e+002 2.763770e+002 Sec
acuSolve: Exchange   CPU/Elapse time= 5.311000e+000 3.868000e+000 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =       3308       3308
acuSolve:   Elem.Form CPU/Elapse time= 3.385310e+002 3.422800e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.524220e+002 1.543620e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =       3308       3308
acuSolve:   Elem.Form CPU/Elapse time= 2.477520e+002 2.508610e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.043540e+002 1.048570e+002 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =       3308       3308
acuSolve:   Elem.Form CPU/Elapse time= 2.607210e+002 2.640140e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.155320e+002 1.166280e+002 Sec
acuSolve: Edem stagger "edem":
acuSolve:   No of external code calls=       1000
acuSolve:   EDEM ext. code CPU/Elapse time= 4.201400e+001 2.760960e+002 Sec
acuSolve: Field-Velocity stagger "field_velocity: EDEM Particle":
acuSolve:   No of res/lhs formations =       1000       1000
acuSolve:   Elem.Form CPU/Elapse time= 1.069570e+002 1.082160e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 2.654500e+001 2.677700e+001 Sec
acuSolve: Field-Velocity stagger "field_velocity: Air":
acuSolve:   No of res/lhs formations =       1000          0
acuSolve:   Elem.Form CPU/Elapse time= 7.700000e-002 5.500000e-002 Sec
acuSolve:   Solution  CPU/Elapse time= 2.447600e+001 2.462800e+001 Sec
acuSolve: Fields stagger "EDEM Particle":
acuSolve:   No of res/lhs formations =       1000       1000
acuSolve:   Elem.Form CPU/Elapse time= 7.111100e+001 7.210900e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 3.029300e+001 3.031900e+001 Sec
acuSolve: Fields stagger "Air":
acuSolve:   No of res/lhs formations =       1000          0
acuSolve:   Elem.Form CPU/Elapse time= 4.800000e-002 5.000000e-002 Sec
acuSolve:   Solution  CPU/Elapse time= 3.014600e+001 3.049700e+001 Sec
acuSolve: LnSlv. (host)  Elapse time= 7.152100e+001 Sec
acuSolve: Total CPU/Elapse time     = 2.631969e+003 3.132995e+003 Sec
acuSolve: Total Memory Usage        = 3.900000e+001 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
```

Figure 1507:

Analyze the Results

AcuSolve Post-Processing

1. In HyperMesh CFD, right-click on the AcuSolve run in the **Run Status** dialog and select **Visualize results**.

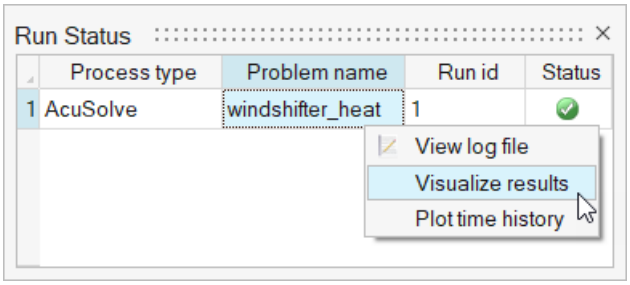


Figure 1508:

The results are loaded in the Post ribbon.

2. Click the **Slice Planes** tool.

Figure 1509:

3. Select the x-z plane as highlighted in the figure below.

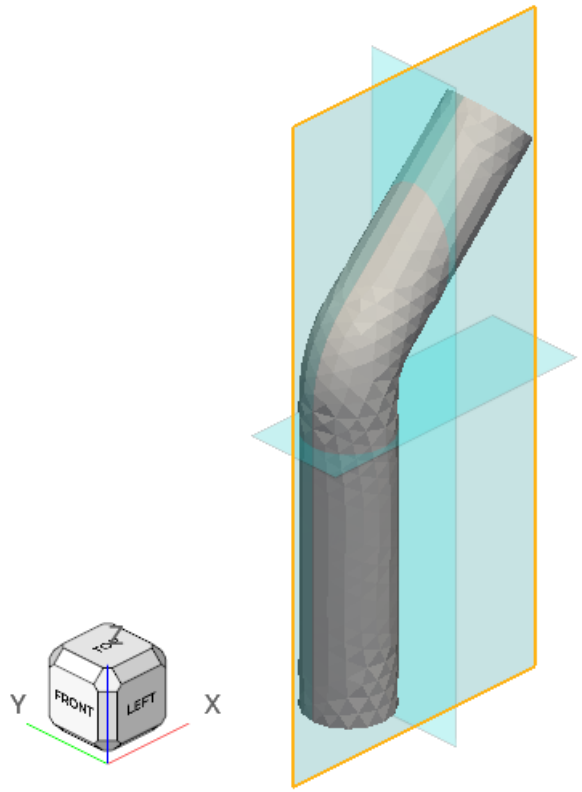


Figure 1510:

4. In the slice plane microdialog, click to create the slice plane.
5. In the display properties microdialog, set the display to **temperature** and activate the **Legend** toggle.
6. Change the bounds of the legend to `293` and `363`.
7. Click and set the Colormap name to **Rainbow Uniform**.

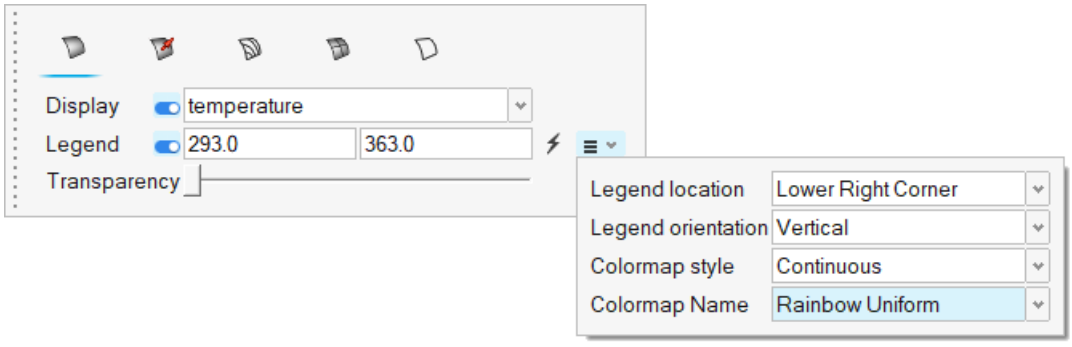


Figure 1511:

8. Click on the guide bar.
9. In the Post Browser, turn off the visibility of **Flow Boundaries** by clicking on its icon.

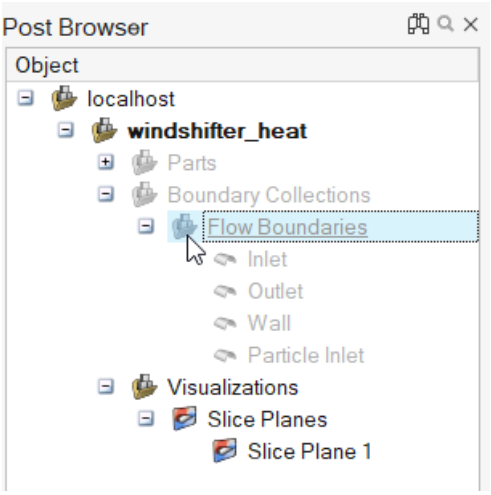


Figure 1512:

10. Select the **Left** face on the view cube to align the model to the x-z plane.

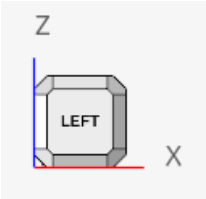


Figure 1513:

11. Click ▶ on the animation toolbar to view the animation of the temperature contour.

Figure 1514:

12. To plot the volume fraction edem particle variable, right-click on **Slice Plane 1** in the Post Browser and select **Edit**.

13. In the display properties microdialog, change the display variable from temperature to **volume fraction edem particle** and the legend bounds to `0` and `0.2190`.

14. Click ✓ on the guide bar.

15. Click ▶ to start the animation.

Figure 1515:

EDEM Post-Processing

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.

2. In the Analyst Tree, expand **Display** > **Geometries** and then click **Wall**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

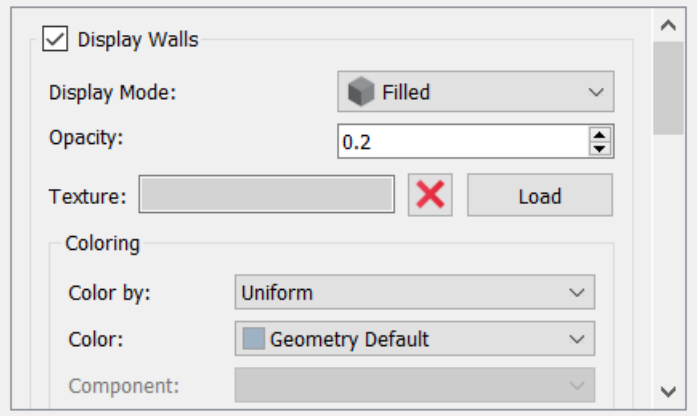


Figure 1516:

4. In the Analyst Tree, click **Particles**.
5. Set the coloring to **Temperature**.
6. Activate the **Auto Update** checkboxes for both the Min and Max Value.
7. Activate the **Show Legend** checkbox.
8. Click **Apply All**.

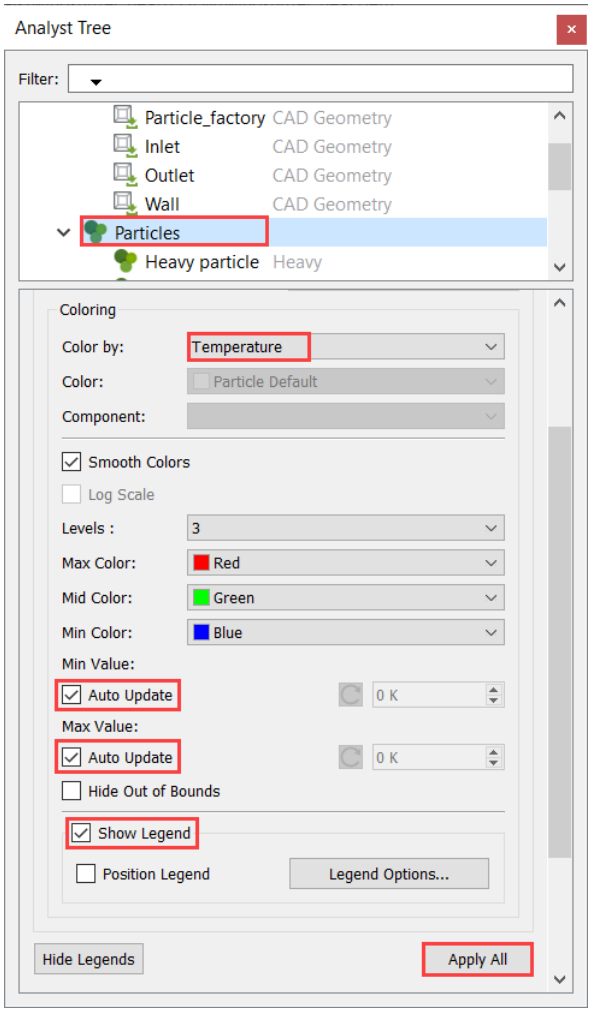


Figure 1517:

9. On the menu bar, set the time to `0` by clicking:

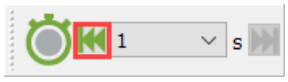


Figure 1518:

10. Set the View plane to **+ Y**.

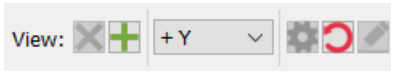


Figure 1519:

11. In the Viewer window, set the Playback Speed to `0.1x` and then click to play the particle flow animation.

    Figure 1520:

    Observe that particles which are at a higher temperature at the time generation become colder as they exchange energy with the fluid phase.

Summary

In this tutorial, you learned how to set up and run a basic AcuSolve-EDEM bidirectional (two-way) coupling problem with heat transfer. In the first part, set up the AcuSolve model in HyperMesh CFD and exported the geometry. Next, you imported the EDEM input files created by HyperMesh CFD and set up the EDEM model. You learned how to set up the thermal properties for the particles as well as their interaction with other particles and equipment. Once the coupled simulation was completed, you learned how to create animations in both HyperMesh CFD Post and EDEM Analyst.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/ubvu05wsj2

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6104 AcuSolve - EDEM Bidirectional Coupling with Non-Spherical Particles

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

This tutorial introduces you to the workflow for setting up and running a basic bidirectional coupling (two-way) simulation with non-spherical particles using AcuSolve and EDEM. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6104_sieve.hm`

`sieveEDEM.zip`

Problem Description

The problem to be solved is shown schematically in the figure below. The model consists of a simple sieve which oscillates at a frequency of 5 Hz and has an amplitude of 0.02 m. Particles are introduced into the system from the top at a velocity of 1 m/s. The particles fall through the openings in the sieve and as air is blown from the inlet at the bottom, the lighter bar-shaped particles get separated from the heavier spherical particles. An average velocity of 6 m/s is applied on the inlet face and the Arbitrary Lagrangian Eulerian approach is used for the mesh motion.

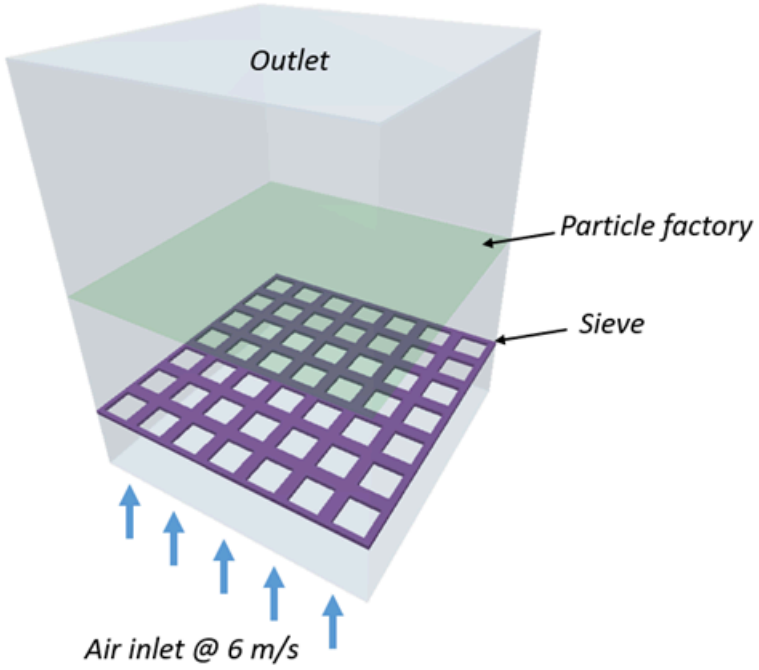


Figure 1521:

A sinusoidal motion along the z-axis is assigned to the sieve. The motion profile is shown below:

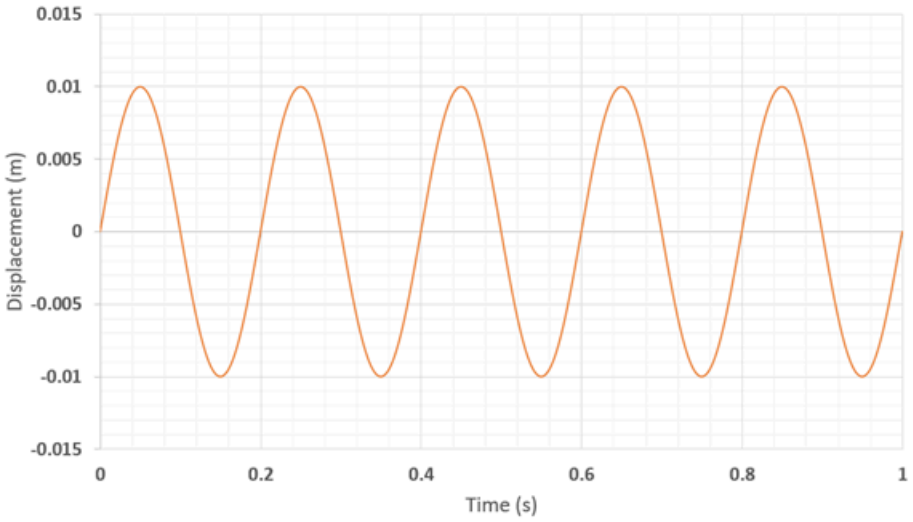


Figure 1522:

AcuSolve-EDEM bidirectional coupling is used to model the interaction between the fluid and particles. In this tutorial, non-spherical drag force models are used to accurately predict the drag forces on the particles by taking the shape of particles into consideration. The length scale used for factoring in the shape of the particles is the aspect ratio. Bar shaped particles with an aspect ratio of 2.75 and spherical

particles with aspect ratio 1 are used for this simulation. The Rong drag model is used in conjunction with the Holzer-Sommerfeld non-spherical drag coefficient model.

The particle properties used for this simulation are listed in the table below:

L - length, D - diameter

Particle Type	**Density (kg/m^3)**	**Size (m)**	**Number of particles**
Bar	200	0.022/0.008 (L/D)	100
Sphere	1500	0.008 (D)	200

There are two categories of non-spherical drag coefficient models supported in AcuSolve-EDEM bidirectional coupling:

1. Isometric – where orientation of the particle is not considered in the drag force calculation. This model is applicable for shapes such as rocks, certain beans and ore particles. The Haider Levenspiel drag model is available for such scenarios and it requires particle's sphericity and volume as inputs.
2. Non-spherical – the orientation of the particle is considered for the drag force calculation. This option is applicable for shapes such as disc, plate and cylinder where the orientation of the particle with respect to the flow velocity influences the magnitude and direction of the force. Two options are available for such cases: Ganser and Holzer-Sommerfeld models. Both models require aspect ratio and volume of particles as inputs.

Part 1 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

1. From the menu bar, click **File** > **Open**.
2. In the dialog, browse to your problem directory and open the `sieve.dem` file.
   The geometry is loaded.

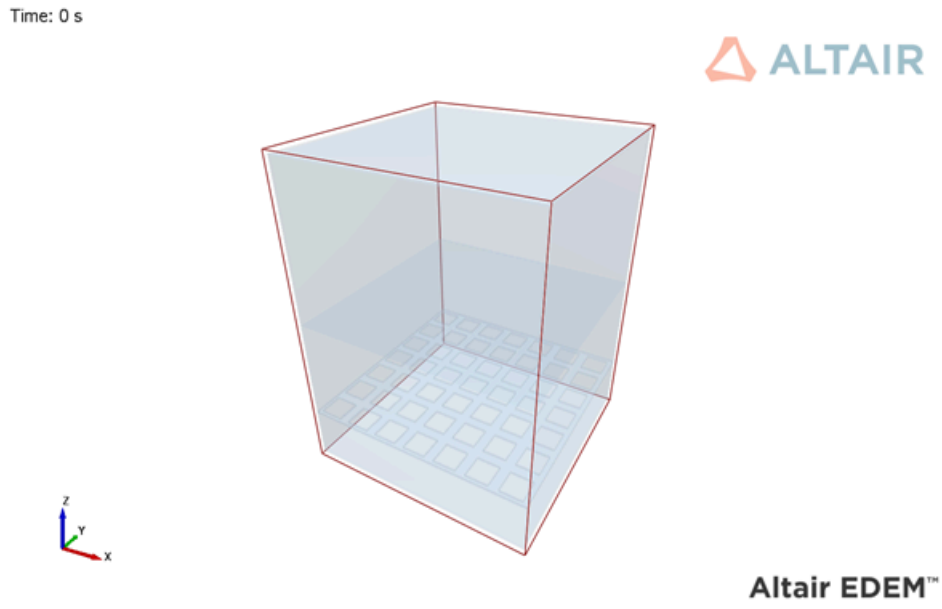


Figure 1523:

Define the Bulk Materials and Equipment Material

In this step, you will define the material models for the bar and sphere-shaped particles.

1. In the Creator Tree, right-click **Bulk Material** and select **Add Bulk Material**.
2. Rename the material to `bar`.
3. In the Creator Tree, set the Solids Density property to `200` kg/m^3 and the Shear Modulus to `1e7` Pa.
4. Click below Interaction to define the interaction properties for collisions among the bar shaped particles. In the dialog, click **OK**.
5. Set the interaction coefficients as shown in the figure below.

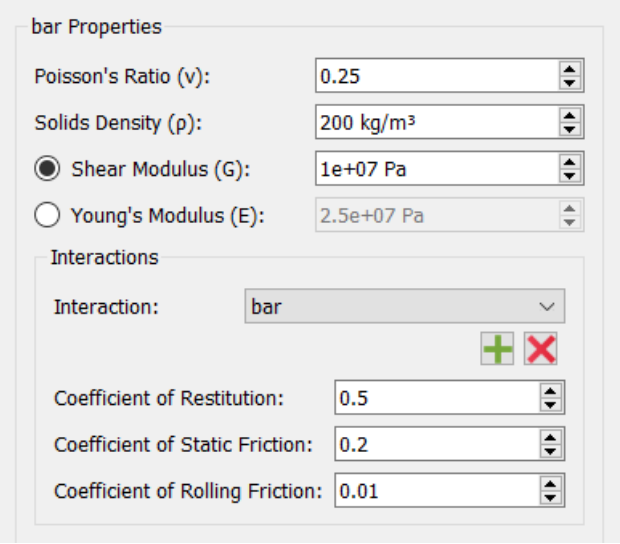
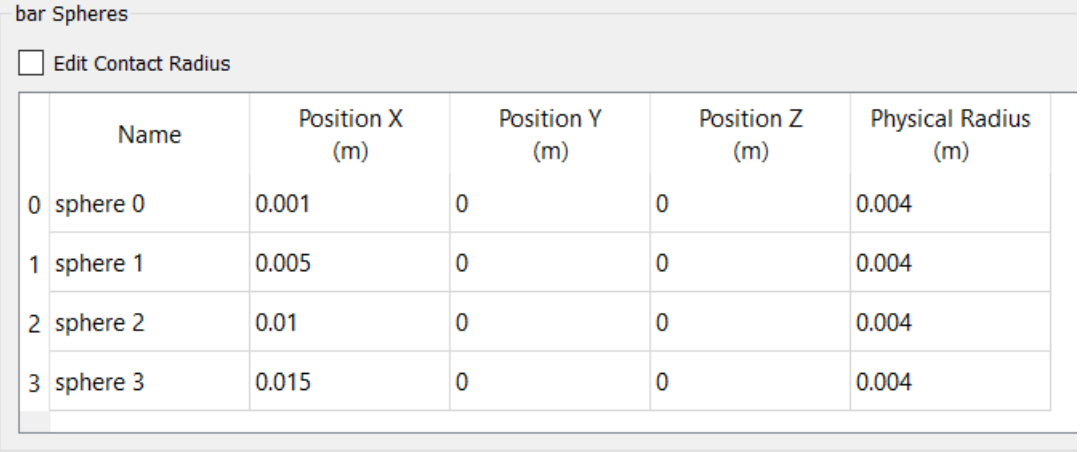


Figure 1524:

6. In the Creator Tree, right click **bar** and select **Add Shape from Library** > **Straight Four Sphere Shape**.
7. Rename the particle to `bar`.
8. Under bar, click **Properties**.
9. In the bar Spheres panel, set the Physical Radius and the XYZ positions as shown in the figure below.



	Name	Position X (m)	Position Y (m)	Position Z (m)	Physical Radius (m)
0	sphere 0	0.001	0	0	0.004
1	sphere 1	0.005	0	0	0.004
2	sphere 2	0.01	0	0	0.004
3	sphere 3	0.015	0	0	0.004

Figure 1525:

Note: When defining non-spherical shapes, you need to make sure that the principal axis of the particle is aligned with the x-axis, as shown in the figure below. The principal axis definition for some standard shapes is explained later in this tutorial.

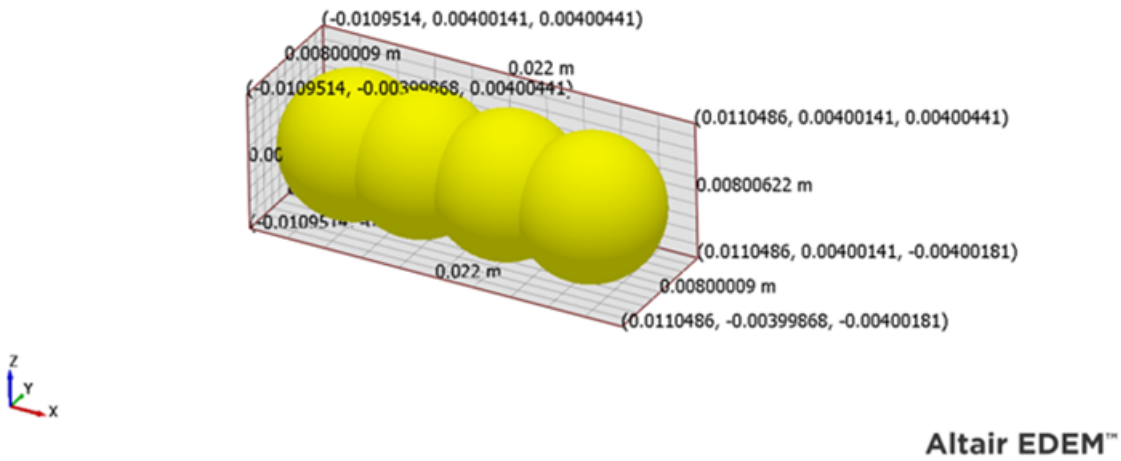


Figure 1526:

10. In the Creator Tree, click **Calculate Properties**.

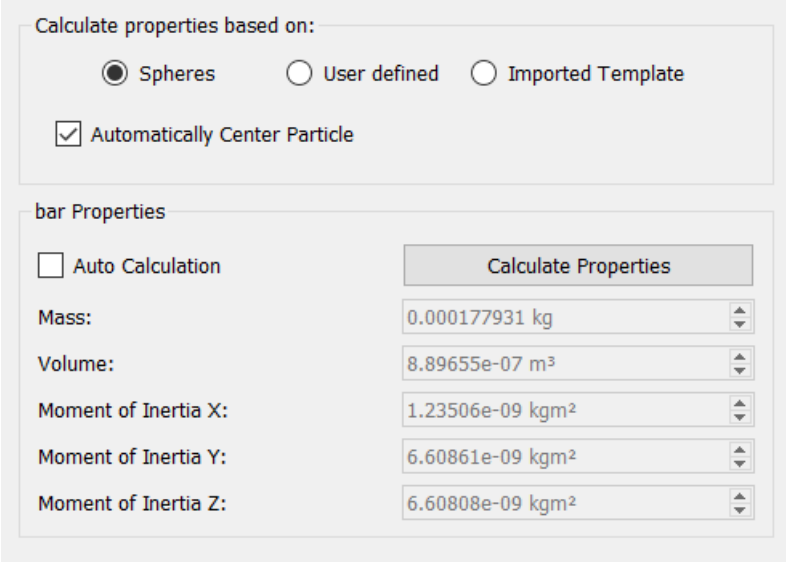


Figure 1527:

Note: Since the "Automatically Center Particle" option is active, EDEM will internally adjust the positions of the spheres in the particle. Although you might see minor changes in the location of the spheres, the overall dimensions of the particle stay the same.

11. Right-click **Bulk Material** and select **Add Bulk Material**.

12. Rename the material to `sphere`.
13. Set the Solids Density property to `1500` kg/m^3.
14. Click below Interaction to define the interaction properties for collisions with sphere particles. In the dialog, select **sphere** and then click **OK**.
15. Set the interaction coefficients as shown in the figure below.

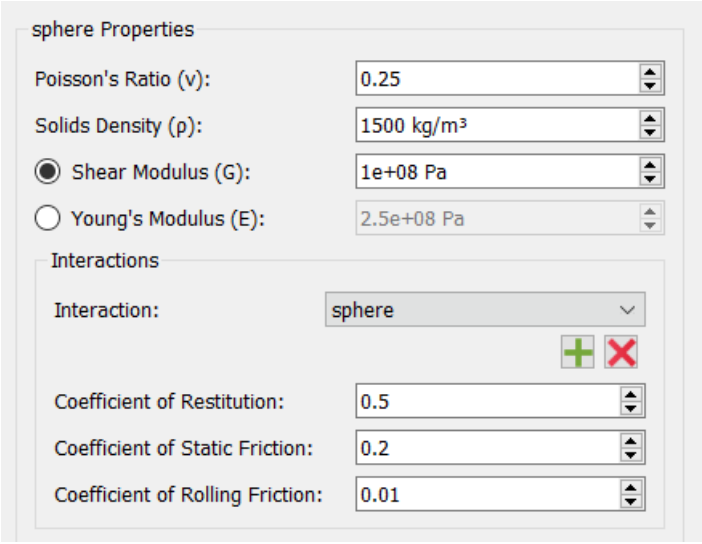


Figure 1528:

16. Click again to define the interaction properties for collisions with bar particles. In the dialog, select **bar** and then click **OK**.
17. Set the interaction coefficients as shown in the figure below.

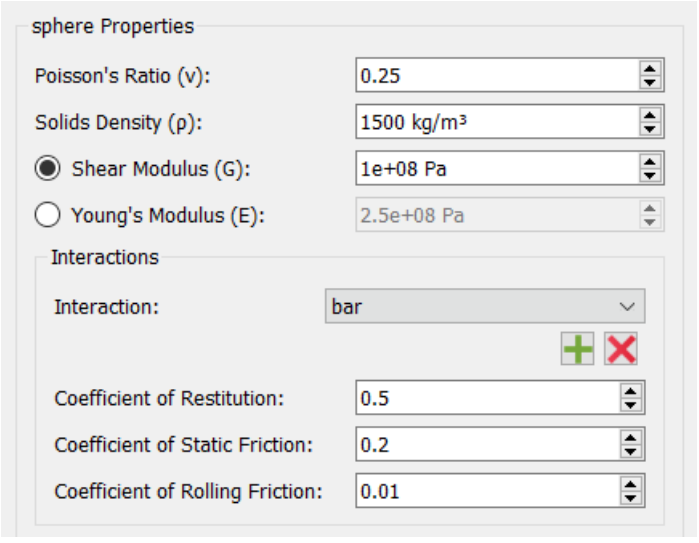


Figure 1529:

18. In the Creator Tree, right click **sphere** and select **Add Shape from Library** > **Single Sphere Shape**.

19. Rename the particle to `sphere`.

20. Under bar, click **Properties**.

21. In the sphere Spheres panel, set the Physical Radius of the sphere to `0.004` m and then press **Enter**.

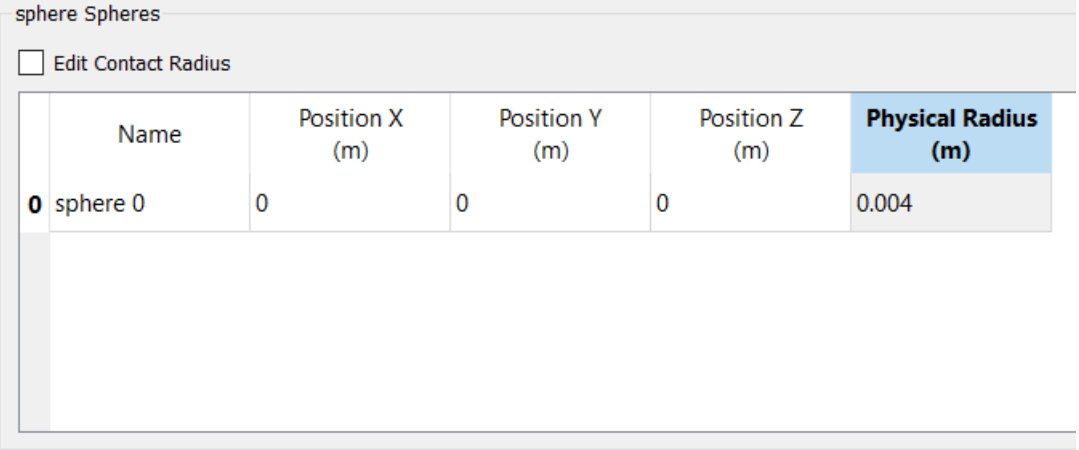


Figure 1530:

22. In the Creator Tree, click **Calculate Properties**.

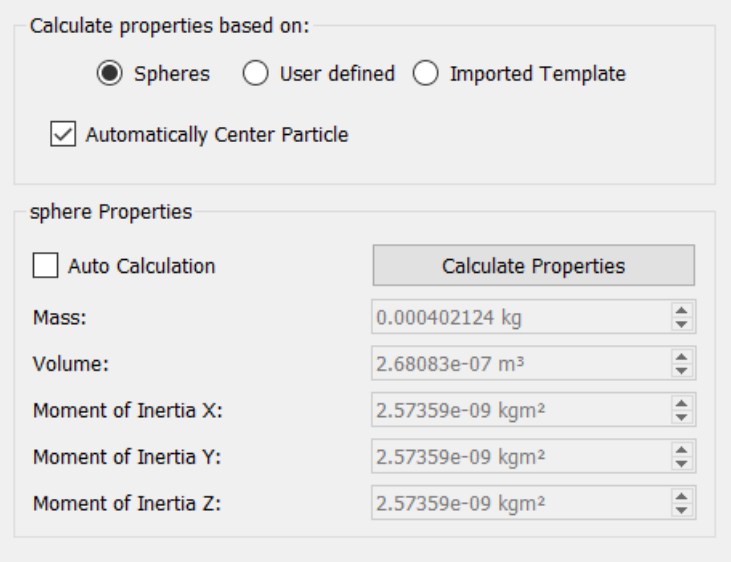


Figure 1531:

23. Right-click **Equipment Material** and select **Add Equipment Material**.

24. Rename the material to `Steel`.

25. Set the Solids Density property to `7800` kg/m^3.

26. Click below Interaction to define the interaction properties for collisions with bar particles. In the dialog, select **bar** and then click **OK**.

27. Set the interaction coefficients as shown in the figure below.

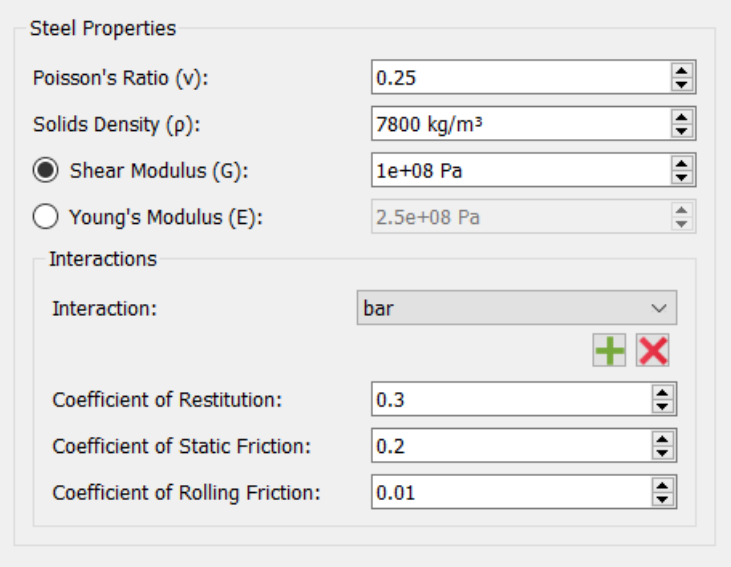


Figure 1532:

28. Click again to define the interaction properties for collisions with sphere particles. In the dialog, select **sphere** and then click **OK**.

29. Set the interaction coefficients as shown in the figure below.

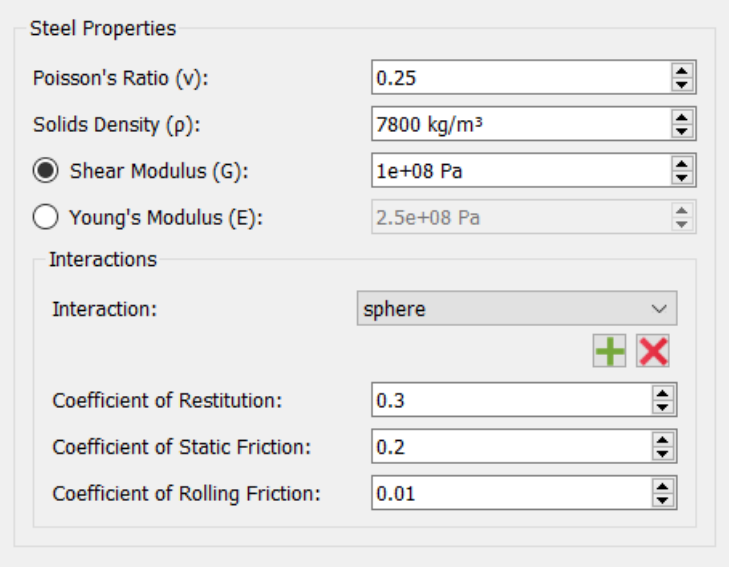


Figure 1533:

30. Save the model.

Define Geometries and Factories

1. In the Creator Tree, expand **Geometries**.

2. Click **Sieve** and verify that the type is set to Physical.
3. Right-click **Sieve** and select **Add Motion** > **Add Sinusoidal Translation Kinematic**.
4. Name it `oscillation` and specify the parameters as shown in the figure below.

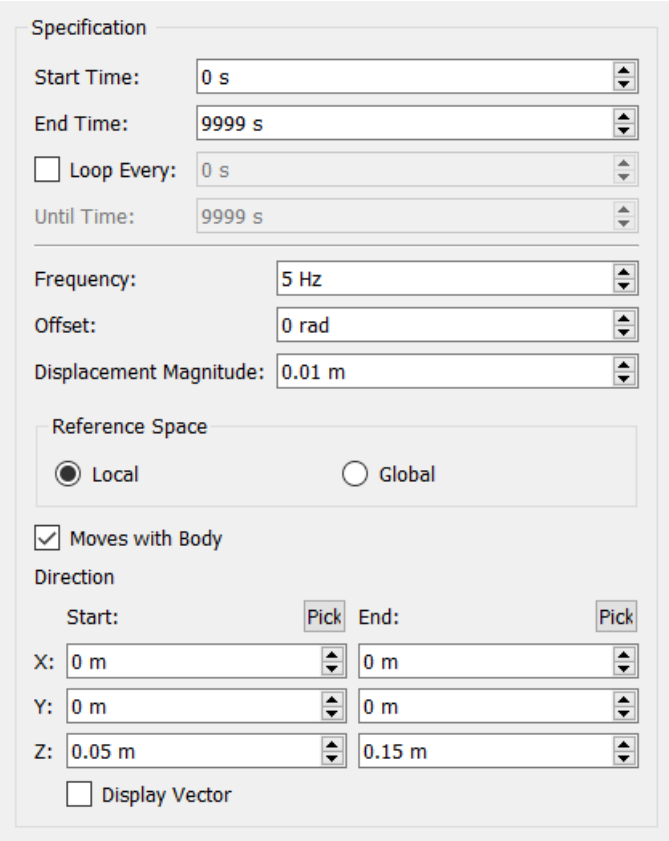


Figure 1534:

5. Click **Outlet** and set the type to **Virtual**.
6. Similarly, set the Inlet type to **Virtual**, the Walls type to **Physical**, and the Material type to **Steel** (if not set already).
7. Click **factory** and set the type to **Virtual**
8. Right-click **factory** and select **Add Factory**. Name it `bar factory`.
9. Set the particle generation parameters as shown in the figure below.

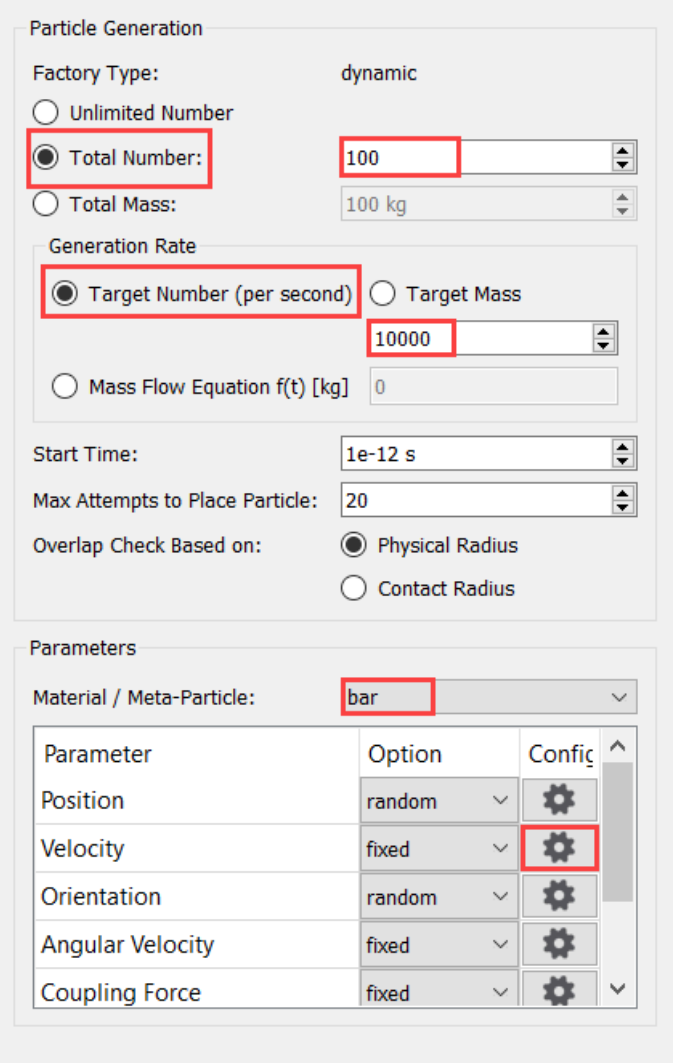


Figure 1535:

10. Click ⚙ besides Velocity, set the Z-velocity to `-1` m/s, and then click **OK**.
11. Repeat steps 8-10 to create another factory named `sphere factory`. Use the same parameters but set the Total Number to `200` and the Material to **sphere**.
12. Save the model.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).

When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.

3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and de-activate the **Auto Time Step** checkbox.

   Note: The Auto time step should never be used for AS-EDEM bidirectional coupled simulations.

3. Set the Fixed Time Step to `1e-5` s.

   Note: Generally, a value of 20-40% of the Rayleigh Time Step is recommended as the time step size to ensure stability of the DEM simulation. When setting the time step for the DEM model, make sure that the CFD time step is an integer multiple of DEM time step. In this case, the CFD time step is 1e-3, which is 50 times the DEM time step. This ensures that the physical time in both the solvers match at the end of each data exchange interval. Also, the recommended range for the CFD-to-DEM time step ratio is 50-200 for most cases.

4. Set the Total Time to `1` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `2.5` R min.

   Generally, a value in the range of 2-6 R_{min} is recommended as the optimum cell size.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

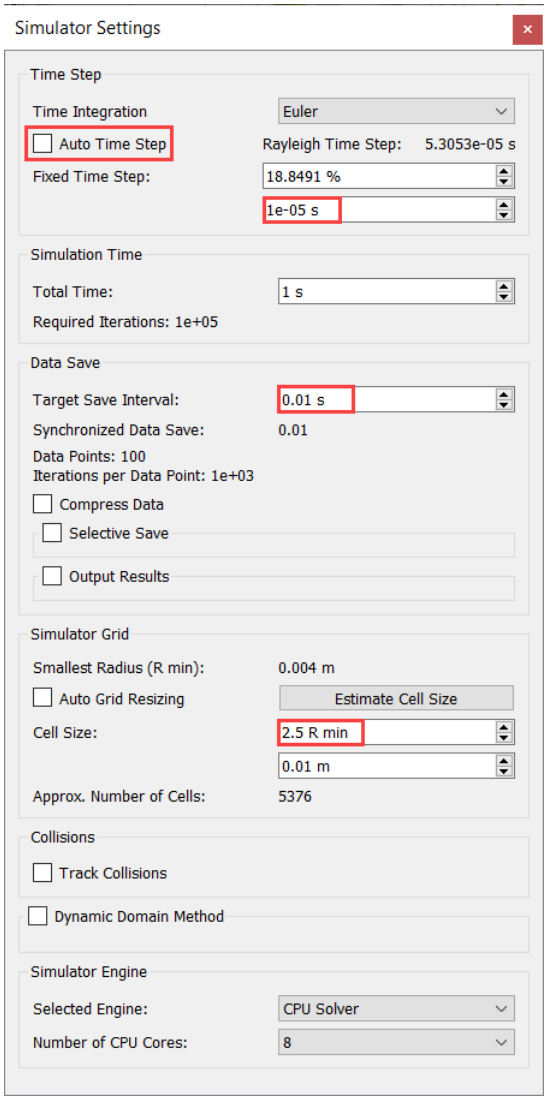


Figure 1536:

7. Once the simulation settings have been defined, save the model.

Part 2 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1537:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6104_sieve.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Save the database as `sieve_nonspherical` in the same directory as the other input files.

   This will be the working directory and all the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1538:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

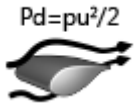


Figure 1539:

The **Setup** dialog opens.

2. Under the Physics models setting, select the **Multiphase flow** radio button.

3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the Material Library dialog, select **EDEM 2 Way Multiphase**, switch to the **My Material** tab, then click + to add a new material model.
6. In the microdialog, click **EDEM Bidirectional Material** in the top-left corner and change the name to `Air-Particle`.
7. Set the Carrier field to **Air** (if not set already).
8. Set the particle shape to **Non-Spherical**.
9. Set the drag model to **Rong**.
10. Set the Non spherical drag coefficient model to **Holzer-Sommerfeld**.

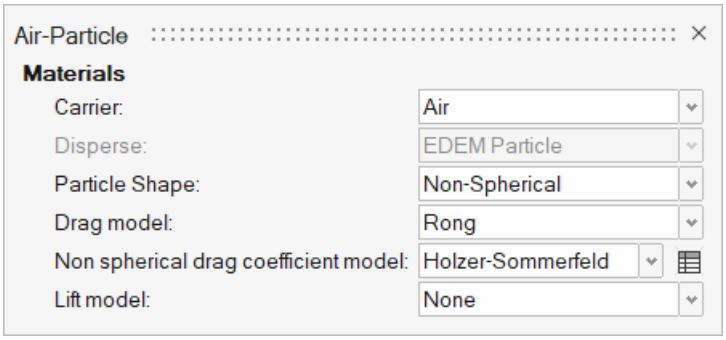


Figure 1540:

11. Click the table icon beside the drag coefficient model drop-down.
12. In the new dialog, click ✚ to add a new row, then enter the values for Aspect Ratio and Volume as shown in the figure below.

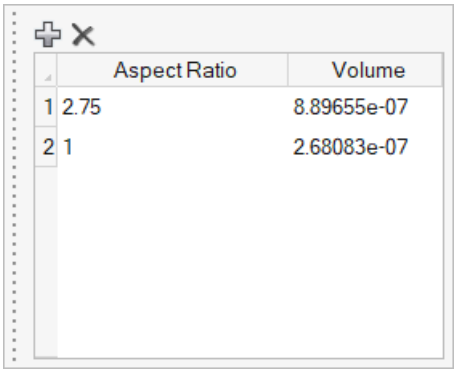


Figure 1541:

Note: The volumes entered here should be obtained from EDEM under particle properties. There might be slight variation in the volume values that you may see in your EDEM model but that is fine.

Here the aspect ratios are calculated based on the ratio of the length to diameter of the particles. The AR of 2.75 and 1 are for bar and sphere-shaped particles respectively. The volume of the particles can be obtained from EDEM under the particle properties. If the simulation has particles of a single shape, then the volume field can be set to zero.

The image below explains the aspect ratio definition for some standard non-spherical particle shapes.

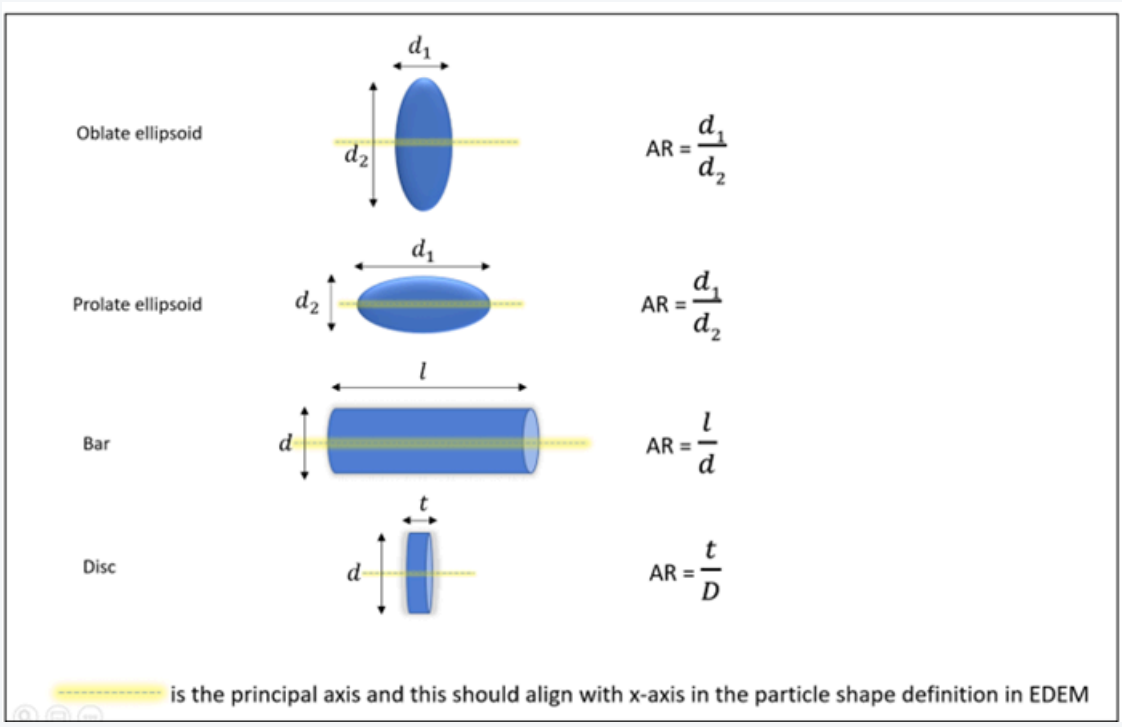


Figure 1542:

13. Press **Esc** to close the dialog and then close the material model and Material Library dialogs as well.
14. In the Setup dialog, set the Eulerian Material to **Air-Particle**.
15. Set Time step size and Final time to `0.005` and `1`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
16. Verify that Gravity is set to `0, 0, -9.81`.
17. Set the Pressure scale to **Absolute**.

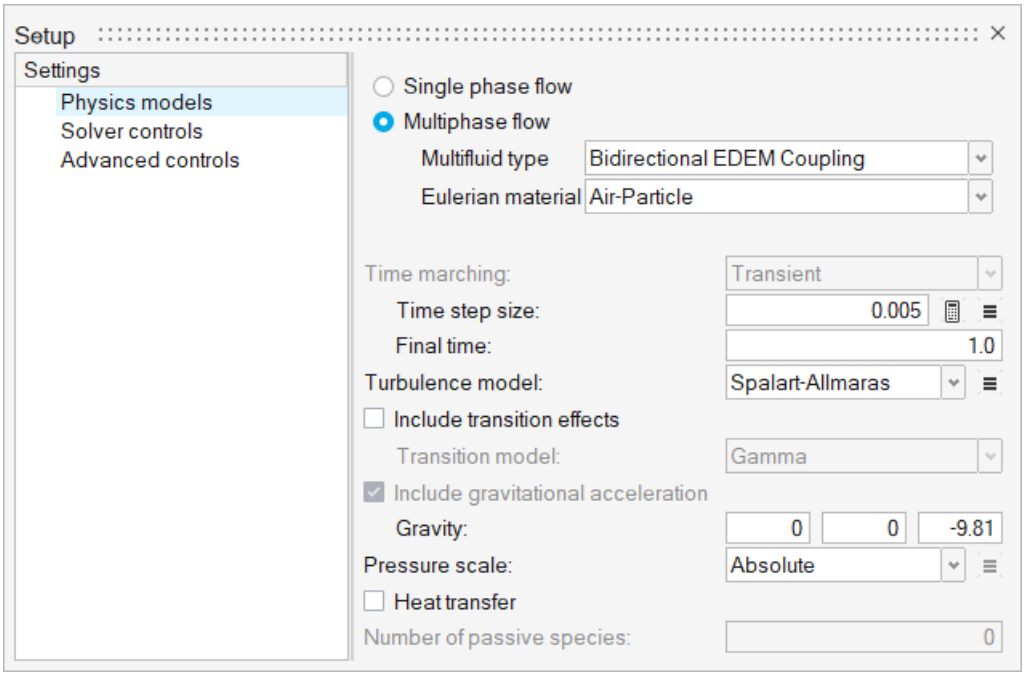


Figure 1543:

18. Click the **Solver controls** setting and set the Maximum stagger iterations to `3`.

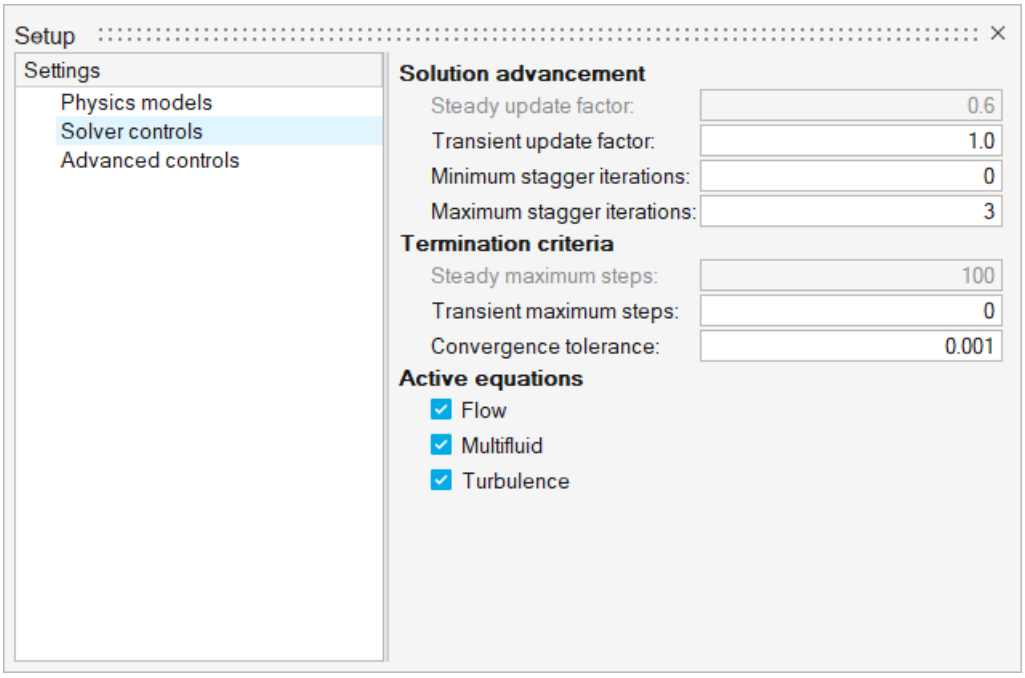


Figure 1544:

19. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1545:

2. Verify that Air-Particle has been assigned as the material.
3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1546:

2. Click on the inlet face highlighted in the figure below. In the microdialog, enter a value of `6` m/s for the Average velocity and `1.0` for the Carrier fluid volume fraction.

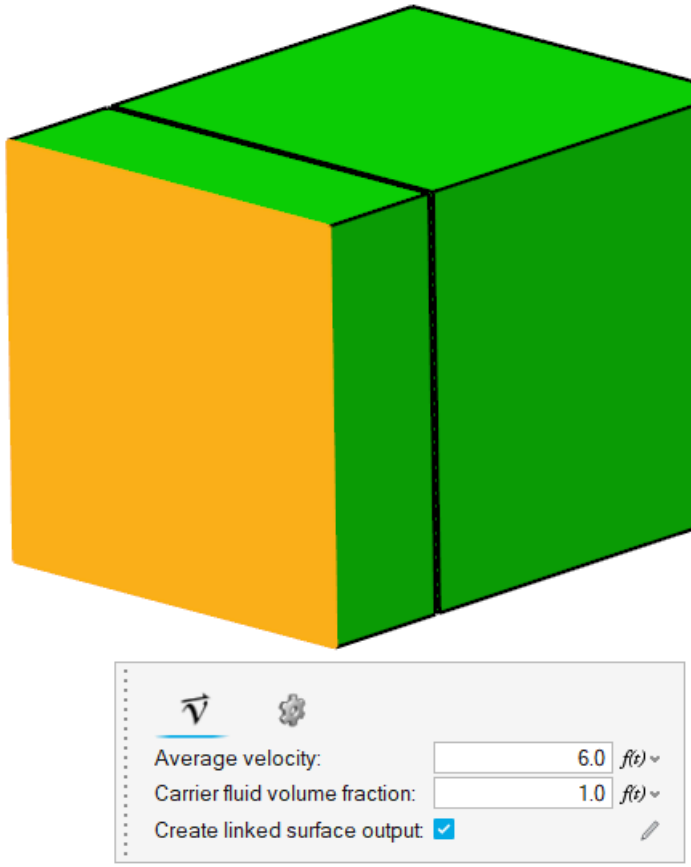


Figure 1547:

3. On the guide bar, click ✓ to execute the command and exit the tool.
4. Click the **Outlet** tool.

Figure 1548:

5. Select the face highlighted in the figure below then click ✓ on the guide bar.

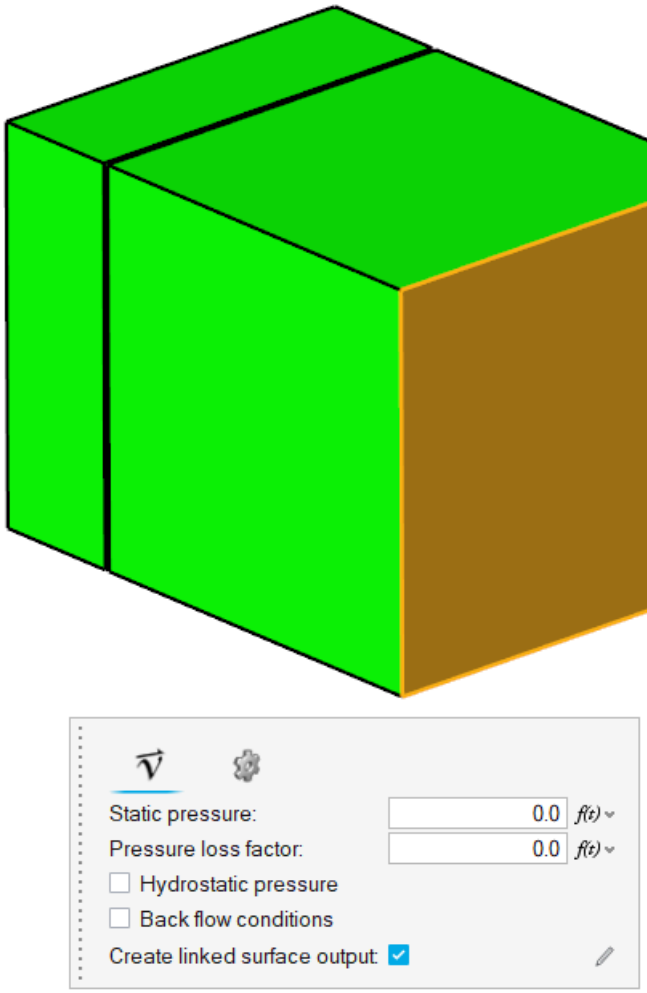


Figure 1549:

6. Save the model.

Set Up Motion

Select the Motion Type

1. From the Motion ribbon, click the **Settings** tool.

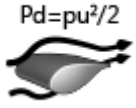


Figure 1550:

2. In the Setup dialog, set the Mesh motion to **Arbitrary** and close the dialog.

Define the Mesh Boundary Conditions

1. From the Motion ribbon, click the **Planar Slip** tool.

Figure 1551:

2. Select the two surfaces highlighted in the figure below.
   The surfaces selected have the minimum y-coordinate.

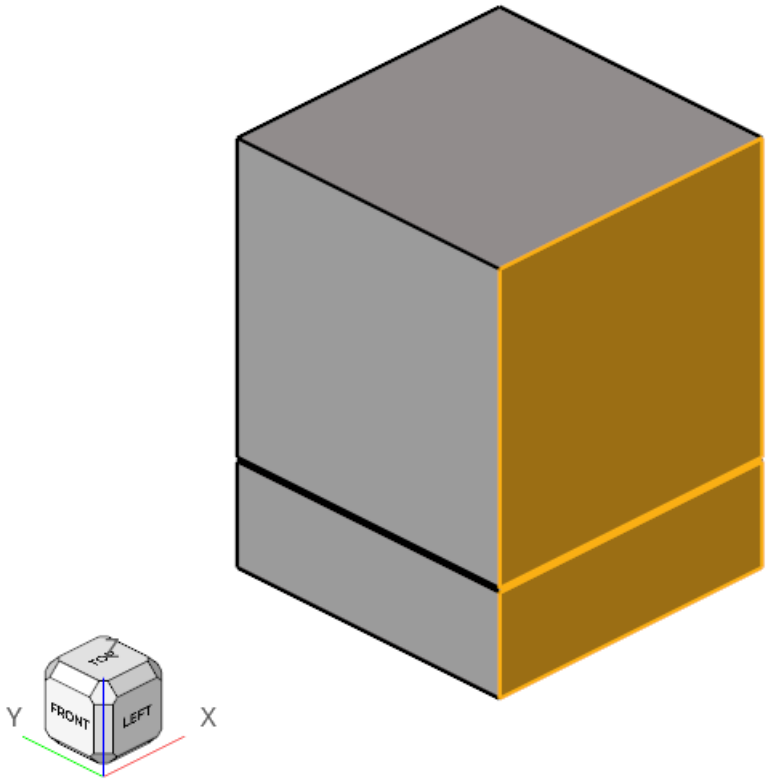


Figure 1552:

3. In the Mesh Motion legend, double-click **Planar Slip** and change the name to `y_neg`.
4. On the guide bar, click ▸ to execute the command and remain in the tool.

5. Repeat steps 2-4 to create three more planar slip boundary conditions for the surfaces with the maximum y-coordinates, and the minimum and maximum x-coordinates. Name these groups `y_pos`, `x_neg`, and `x_pos` respectively.

Once the four planar slip conditions are defined, the Mesh Motion legend should look like the one shown below.

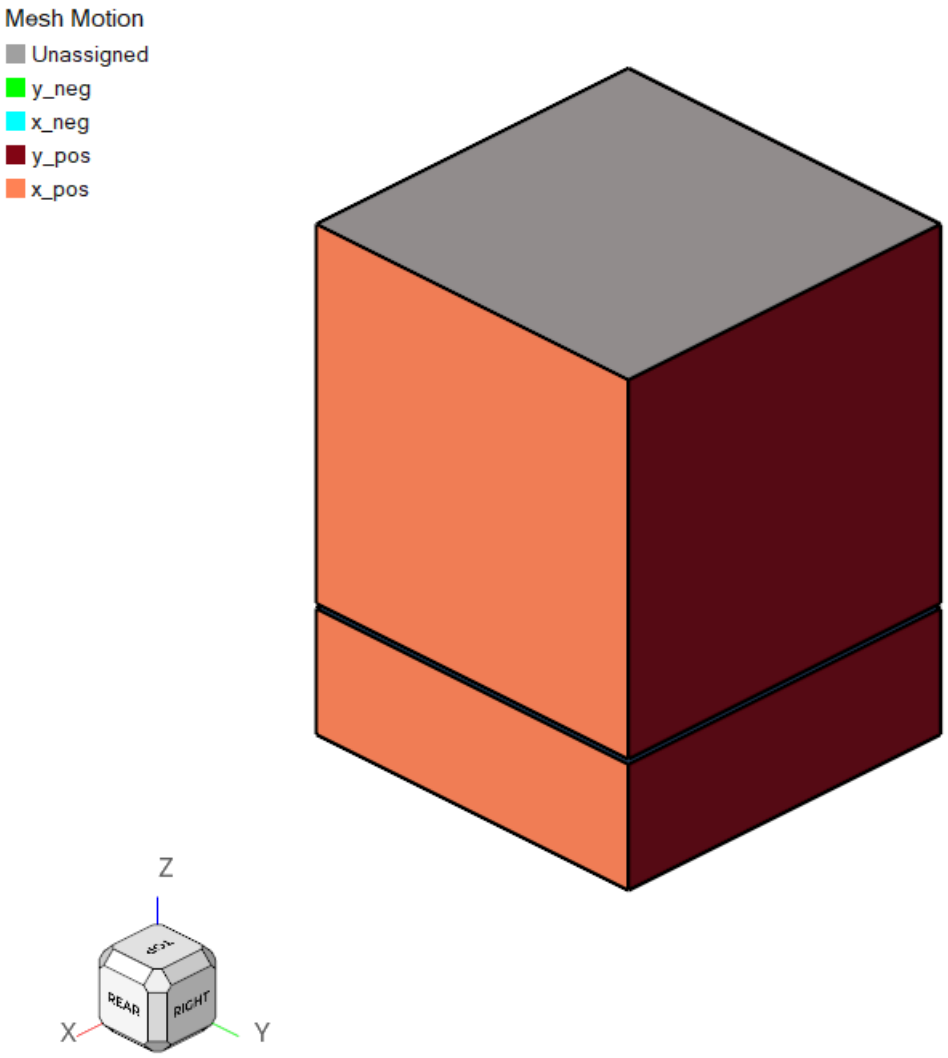


Figure 1553:

6. Save the model.

Define Sinusoidal Translational Motion

1. From the Motion ribbon, click the **Translation** tool.

Figure 1554:

2. In the Mesh Motion legend, right-click **Unassigned** and select **Isolate**.
3. On the guide bar, change the active selector to **Surfaces**.
4. Using box selection, select all the sieve surfaces as shown in the figure below. Ensure that you are not selecting the inlet or outlet faces.

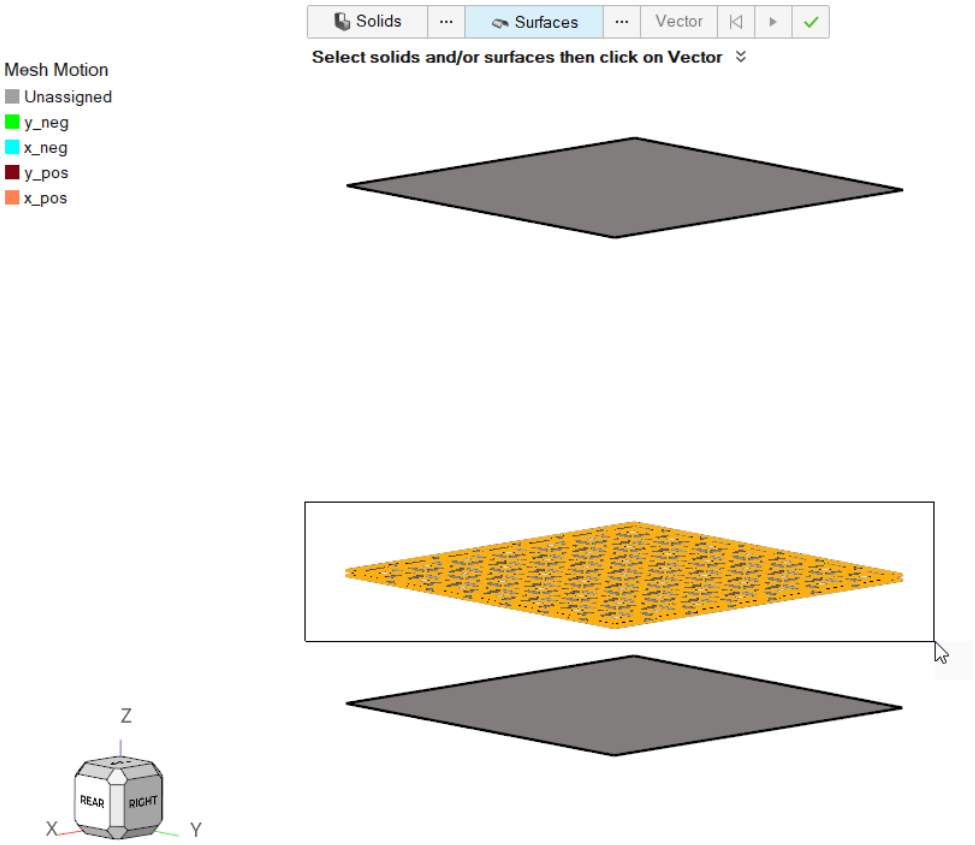


Figure 1555:

5. Click **Vector** on the guide bar.
6. Click anywhere on the sieve surfaces in the modeling window.
7. In the microdialog, click **Z** to align the axis to the global z axis.

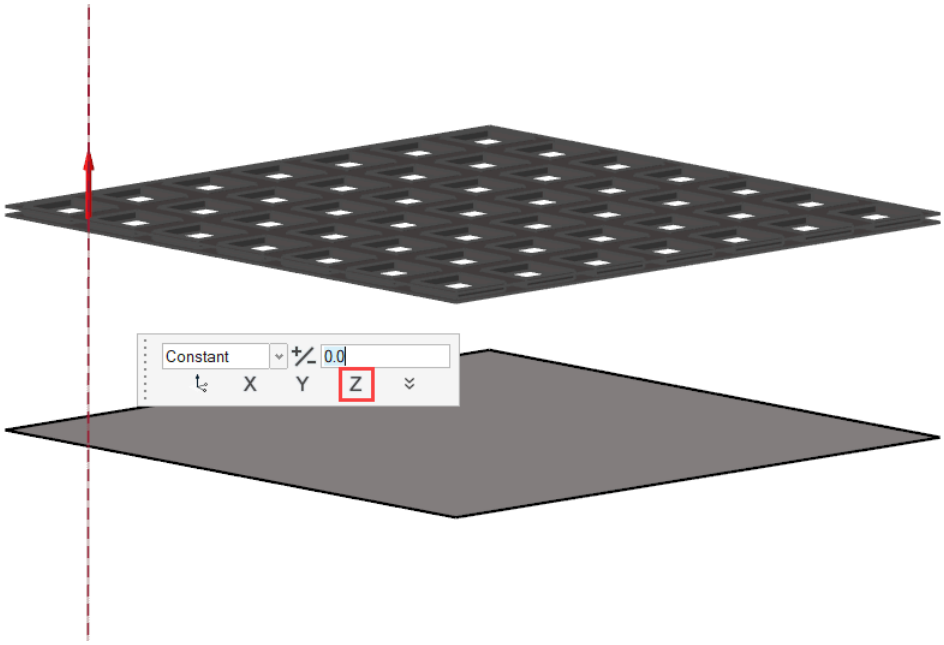


Figure 1556:

8. In the microdialog, change the type to **Time-varying** and click the plot icon.

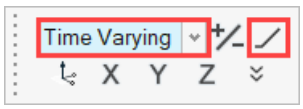


Figure 1557:

9. In the new plot dialog, click the drop-down beside the table icon for function specification and select the oscillating option.

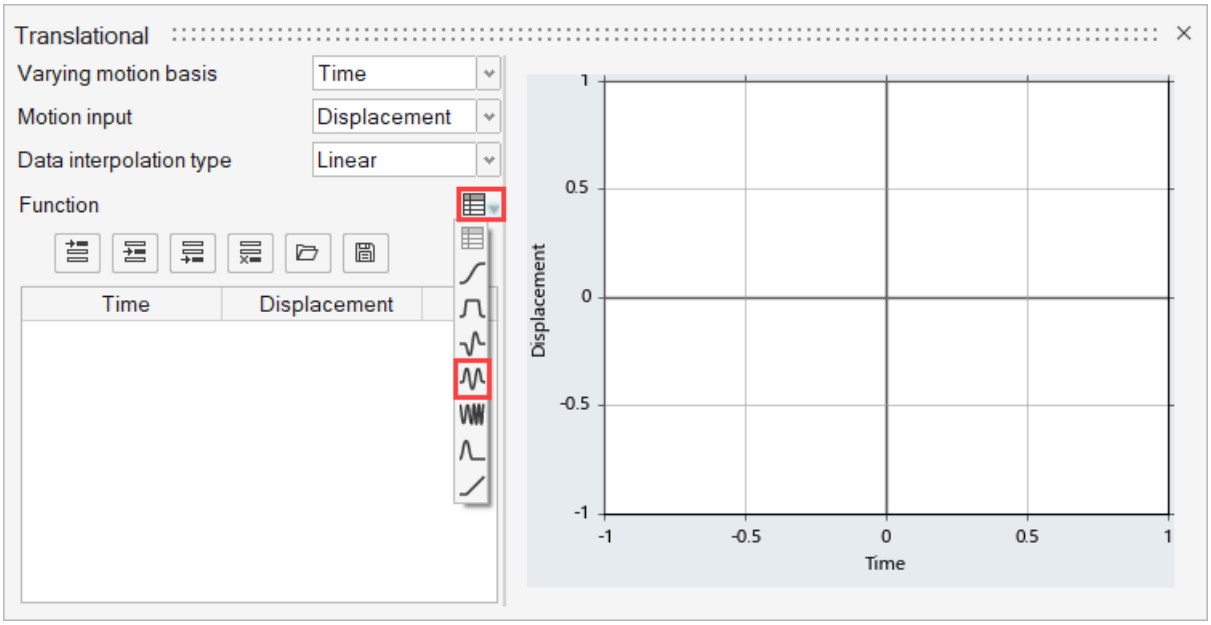


Figure 1558:

10. Enter the values for the sine function as shown in the figure below.

Figure 1559:

11. Click **Convert** to transform the sine function into an array for the multiplier function.
The motion profile for the translation motion should look like the one shown below.

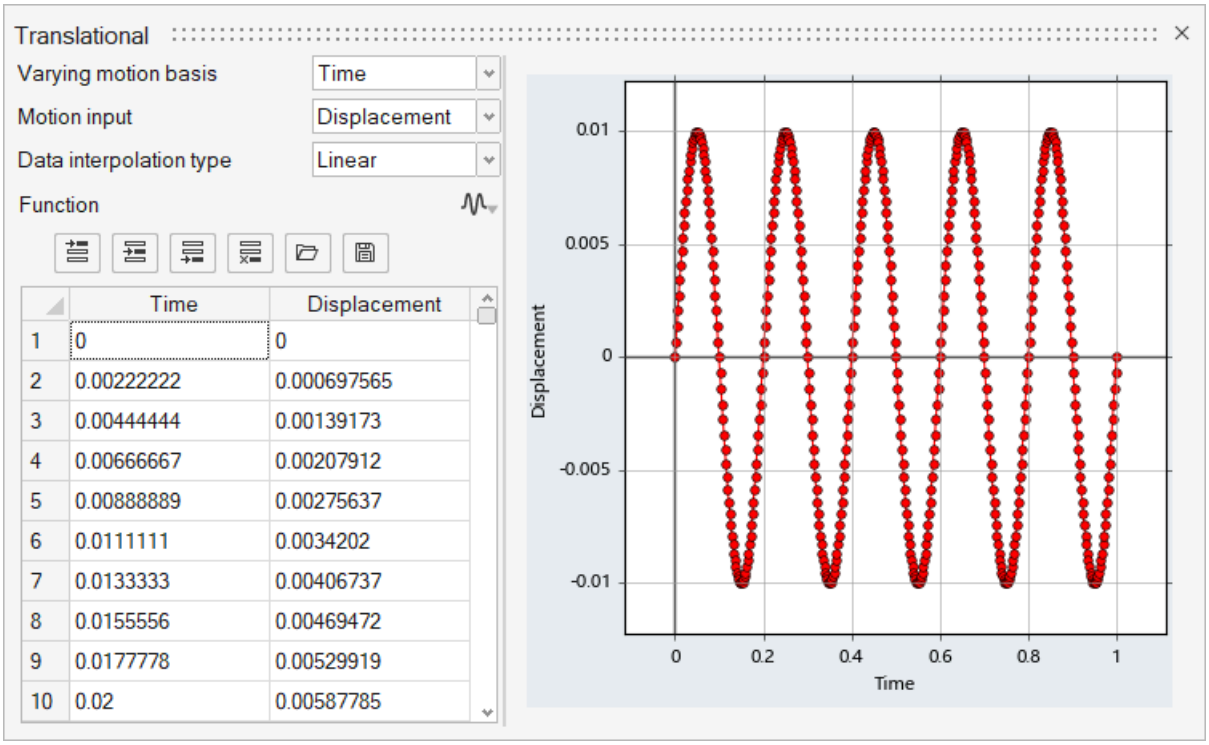


Figure 1560:

12. Click ✓ on the guide bar.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

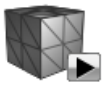


Figure 1561:

The **Meshing Operations** dialog opens.

2. Set the Mesh size to **Maximum size** and change the Maximum element size to `0.03`.

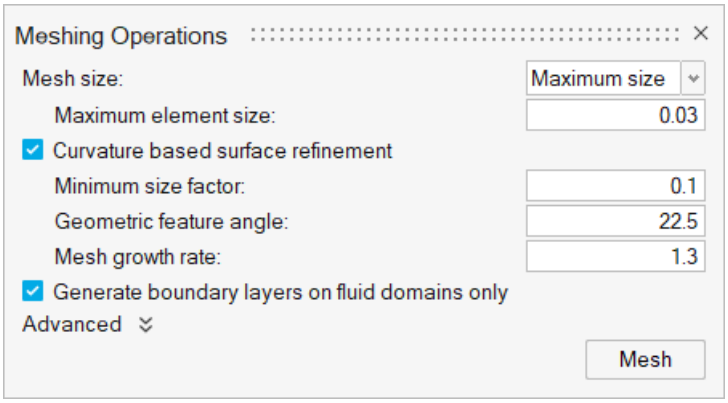


Figure 1562:

3. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

4. Save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

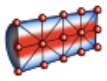


Figure 1563:

   The **Field Output** dialog opens.
2. Check the box for **Write Initial Conditions**.
3. Uncheck the box for **Write results at time step interval**
4. Check the box for **Write results at time interval**.
5. Set the Time step interval to `0.05`.

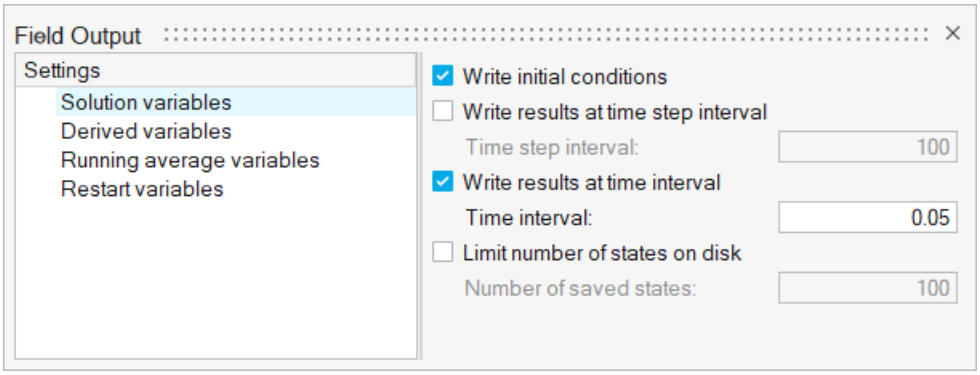


Figure 1564:

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1565:

Once the Coupling server is activated, the icon changes.

Figure 1566:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1567:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Optional: Set the number of processors to **4** or **8** based on availability.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the z-velocity value to `6`. Uncheck **Pre-compute Turbulence**.
7. Click **Run** to launch AcuSolve.

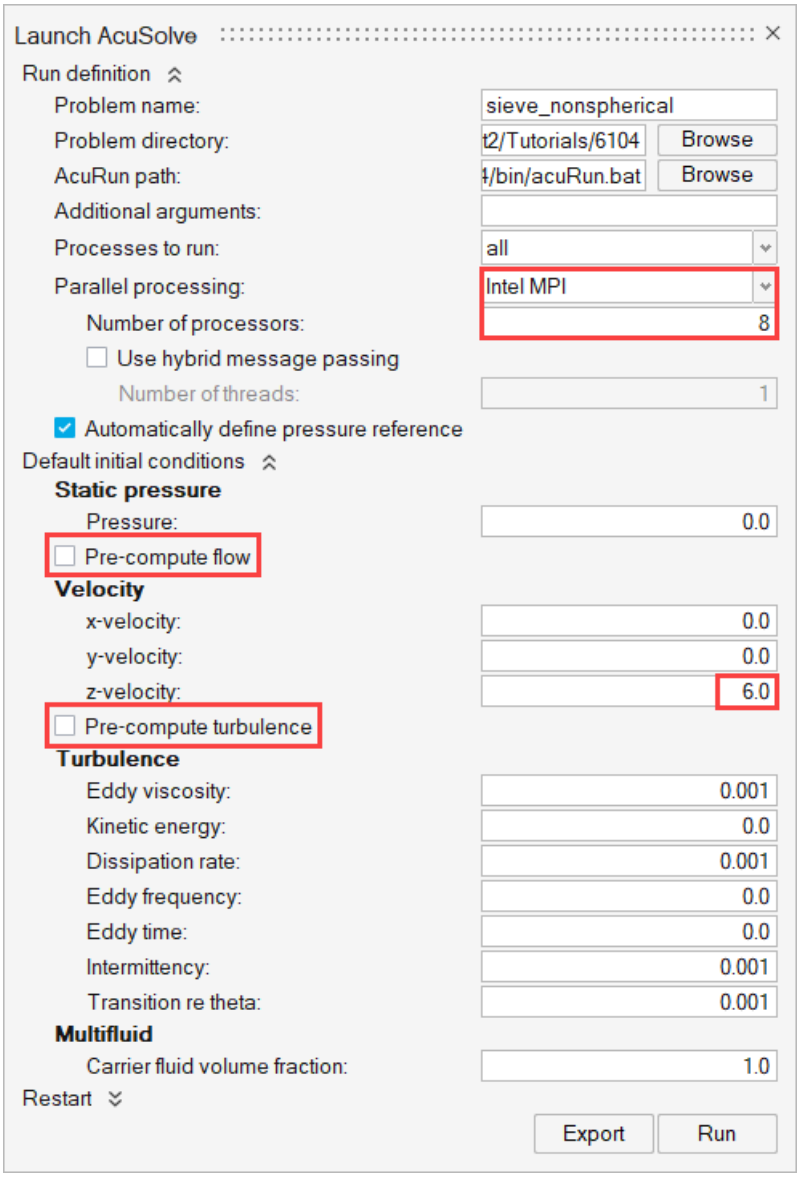


Figure 1568:

Once the AcuSolve run is launched, the **Run Status** dialog opens.

8. In the dialog, right-click on the AcuSolve run and select **View log file**.

   If the coupling with EDEM is successful, that information is printed in the log file.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 3.499000e+000 2.099300e+002 Sec
acuSolve: Input Memory Usage        = 1.930000e+002 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Interface thickness is 7.166633e-003
acuSolve: Init. CPU/Elapse time     = 5.470000e-001 7.800000e-002 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 5.000000e-003 ; time= 0.000000e+000
```

Figure 1569:

Once the simulation is complete, the summary of the run time is printed at the end of the log file.

```
acuSolve: ---------- End Time Step ----------
acuSolve: ---------- Offset Values ----------
acuSolve:          Pressure Offset = 0.000000e+000
acuSolve:       Temperature Offset = 0.000000e+000
acuSolve: ----------------------------------------------------------
acuSolve: Input      CPU/Elapse time= 4.046000e+000 2.100080e+002 Sec
acuSolve: Output     CPU/Elapse time= 4.941280e+002 6.421500e+001 Sec
acuSolve: Exchange   CPU/Elapse time= 8.689300e+001 1.427700e+001 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Mesh-Displacement stagger "mesh_displacement":
acuSolve:  No of res/lhs formations =       658       658
acuSolve:  Elem.Form CPU/Elapse time= 2.275490e+002 2.958800e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 2.984240e+002 3.927900e+001 Sec
acuSolve: Flow stagger "flow":
acuSolve:  No of res/lhs formations =       489       489
acuSolve:  Elem.Form CPU/Elapse time= 1.078415e+003 1.416500e+002 Sec
acuSolve:  Solution  CPU/Elapse time= 8.442620e+002 1.107780e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:  No of res/lhs formations =       488       488
acuSolve:  Elem.Form CPU/Elapse time= 6.518830e+002 8.497400e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 1.032560e+002 1.330500e+001 Sec
acuSolve: Edem stagger "edem":
acuSolve:  No of external code calls=       200
acuSolve:  EDEM ext. code CPU/Elapse time= 5.343860e+002 4.907760e+002 Sec
acuSolve: Field-Velocity stagger "field_velocity: EDEM Particle":
acuSolve:  No of res/lhs formations =       200       200
acuSolve:  Elem.Form CPU/Elapse time= 4.743050e+002 6.217200e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 4.733100e+001 6.066000e+000 Sec
acuSolve: Field-Velocity stagger "field_velocity: Air":
acuSolve:  No of res/lhs formations =       200         0
acuSolve:  Elem.Form CPU/Elapse time= 1.250000e-001 1.800000e-002 Sec
acuSolve:  Solution  CPU/Elapse time= 3.435500e+001 4.461000e+000 Sec
acuSolve: Fields stagger "EDEM Particle":
acuSolve:  No of res/lhs formations =       200       200
acuSolve:  Elem.Form CPU/Elapse time= 2.881690e+002 3.729400e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 4.368800e+001 5.736000e+000 Sec
acuSolve: Fields stagger "Air":
acuSolve:  No of res/lhs formations =       200         0
acuSolve:  Elem.Form CPU/Elapse time= 0.000000e+000 1.400000e-002 Sec
acuSolve:  Solution  CPU/Elapse time= 3.478200e+001 4.581000e+000 Sec
acuSolve: LnSlv. (host)  Elapse time= 1.385390e+002 Sec
acuSolve: Total CPU/Elapse time     = 6.128562e+003 1.431686e+003 Sec
acuSolve: Total Memory Usage        = 2.270000e+002 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
```

Figure 1570:

Analyze the Results

AcuSolve Post-Processing

1. In HyperMesh CFD, right-click on the AcuSolve run in the **Run Status** dialog and select **Visualize results**.

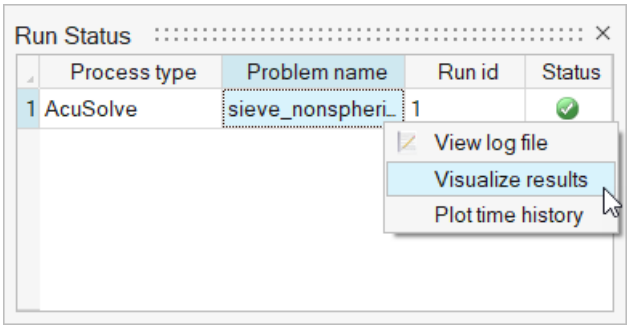


Figure 1571:

The results are loaded in the Post ribbon.

2. Click the **Slice Planes** tool.

Figure 1572:

3. Select the x-z plane as highlighted in the figure below.

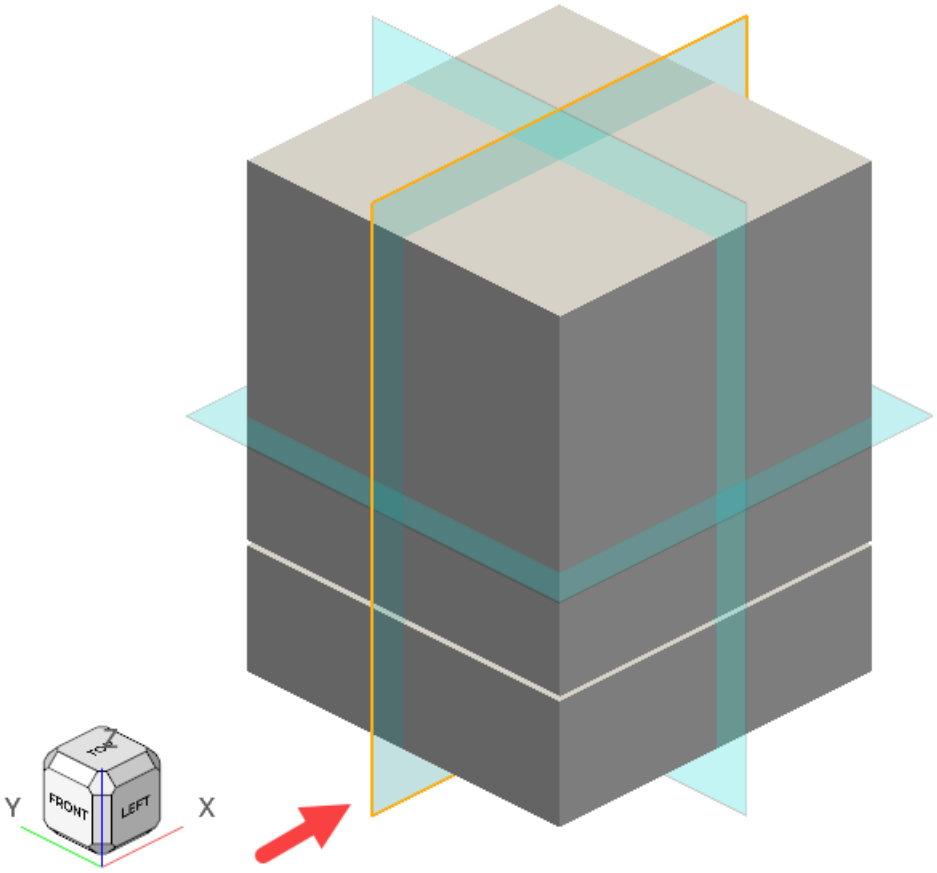


Figure 1573:

4. In the slice plane microdialog, click to create the slice plane.
5. In the display properties microdialog, set the display to **velocity** and activate the **Legend** toggle.
6. Change the bounds of the legend to `0` and `15`.
7. Click and set the Colormap name to **Rainbow Uniform**.

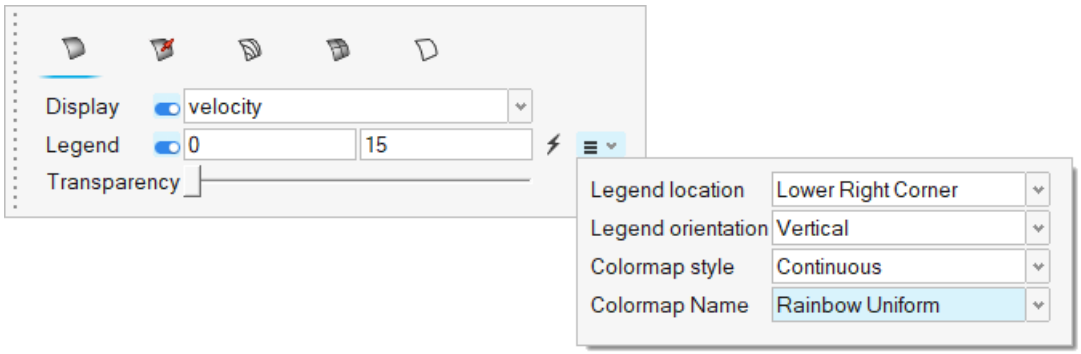


Figure 1574:

8. Click ✓ on the guide bar.

9. In the Post Browser, turn off the visibility of **Flow Boundaries** by clicking on its icon.

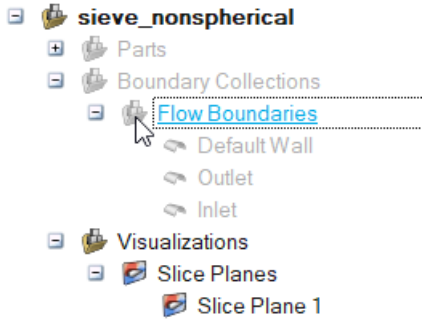


Figure 1575:

10. Select the **Left** face on the view cube to align the model to the x-z plane.

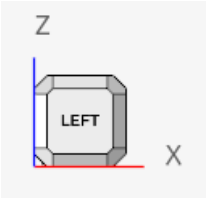


Figure 1576:

11. Click ▶ on the animation toolbar to view the animation of the velocity contour.

Figure 1577:

EDEM Post-Processing

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then select all geometries by holding the Ctrl key.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

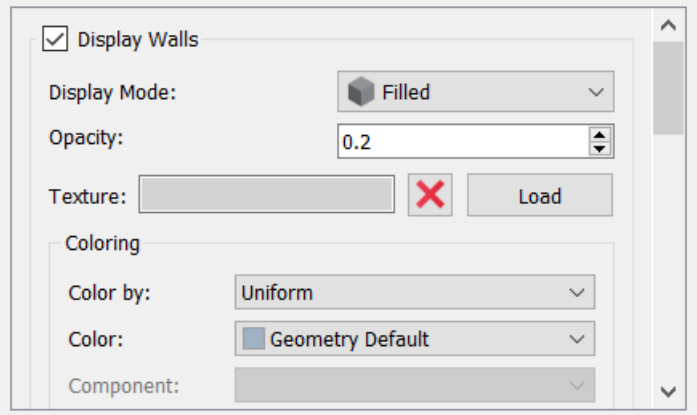


Figure 1578:

4. In the Analyst Tree, click **Particles**.
5. Set the coloring to **Velocity**.
6. Activate the **Auto Update** checkboxes for both the Min and Max Value.
7. Activate the **Show Legend** checkbox.
8. Click **Apply All**.

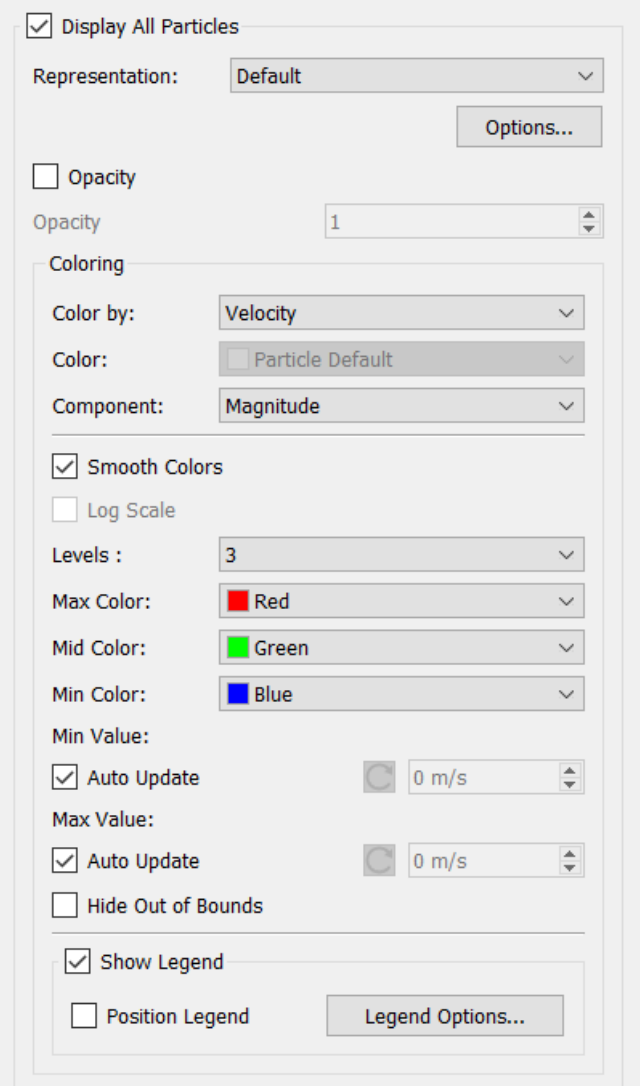


Figure 1579:

9. On the menu bar, set the time to 0 by clicking:

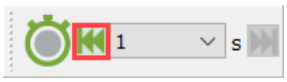


Figure 1580:

10. Set the View plane to **+ Y**.

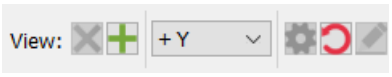


Figure 1581:

11. In the Viewer window, set the Playback Speed to `0.1x` and then click ▶ to play the particle flow animation.

Figure 1582:

Observe that the lighter bar shaped particles are blown away by the incoming fluid while the heavier particles fall through the sieve openings.

Summary

In this tutorial, you learned how to set up and run a basic AcuSolve-EDEM bidirectional (two-way) coupling problem with non-spherical particles. You learned how to create a non-spherical particle in EDEM and set up the AcuSolve model to consider the effect of the particle shape for the fluid-particle interaction forces. Once the simulation was complete, you learned how to post-process the AcuSolve results using HyperMesh CFD and the EDEM results.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/5koc9uaegv

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6105 Single Particle Sedimentation – Effect of Lift and Torque

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial demonstrates the effect of lift and torque forces on a non-spherical particle falling through a fluid using AcuSolve-EDEM bidirectional coupling. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction and HyperMesh CFD, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6105_cylinder.hm`

`cylinder_EDEM.zip`

Problem Description

The problem to be solved is shown schematically in the figures below. The model consists of simple tank filled with water. A cylindrical particle is introduced at the center of the tank and allowed to fall freely through the water column. The particle will be introduced at an orientation such that its principal axis is at 150° to the horizontal axis. As the particle travels in the downward direction, a non-uniform flow field is developed due to the fluid-particle interaction. As a result the hydrodynamic forces on the particles introduce a torque on the particle as the center of pressure does not coincide with the center of mass. In addition, because of the orientation and the non-spherical shape, an additional lift force makes the particle to steer away from the central line of path.

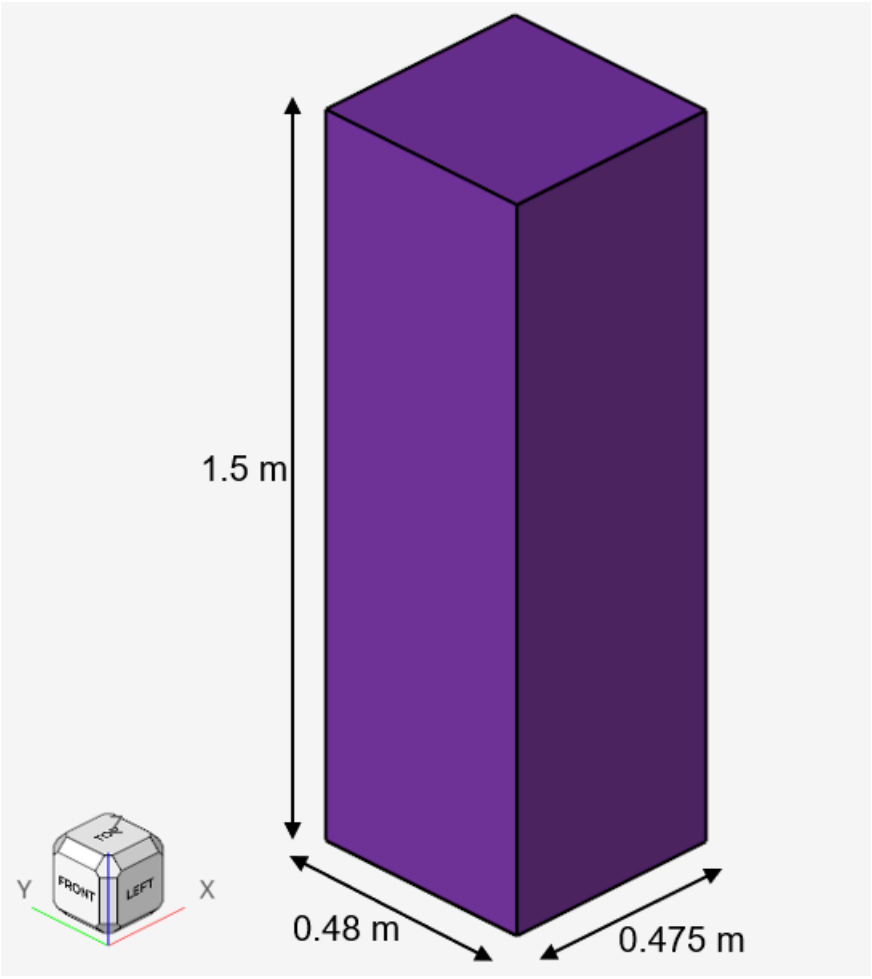


Figure 1583:

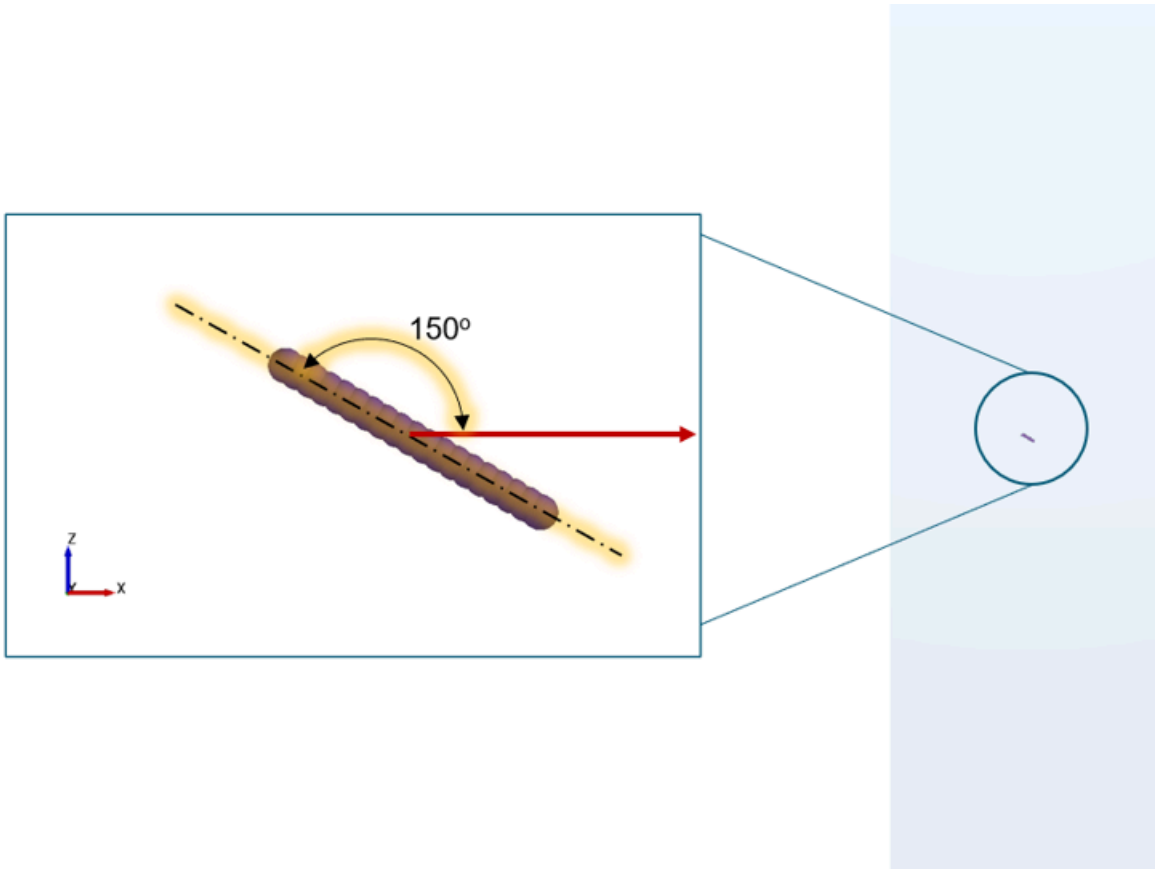


Figure 1584:

AcuSolve-EDEM bidirectional coupling is used to model the interaction between the fluid and particle. In this tutorial, the Rong drag model in conjunction with the Ganser non-spherical drag-coefficient model is used such that the effect of non-spherical shape is considered. The length scale used for factoring in the shape of the particles is the aspect ratio. A cylindrical particle with an aspect ratio of 9.242 (L/D = 0.05 m/0.00541 m) is used for this simulation.

This tutorial also introduces you to the concept of generating particles in EDEM at a specific location and with a specific orientation. When defining the particle factory, you have the option to define certain properties of particles such as velocity, angular velocity, position, orientation, temperature, and so on. Properties such as orientation and position can be defined by you or set to random. The steps listed below illustrate the general process for calculating the orientation matrix required in EDEM for creating particles with a specific orientation.

1. Generating particles at a certain orientation requires an orientation matrix that transforms the particle relative to the initial orientation specified during the particle shape definition. For rotation about an axis (n) by an angle (a), the orientation matrix ($R_{a,\hat{n}}$) is given by
2. $$R_{a,\hat{n}} = \begin{bmatrix} a^2K+\cos a & abK-c & \sin a & acK+b & \sin a \\ abK+c & \sin a & b^2K+\cos a & bcK-a & \sin a \\ acK-b & \sin a & bcK+a & \sin a & c^2K+\cos a \end{bmatrix}$$

3. Where, $K = (1 - cosa)$ and a, b, c are the components of the unit vector of the axis of rotation, $\hat{n} = ai + bj + ck$.

   Note: The right-hand thumb rule is used for determining the sign of the angle of rotation where the thumb points towards the axis of rotation and the curl of the fingers points towards the positive direction of the angle of rotation.

4. In the current example, the particle should be rotated by an angle of 30^o about the y-axis. Therefore, $a = 30^o$, $K = 0.133974596$, a=0, b=1 and c=0. By substituting these values, you get the following orientation matrix:

5. $$R_{a,\hat{n}} = \begin{bmatrix} 0.8660254 & 0 & 0.5 \\ 0 & 1 & 0 \\ -0.5 & 0 & 0.8660254 \end{bmatrix}$$

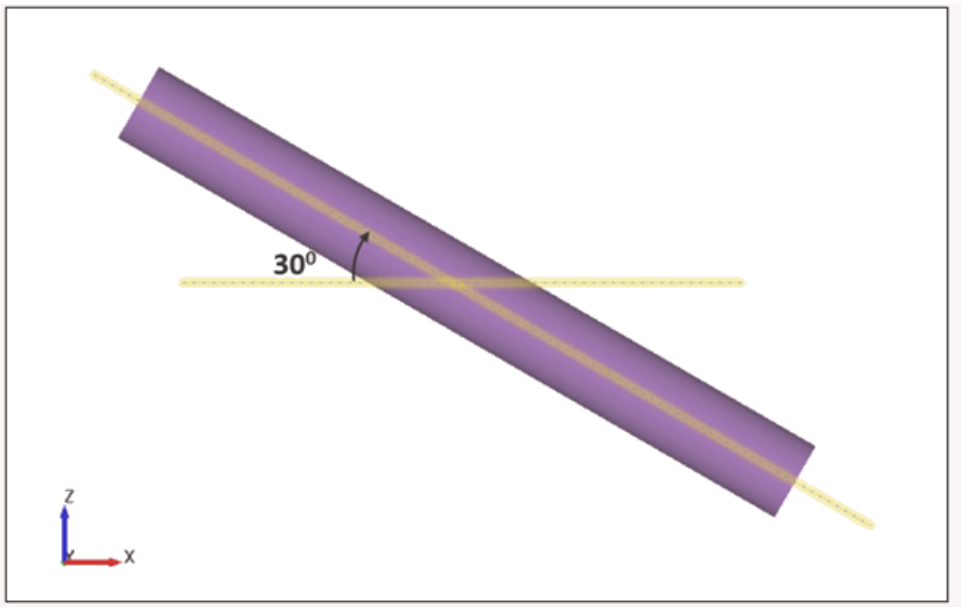
6. 

Figure 1585:

Part 1 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

1. From the menu bar, click **File** > **Open**.
2. In the dialog, browse to your problem directory and open the `cylinder.dem` file.
   The geometry and the materials are loaded.

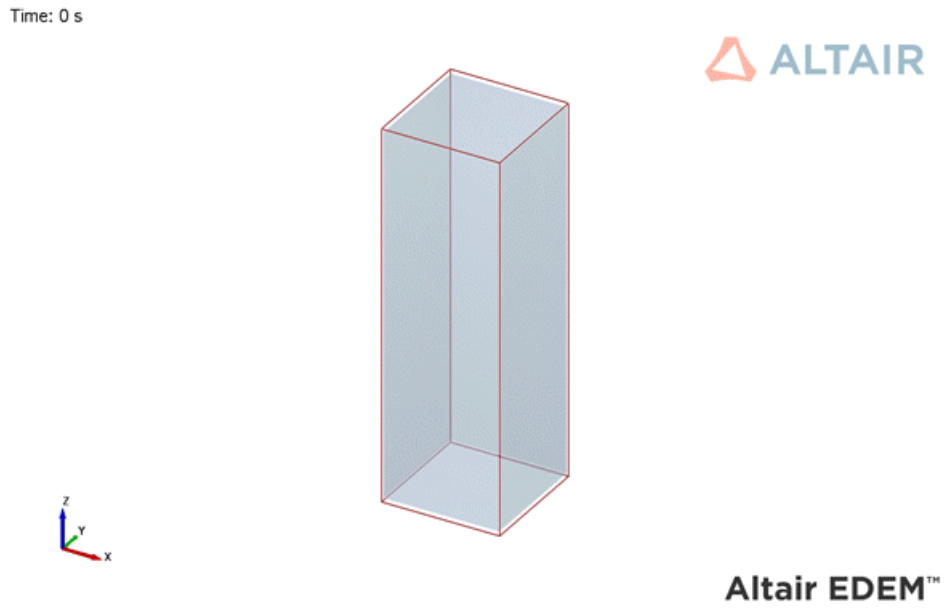


Figure 1586:

Review the Bulk Material and Particle Shape

In this step, you will define the material models for the bar and sphere-shaped particles.

1. In the Creator Tree, expand **Bulk Material** > **cylinder** > **New particle 1**.
2. Click **cylinder** and verify that the properties are set as shown below.

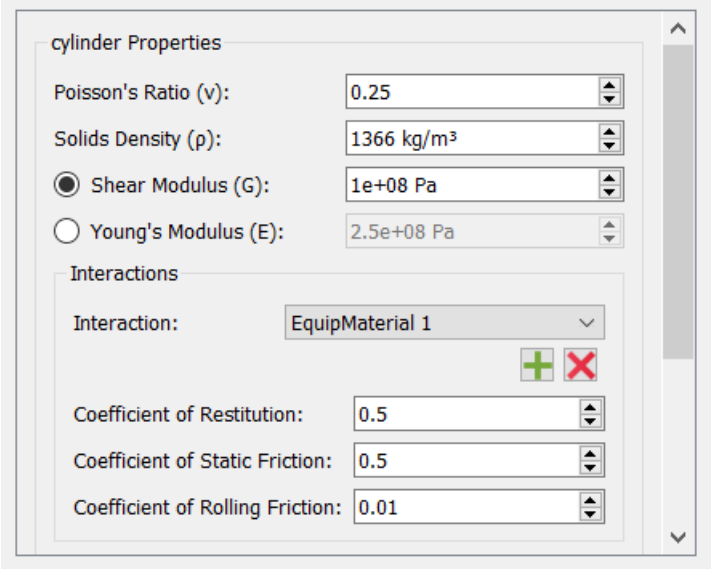


Figure 1587:

3. Click **Properties** under New particle 1.

Observe that the particle shape is created using multiple spheres aligned along the x-axis.

> **Note:** When defining non-spherical shapes, you need to make sure that the principal axis of the particle is aligned with the x-axis, as shown in the figure below.

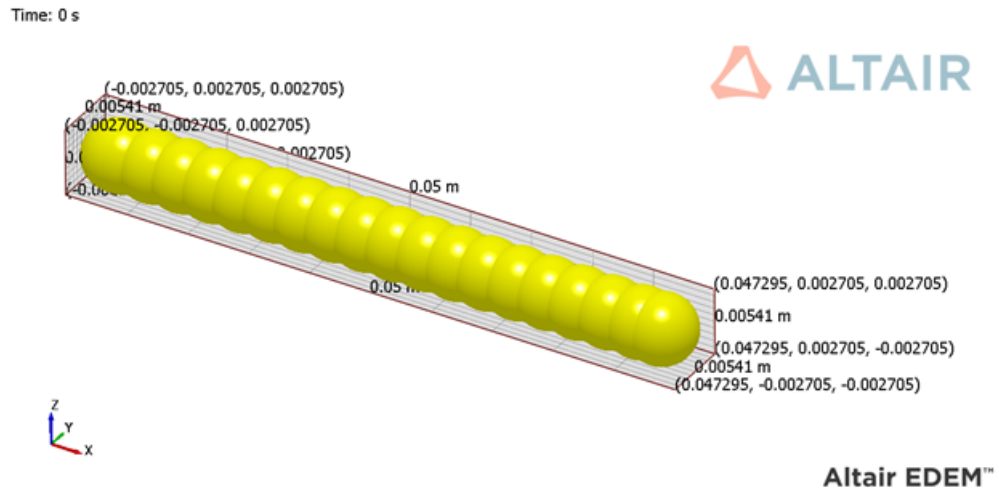


Figure 1588:

The length (L) and diameter (D) of the particle are 0.05 m and 0.00541 m respectively. Therefore, the aspect ratio (= L/D) of the particle is 9.242.

4. Observe the mass and volume of the particle in the Creator Tree.

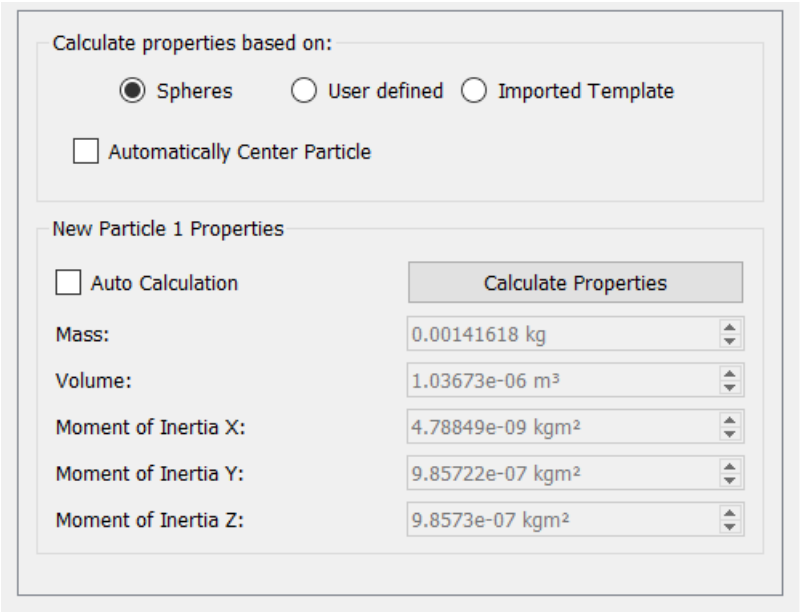


Figure 1589:

Define Geometries and Factories

1. In the Creator Tree, expand **Geometries**.
2. Click **Tank** and verify that the type is set to Physical.

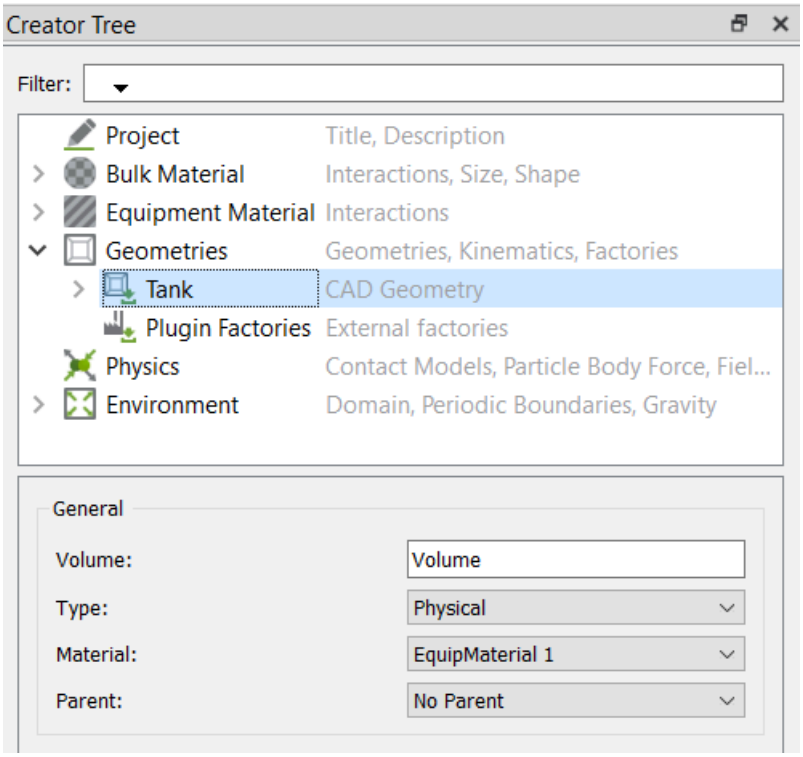


Figure 1590:

3. Right-click **Tank** and select **Add Factory**.
4. Right click **New Factory 1** and select **Change Factory Type**.
5. Verify that the Factory Type is set to static and set the particle generation parameters as shown in the figure below.

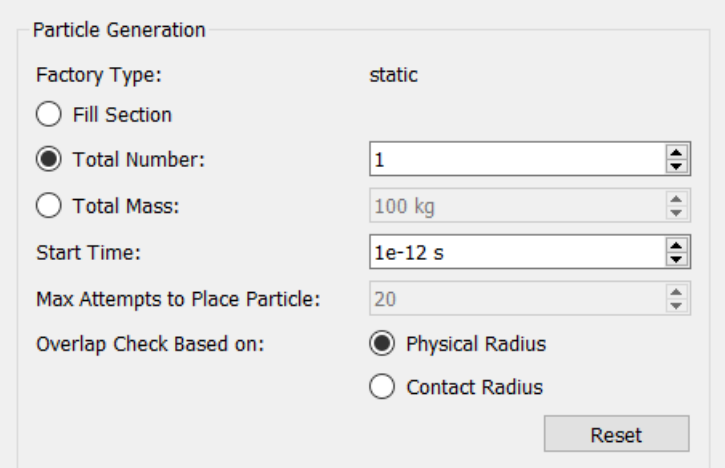


Figure 1591:

6. Under Parameters, set the Position option to **cubic** and click ⚙.

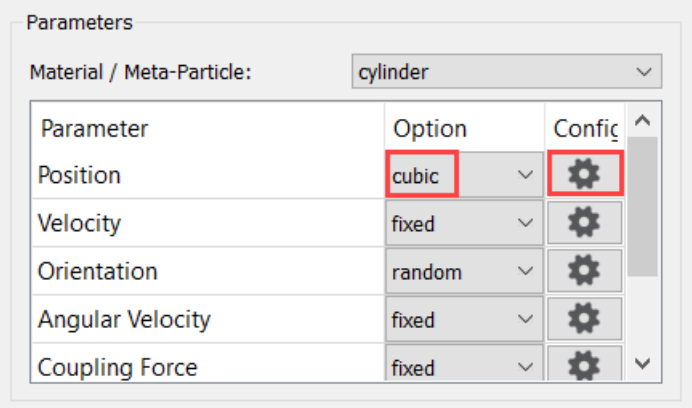


Figure 1592:

7. In the dialog, enter the position parameters as shown in the figure below and then click **OK**.

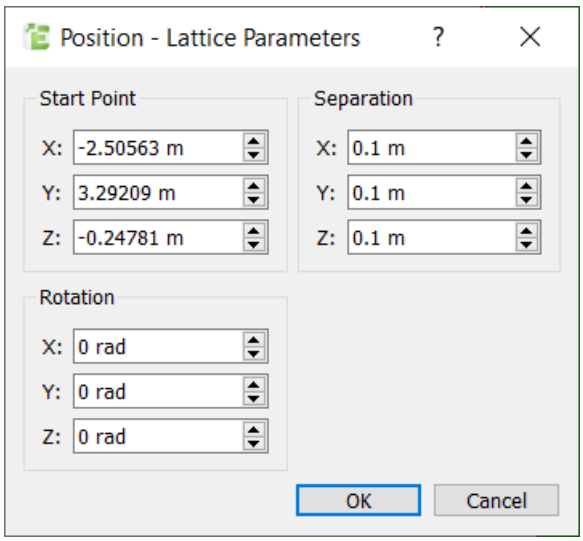


Figure 1593:

8. Under Parameters, set the Orientation option to **fixed** and click .
9. Enter the values for the orientation matrix as shown in the figure below.

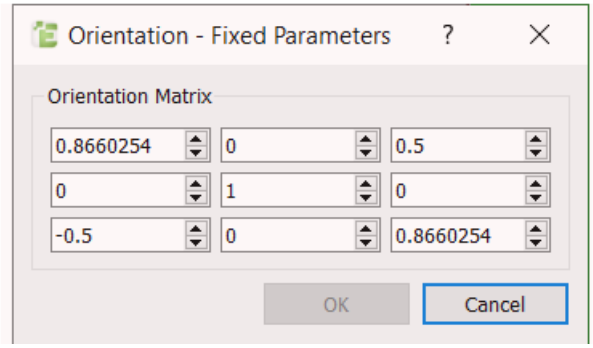


Figure 1594:

10. Click **OK**.

Define the Environment

In this step, you will define the extents of the domain for the EDEM simulation and the direction of gravitational acceleration.

1. In the Creator Tree, click **Environment**.
2. Activate the checkbox for **Auto Update from Geometry** (if not already selected).
   When a moving particle touches the bounding faces of the domain (environment), it will be removed from the simulation.
3. Activate **Gravity** and set the z-value to `-9.81` m/s^2.
4. Save the EDEM deck.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and de-activate the **Auto Time Step** checkbox.
3. Set the Fixed Time Step to `1e-5` s.

   Note: Generally, a value of 20-40% of the Rayleigh Time Step is recommended as the time step size to ensure stability of the DEM simulation.
4. Set the Total Time to `5` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `2.5` R min.

   Generally, a value in the range of 2-6 R_{min} is recommended as the optimum cell size.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

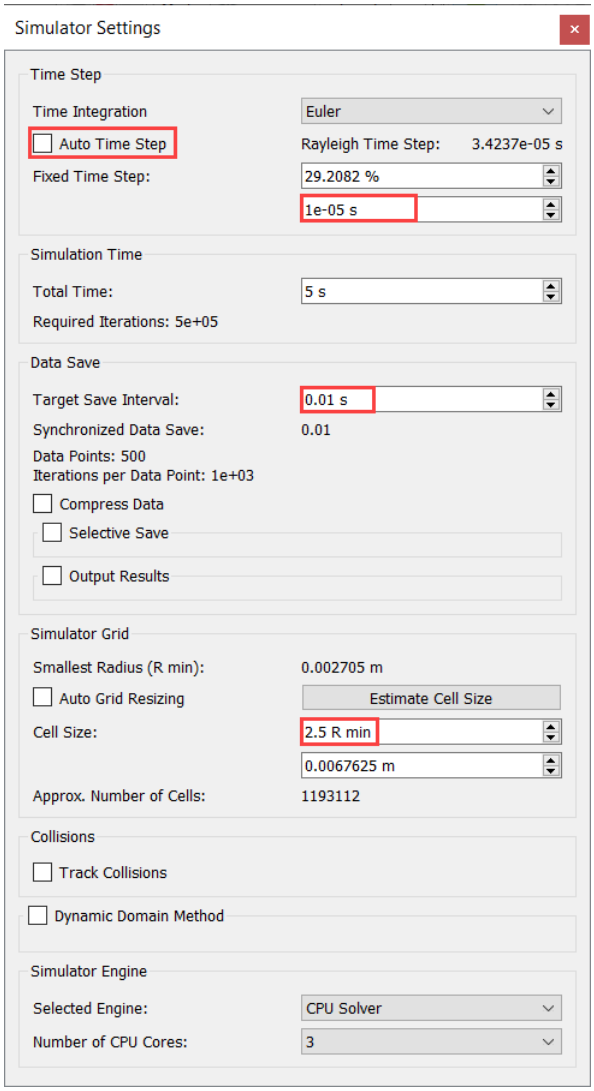


Figure 1595:

7. Once the simulation settings have been defined, save the model.

Part 2 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1596:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6105_cylinder.hm` and click **Open**.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1597:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

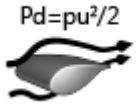


Figure 1598:

The **Setup** dialog opens.

2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.

5. In the Material Library dialog, select **EDEM 2 Way Multiphase**, switch to the **My Materials** tab, then click + to add a new material model.
6. In the microdialog, click **EDEM Bidirectional Material** in the top-left corner and change the name to `Water-Particle`.
7. Set the Carrier field to **Water**.
8. Set the Particle Shape to **Non-Spherical**.
9. Set the Drag model to **Rong**.
10. Set the Non spherical drag coefficient model to **Ganser**.
11. Click the table icon beside the drag coefficient model drop-down.

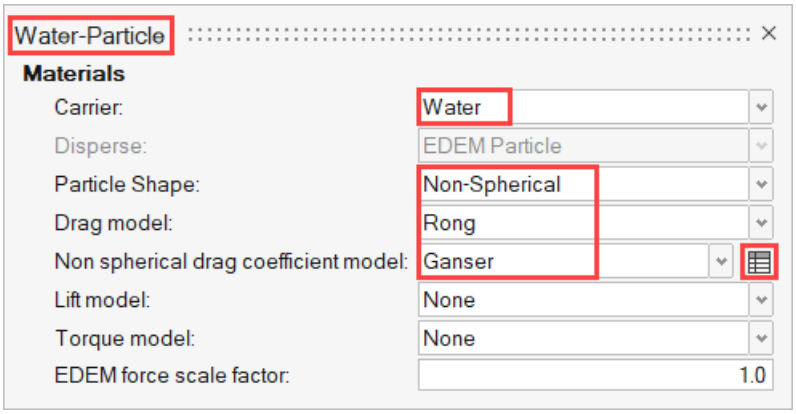


Figure 1599:

12. In the new dialog, enter the values for Aspect Ratio and Volume as shown in the figure below.

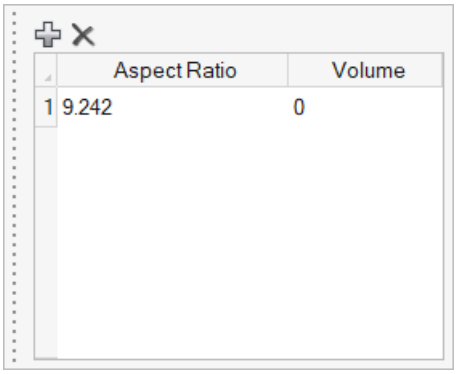


Figure 1600:

> **Note:** Since you have only a single type of particle in this simulation, you don't need to specify the volume of the particle. But, in the case of multiple particles, the volume of each particle should be specified, which can be obtained from EDEM.

13. Press **Esc** to close the dialog.
14. Set the Lift model to **Nonspherical lift**.
15. Set the Torque model to **Pitching Rotational Torque**.

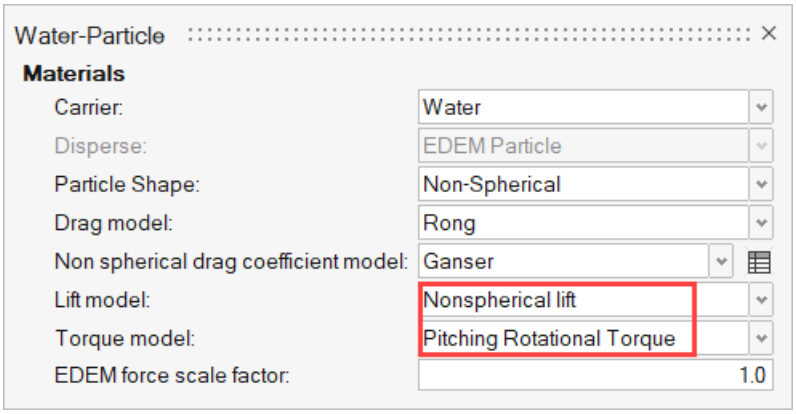


Figure 1601:

16. Close the material model and Material Library dialogs.
17. In the Setup dialog, set the Eulerian Material to **Water-Particle**.
18. Set Time step size and Final time to `0.001` and `5`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
19. Verify that Gravity is set to `0, 0, -9.81`.
20. Set the Pressure scale to **Absolute**.

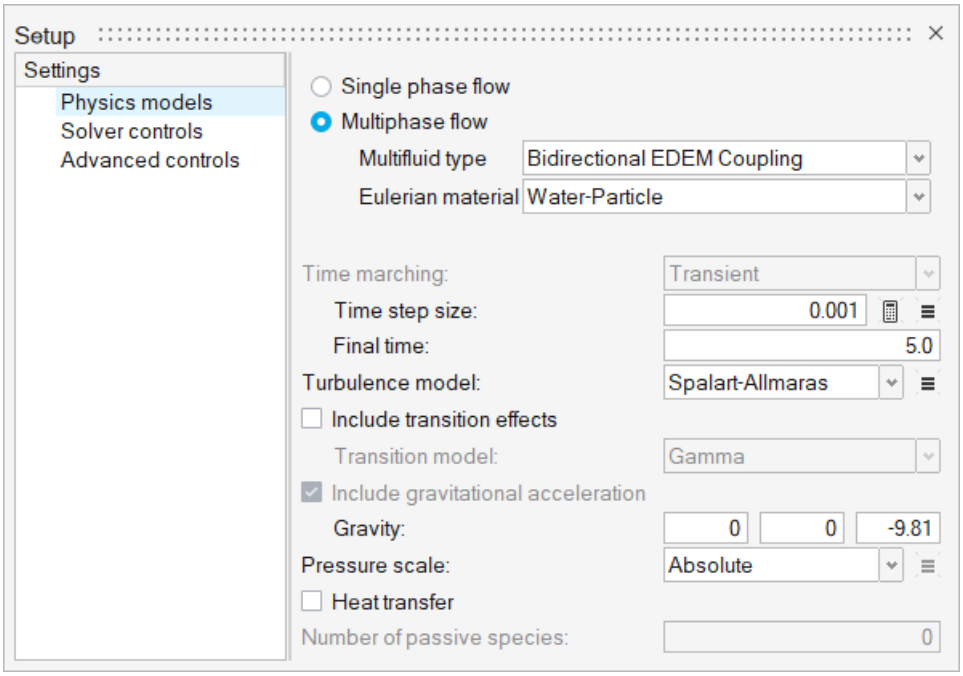


Figure 1602:

21. Click the **Solver controls** setting and set the Maximum stagger iterations to `2`.

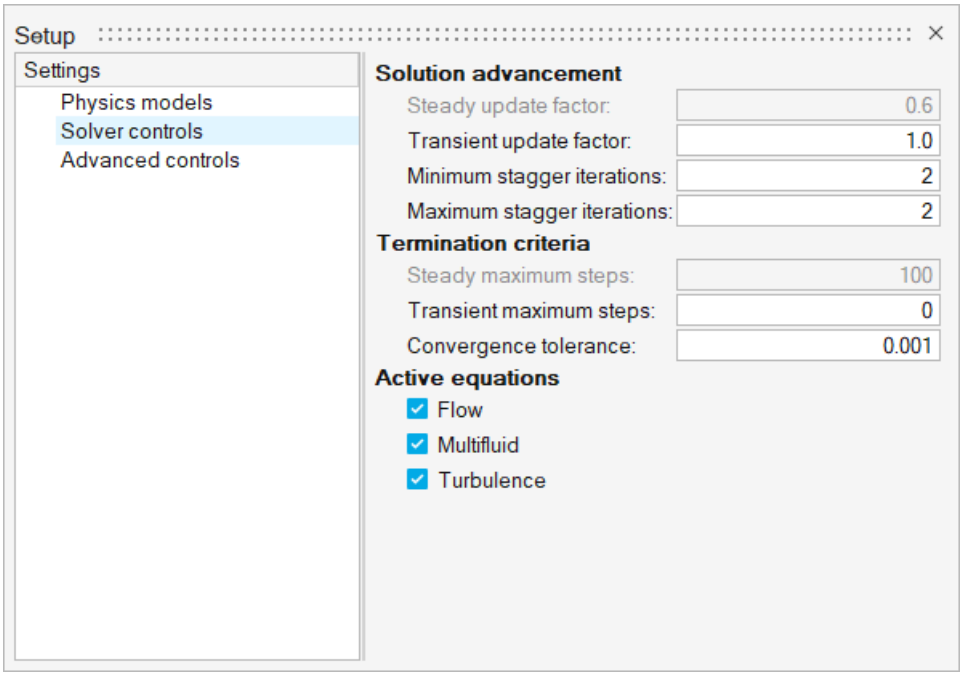


Figure 1603:

22. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1604:

2. Verify that Water-Particle has been assigned as the material.
If not assigned, click the tank geometry and select **Water-Particle** from the microdialog.

3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

Since the domain doesn't have any openings, there is no need to explicitly define a surface boundary condition as HyperMesh CFD assigns a no-slip wall boundary condition to all the surfaces by default.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

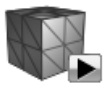


Figure 1605:

The **Meshing Operations** dialog opens.

2. Set the Mesh size to **Average size** and change the Maximum element size to `0.07`.
3. Deactivate **Curvature-based surface refinement** then click **Mesh**.

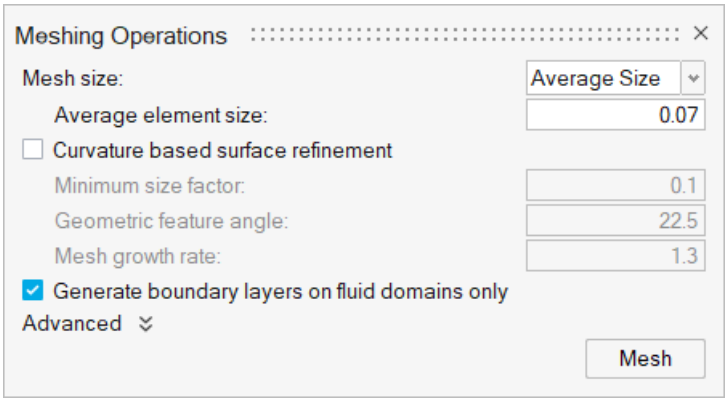


Figure 1606:

4. Once the meshing process is complete, save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

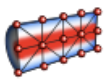


Figure 1607:

The **Field Output** dialog opens.

2. Check the box for **Write initial conditions**.
3. Uncheck the box for **Write results at time step interval**
4. Check the box for **Write results at time interval**.
5. Set the Time step interval to `0.01`.

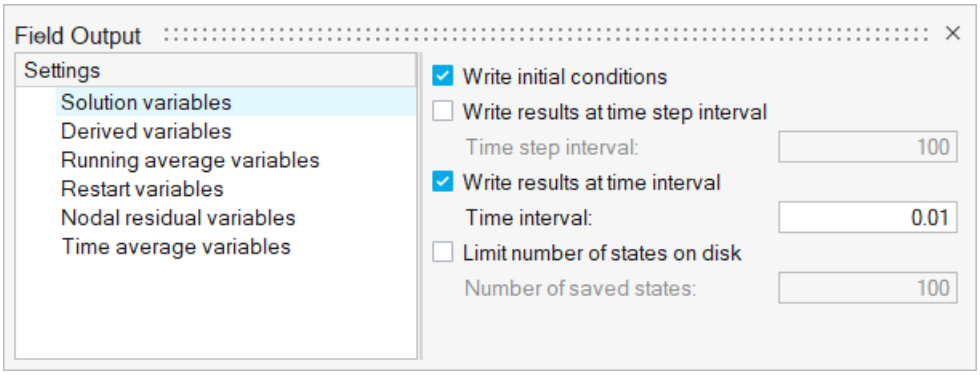


Figure 1608:

6. Close the dialog and save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1609:

Once the Coupling server is activated, the icon changes.

Figure 1610:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1611:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Set the Number of processors to `6`.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
7. Click **Run** to launch AcuSolve.

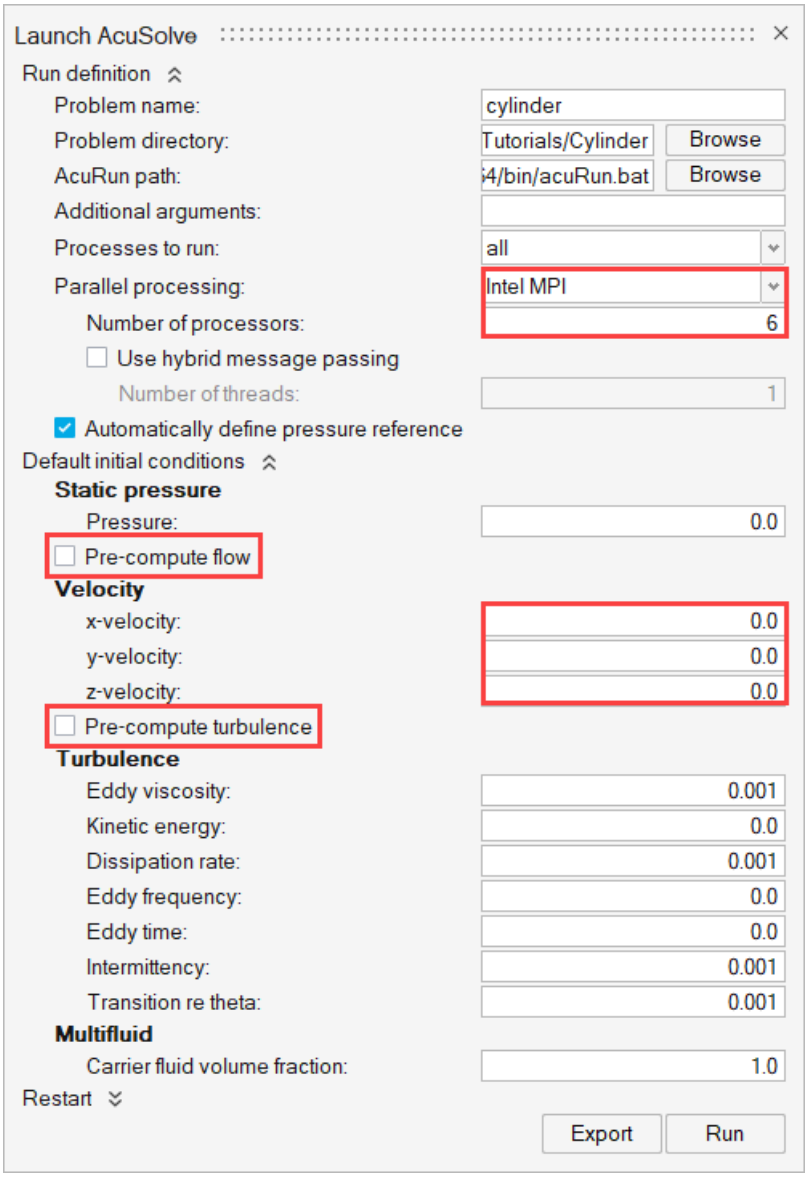


Figure 1612:

Once the AcuSolve run is launched, the **Run Status** dialog opens.

8. In the dialog, right-click the AcuSolve run and select **View log file**.

   If the coupling with EDEM is successful, that information is printed in the `.log` file.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 2.359000e+000 2.208940e+002 Sec
acuSolve: Input Memory Usage        = 6.900000e+001 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Interface thickness is 9.223856e-002
acuSolve: Init. CPU/Elapse time     = 2.820000e-001 8.700000e-002 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 1.000000e-003 ; time= 0.000000e+000
```

Figure 1613:

Once the simulation is complete, the summary of the run time is printed at the end of the `.log` file.

```
acuSolve: ---------- End Time Step ----------
acuSolve: ---------- Offset Values ----------
acuSolve:          Pressure Offset = 0.000000e+000
acuSolve:       Temperature Offset = 0.000000e+000
acuSolve: ------------------------------------------------------------
acuSolve: Input      CPU/Elapse time= 2.641000e+000 2.209810e+002 Sec
acuSolve: Output     CPU/Elapse time= 1.982587e+003 3.613650e+002 Sec
acuSolve: Exchange   CPU/Elapse time= 5.049100e+002 1.273880e+002 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =       10000       10000
acuSolve:   Elem.Form CPU/Elapse time= 1.942282e+003 3.444490e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 7.915960e+002 1.433350e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =       10000       10000
acuSolve:   Elem.Form CPU/Elapse time= 1.209830e+003 2.107650e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 2.255740e+002 4.124800e+001 Sec
acuSolve: Edem stagger "edem":
acuSolve:   No of external code calls=       5000
acuSolve:   EDEM ext. code CPU/Elapse time= 5.583740e+002 7.534340e+002 Sec
acuSolve: Field-Velocity stagger "field_velocity: EDEM Particle":
acuSolve:   No of res/lhs formations =       5000       5000
acuSolve:   Elem.Form CPU/Elapse time= 1.157050e+003 2.231180e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.897960e+002 4.000700e+001 Sec
acuSolve: Field-Velocity stagger "field_velocity: Water":
acuSolve:   No of res/lhs formations =       5000         0
acuSolve:   Elem.Form CPU/Elapse time= 1.467000e+000 5.300000e-001 Sec
acuSolve:   Solution  CPU/Elapse time= 1.197560e+002 2.222100e+001 Sec
acuSolve: Fields stagger "EDEM Particle":
acuSolve:   No of res/lhs formations =       5000       5000
acuSolve:   Elem.Form CPU/Elapse time= 7.474840e+002 1.396540e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.396670e+002 2.790500e+001 Sec
acuSolve: Fields stagger "Water":
acuSolve:   No of res/lhs formations =       5000         0
acuSolve:   Elem.Form CPU/Elapse time= 2.357000e+000 4.130000e-001 Sec
acuSolve:   Solution  CPU/Elapse time= 1.055740e+002 2.113900e+001 Sec
acuSolve: LnSlv. (host)   Elapse time= 1.273900e+002 Sec
acuSolve: Total CPU/Elapse time     = 1.144244e+004 2.960139e+003 Sec
acuSolve: Total Memory Usage        = 9.100000e+001 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
```

Figure 1614:

Analyze the Results

AcuSolve Post-Processing

1. In HyperMesh CFD, right-click the AcuSolve run in the **Run Status** dialog and select **Visualize results**.

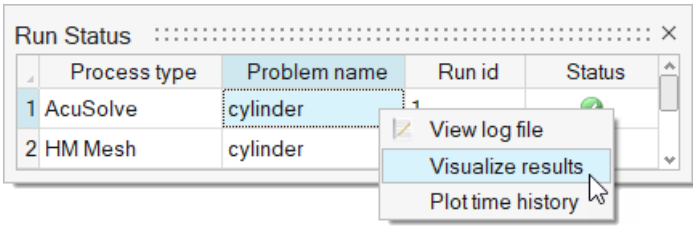


Figure 1615:

The results are loaded in the Post ribbon.

2. Click the **Slice Planes** tool.

Figure 1616:

3. Select the x-z plane as highlighted in the figure below.

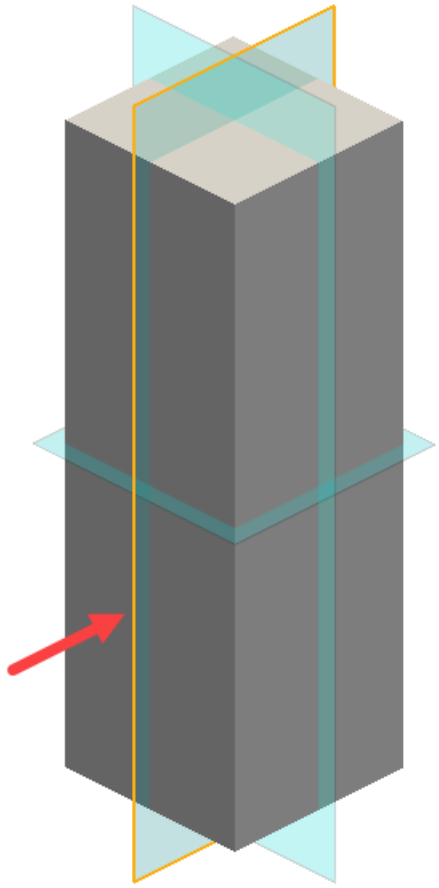


Figure 1617:

4. In the slice plane microdialog, click to create the slice plane.
5. In the display properties microdialog, set the display to **volume fraction edem particle** and activate the **Legend** toggle.
6. Change the bounds of the legend to `0` and `0.001`.
7. Click and set the Colormap name to **Rainbow Uniform**.

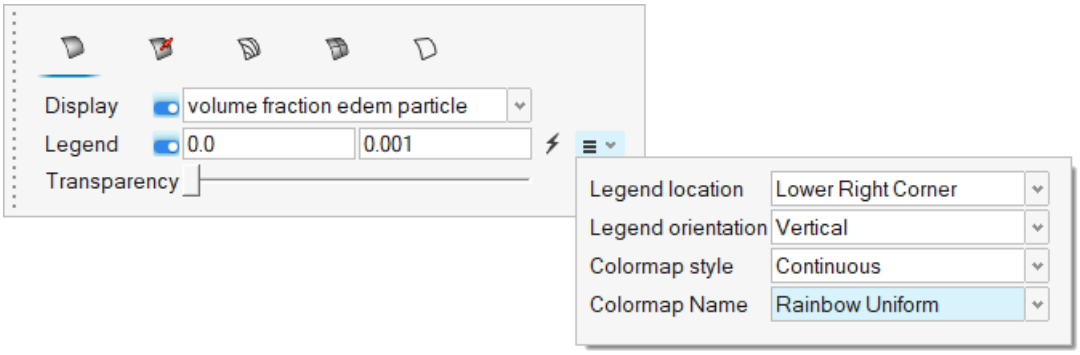


Figure 1618:

8. Click the **Vector Display** tab.
9. Activate the **Display** toggle and set the variable to **velocity**.
10. Set the size to `0.05` and the Color to **Constant**.
11. Click ≡ and set the Vector length to **Uniform**.

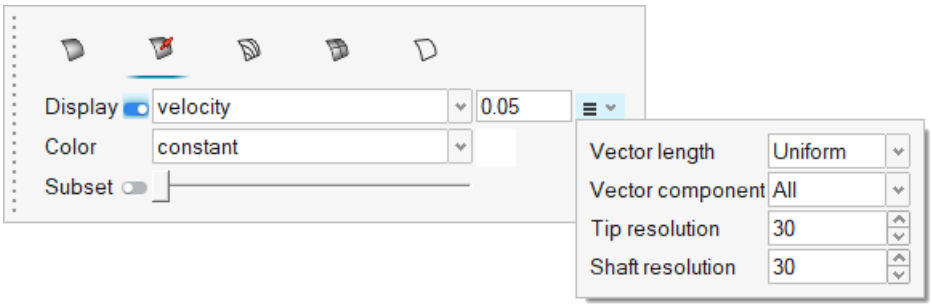


Figure 1619:

12. Click ✓ on the guide bar.
13. In the Post Browser, turn off the visibility of **Flow Boundaries** by clicking on its icon.

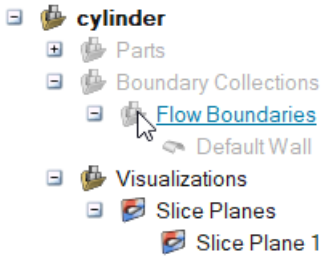


Figure 1620:

14. Select the **Left** face on the view cube to align the model to the x-z plane.

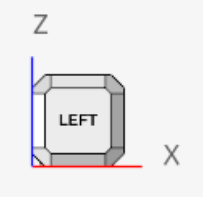


Figure 1621:

15. Click ▶ on the animation toolbar to view the animation.

Figure 1622:

EDEM Post-Processing

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then select **Tank**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

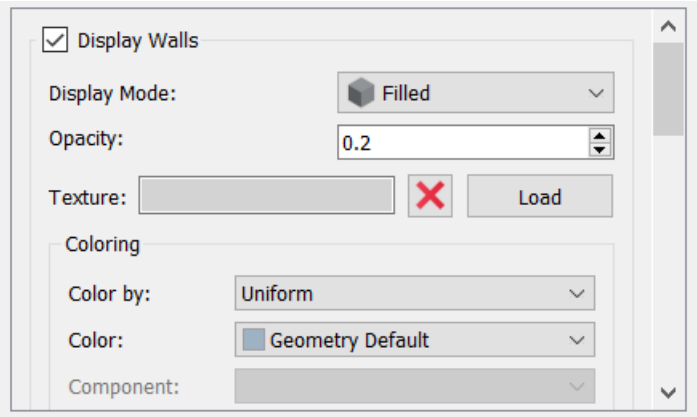


Figure 1623:

4. In the Analyst Tree, click **Particles**.
5. Set the coloring to **Contact Default**.
6. Click **Apply All**.

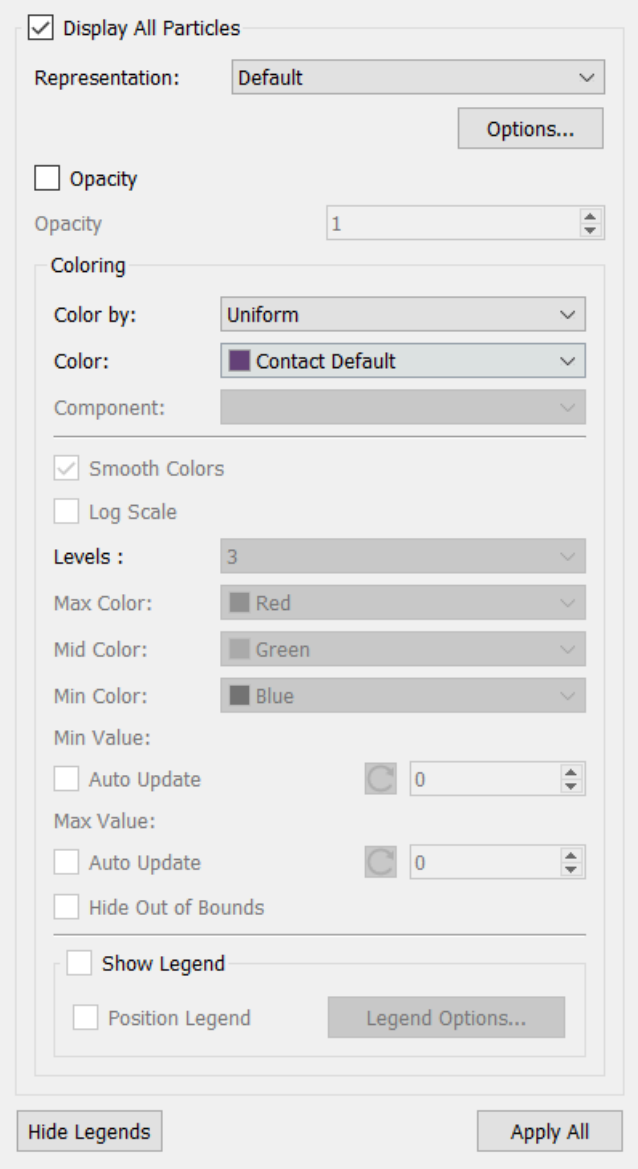


Figure 1624:

7. On the menu bar, set the time to `0` by clicking:

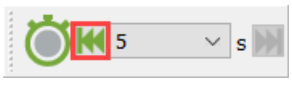


Figure 1625:

8. Set the View plane to **+ Y**.

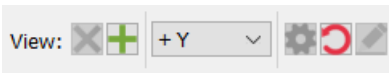


Figure 1626:

9. In the Viewer window, set the Playback Speed to `0.2x` and then click ▶ to play the particle flow animation.

   Figure 1627:

   Observe that the particle rotates and translates as it falls down. Without lift and torque, you will not see this phenomenon.

Summary

In this tutorial, you learned how to set up and run a basic AcuSolve-EDEM bidirectional (two-way) coupling problem with non-spherical particles. You learned how to create particles in EDEM with a specific orientation and position. Next, you set up the AcuSolve model to consider the effect of the lift and torque forces. Once the simulation is complete, you learned how to post-process the AcuSolve results using HyperMesh CFD and EDEM.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve.

This tutorial demonstrates the set up of a flow solution in SimLab. Before you begin, download the CAD geometry.

You will need an internet connection to watch the video tutorials and download the files.

Tutorial Video

https://fast.wistia.net/embed/iframe/x6hc7ffgar

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6106 AcuSolve - EDEM Bidirectional Coupling with Mass Transfer

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial introduces you to the workflow for setting up and running a basic AcuSolve-EDEM bidirectional coupling simulation with mass transfer. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction and HyperMesh CFD, and have a basic understanding of HyperMesh CFD, AcuSolve, and EDEM. To run this simulation, you will need access to a licensed version of HyperMesh CFD, AcuSolve, and EDEM.

Before you begin, copy the file(s) used in this tutorial to your working directory and unzip the files.

`ACU-T6106_tablet.hm`

`tablet_EDEM.zip`

Problem Description

The spray coating update physics models allows you to simulate the coating process in which the volume of a 'spray' particle is transferred to a target particle upon contact. In this tutorial, a tablet coating process is simulated, in which spray particles are sprayed onto a row of tablets. The spray particles are modeled as solid particles in EDEM, and their properties are set to that of water. When a spray particle touches the tablet, the volume of spray particle is transferred to the tablet and the amount of coating is registered as a custom property called 'Volume Added'.

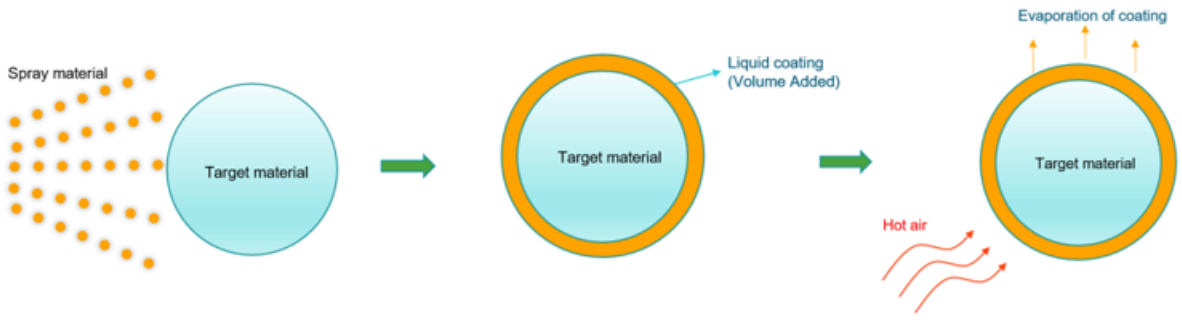


Figure 1628:

In a typical tablet coater, hot air is injected during the coating process. This allows for the solvent to be evaporated and the solid content of the coating gets deposited on the surface of the tablet. By coupling the spray coating model with the heat and mass transfer models in AcuSolve, you can simulate the drying process as well. In this tutorial, the initial temperature of the air is set to 363 K and the walls are considered as adiabatic. The tablet particles are created with an initial temperature of 350 K. The spray particles are injected for the first 0.25 s of the simulation and then the coating is left to evaporate.
In addition to the Volume Added (coating volume), the evaporation rate on each particle can also be monitored.

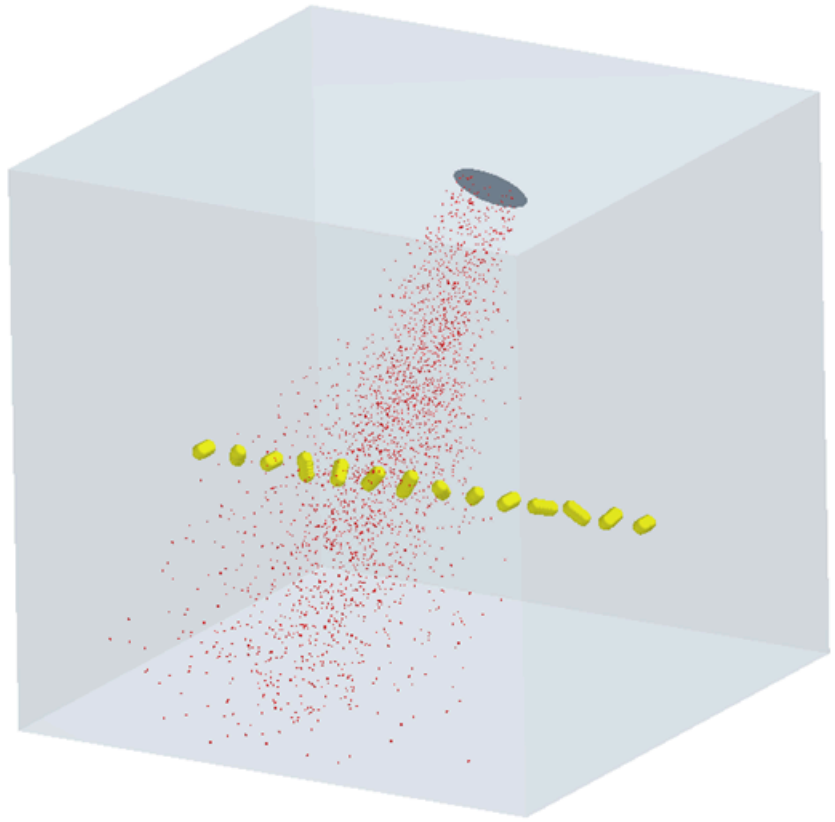


Figure 1629:

Some assumptions of the spray coating model:

- There is no momentum exchange between the spray material and the target material or the geometry. Although, there is momentum exchange between the spray particles and the carrier gas.
- The spray particles do not exchange thermal energy with the carrier gas or other particles/ geometries.
- The coating on a target particle is assumed to be transferred instantaneously and uniformly across the surface of the particle.
- The spray particles do not interact with each other. Hence, there is no coalescence or breakup of spray droplets.

Part 1 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

1. From the menu bar, click **File** > **Open**.
2. In the dialog, browse to your problem directory and open the `tablet.dem` file.
   The geometry and the materials are loaded.

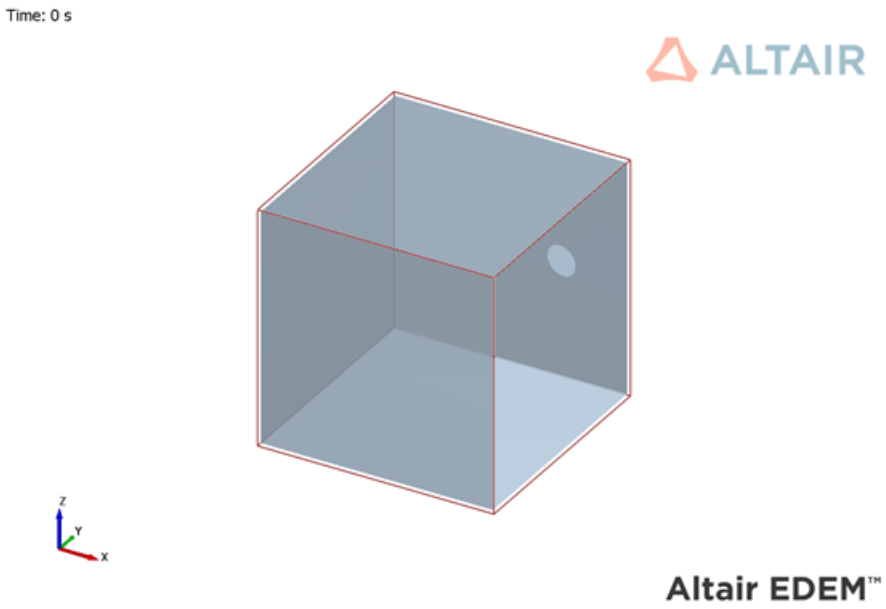


Figure 1630:

Review the Bulk Material and Particle Shape

1. Right-click in the Creator Tree and select **Expand All**.
2. Under Bulk Material, click **spray** and verify that the properties are set as shown below.

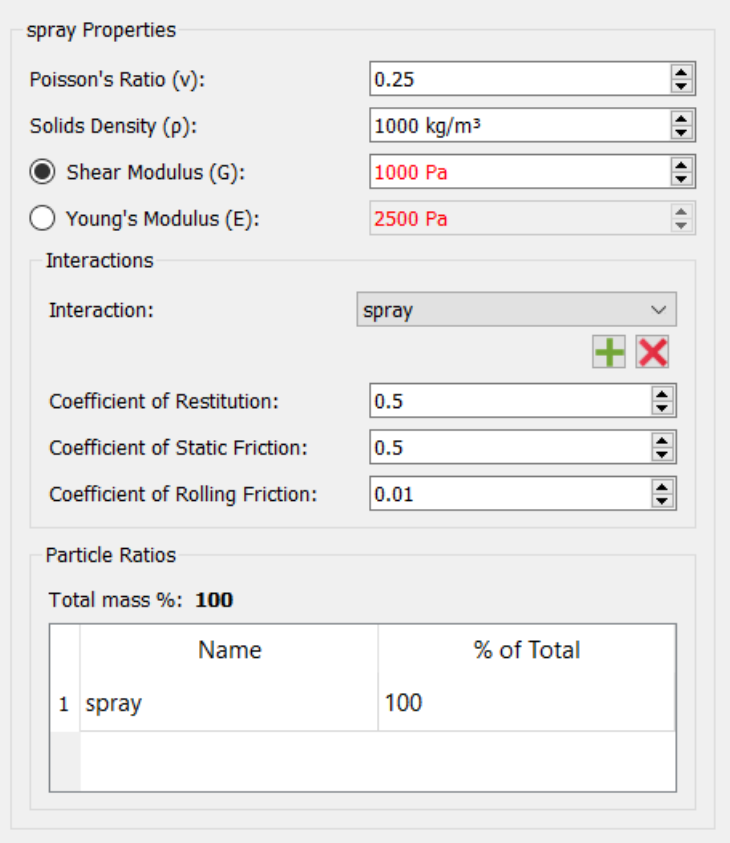


Figure 1631:

3. Click **Properties** under spray. Verify that the particle radius is set to `0.0001` m.
4. Click the **tablet** bulk material and verify that the properties are set as shown below.

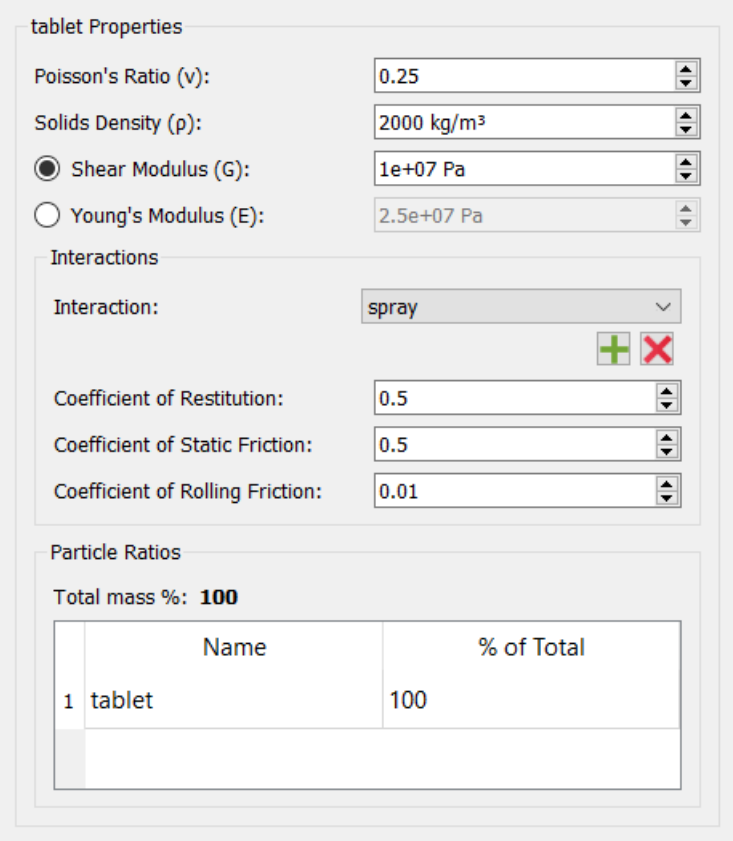


Figure 1632:

5. Click **Properties** under tablet and review the tablet shape.

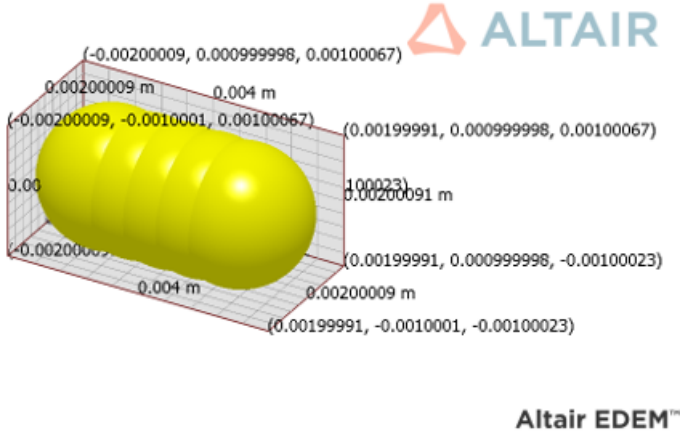


Figure 1633:

The coordinate digit in the figure might be slightly different from those in the database.

Set the Physics Models

1. In the Creator Tree, click **Physics**.
2. Set the Interaction to **Particle to Particle** then click **Edit Contact Chain**.

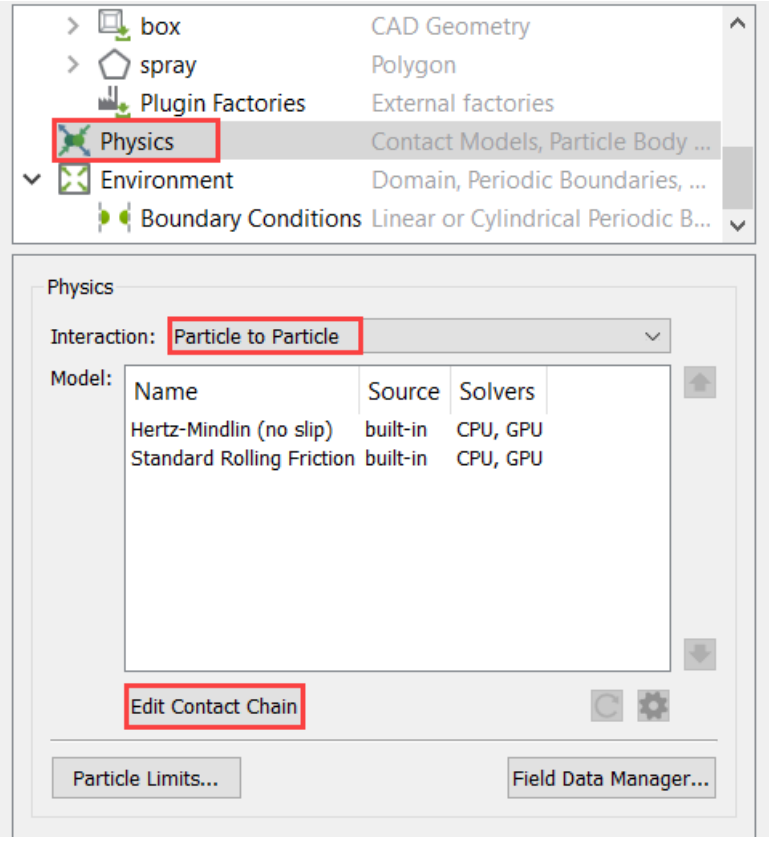


Figure 1634:

3. In the dialog, activate the checkbox for the **Spray Coating** model. Set the Base and Friction models as shown in the figure below and click **OK**.

   The tutorial is set up in such a way that the tablet particles do not interact with each other. Hence, the base contact models can be turned off for a faster turnaround time.

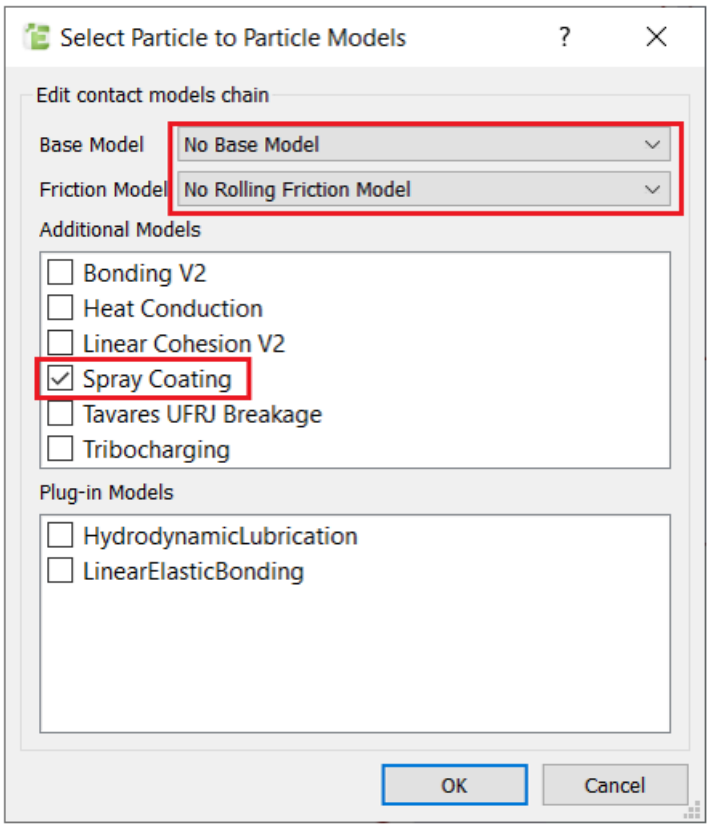


Figure 1635:

4. In the Creator Tree, select **Spray Coating** and then click .

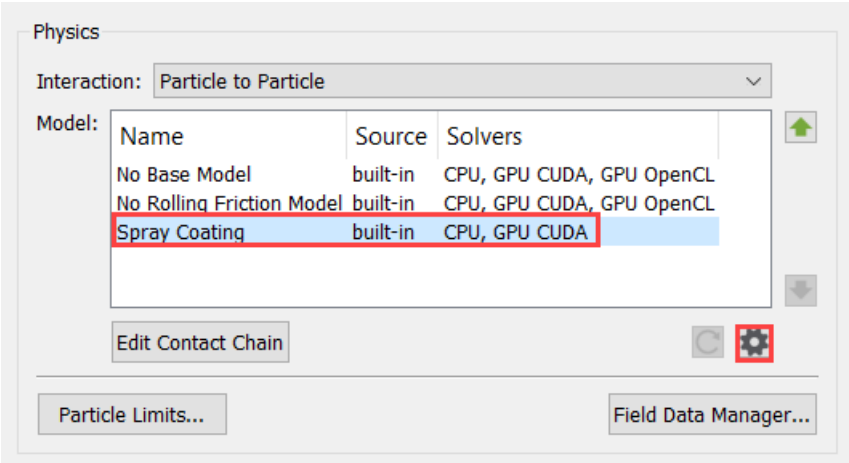


Figure 1636:

5. In the **Spray Coating Model Configuration** dialog, click , select **spray** from the list of materials, and then click **OK**.

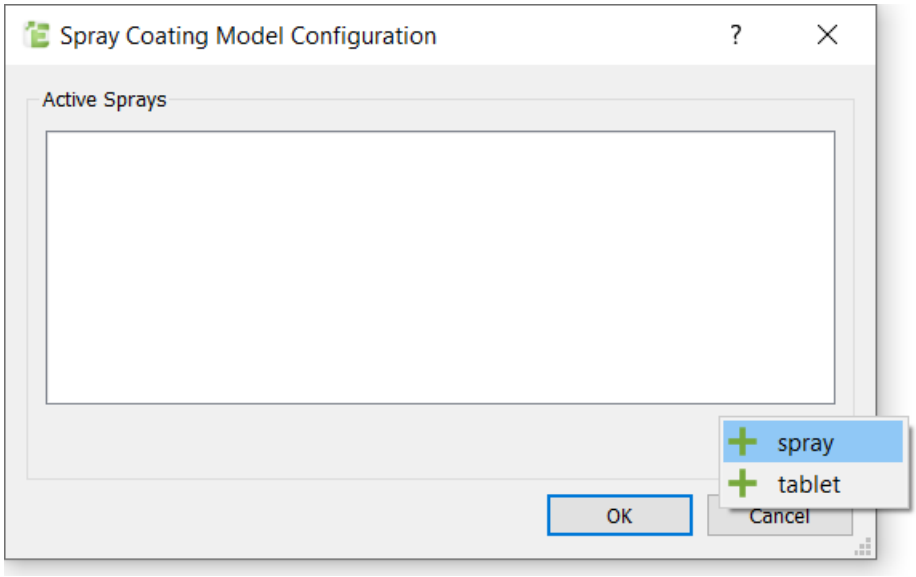


Figure 1637:

The active spray material should be defined in both the 'Particle to Particle' and 'Particle to Geometry' interactions to ensure that the simulations stay stable even when the DEM time step is chosen based on the size of target material.

6. In the Creator Tree, change the interaction to **Particle to Geometry** and then click **Edit Contact Chain**.
7. In the dialog, set the same parameters as above and then click **OK**.

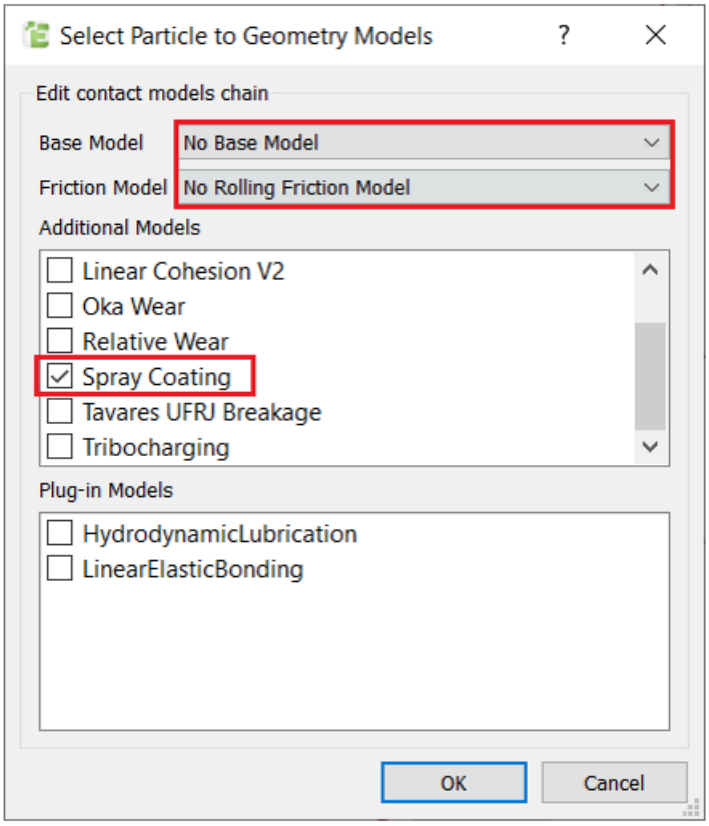


Figure 1638:

8. In the Creator Tree, select **Spray Coating** and then click .

9. In the **Spray Coating Model Configuration** dialog, click , select **spray** from the list of materials, and then click **OK**.

   Adding a material to the Particle to Geometry spray model will mean that when a particle from this material contacts geometry, there will be no force applied to the particle or geometry. This is important to stop unstable behavior when running with a Time Step appropriate for the target material.

10. In the Creator Tree, change the interaction to **Particle Body Force** and then click **Edit Contact Chain**.

    Note: In this simple example, the tablet particles are created such that there are no tablet-tablet and tablet-geometry contacts. Hence the contact models are turned off to save computational time. The spray-tablet contacts are still active and are handled by the spray coating physics model.

11. In the dialog, activate the checkboxes for **Temperature Update** and **Spray Coating Update** and then click **OK**.

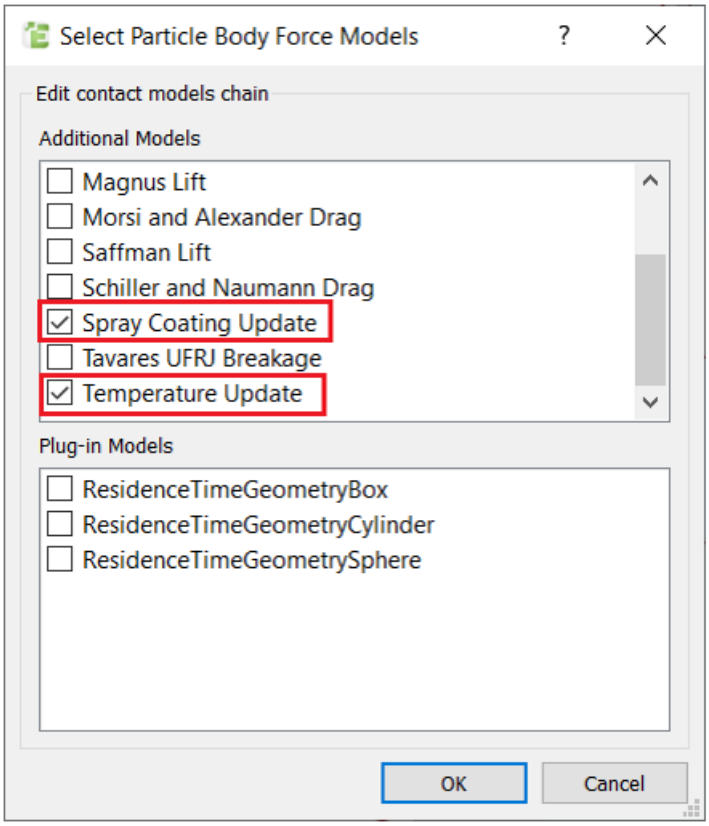


Figure 1639:

By including the spray coating model in the particle body force, the target material's volume is updated. Without this, only the target material's Volume Added property will increase.

12. In the Creator Tree, select **Temperature Update** and then click .

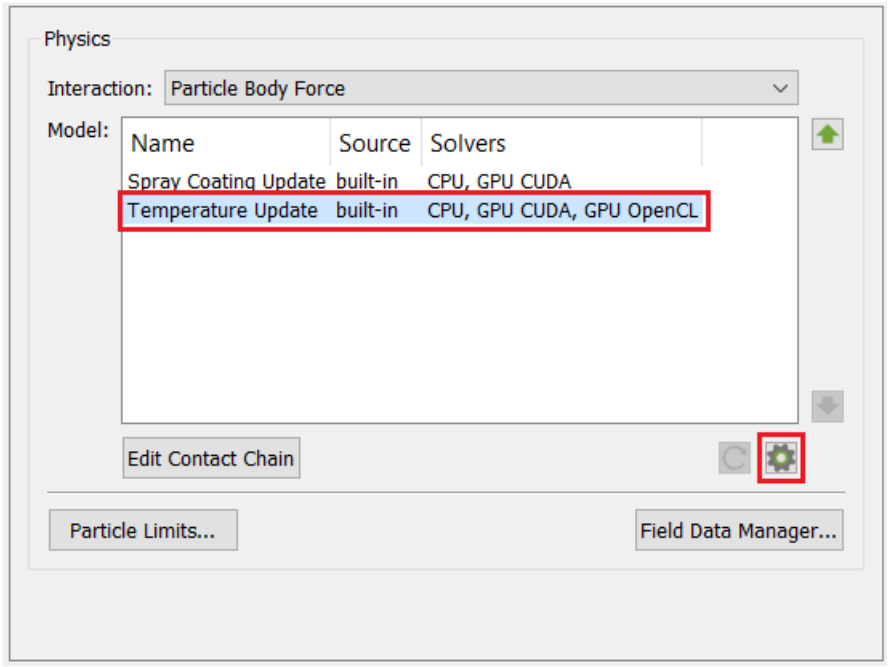


Figure 1640:

13. Set the specific heat capacity of tablet and spray to `1200` and `4000` J/kg-K, respectively, and then click **OK**.
14. Save the model.

Define Geometries and Factories

1. Under Geometries, click **box** and verify that the type is set to **Virtual**.
2. Right-click **box** and select **Add Factory** > **Add Static Factory**.

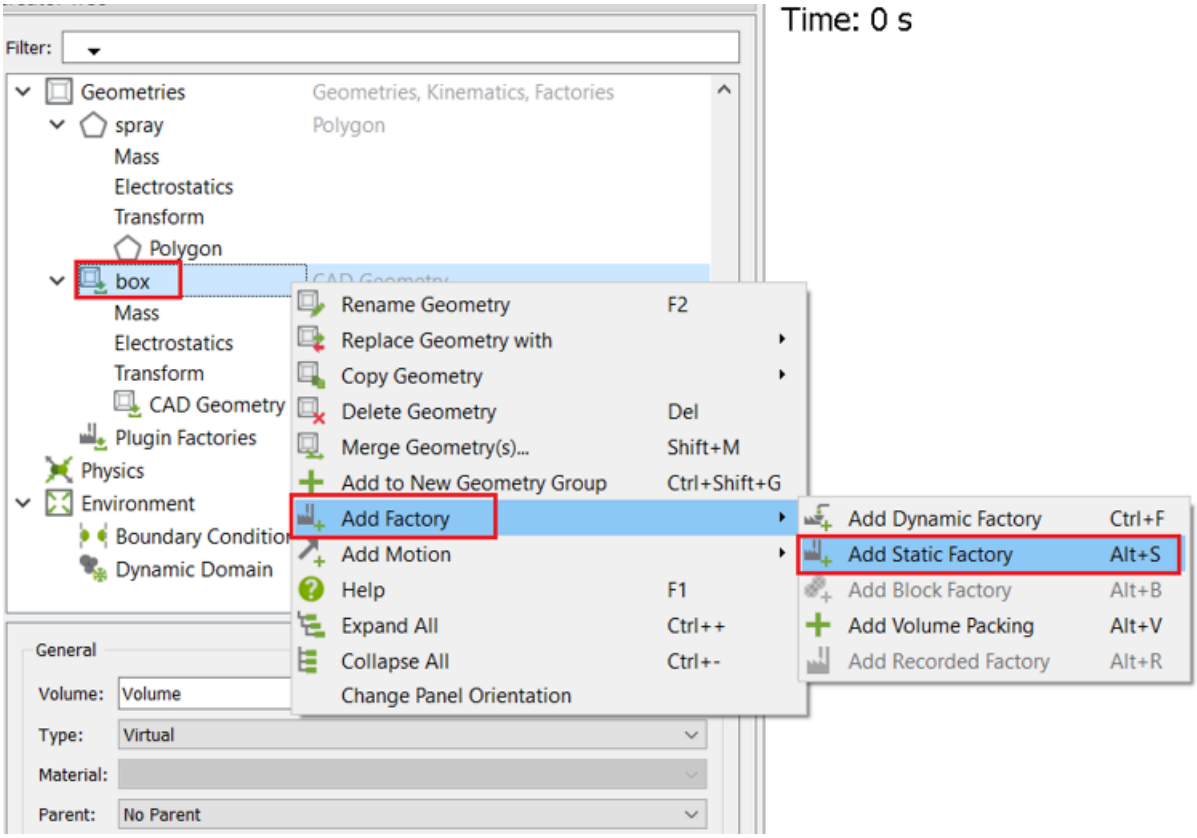


Figure 1641:

3. Set the particle generation parameters as shown in the figure below. Make sure that the Material/ Meta-Particle is set to **tablet**.

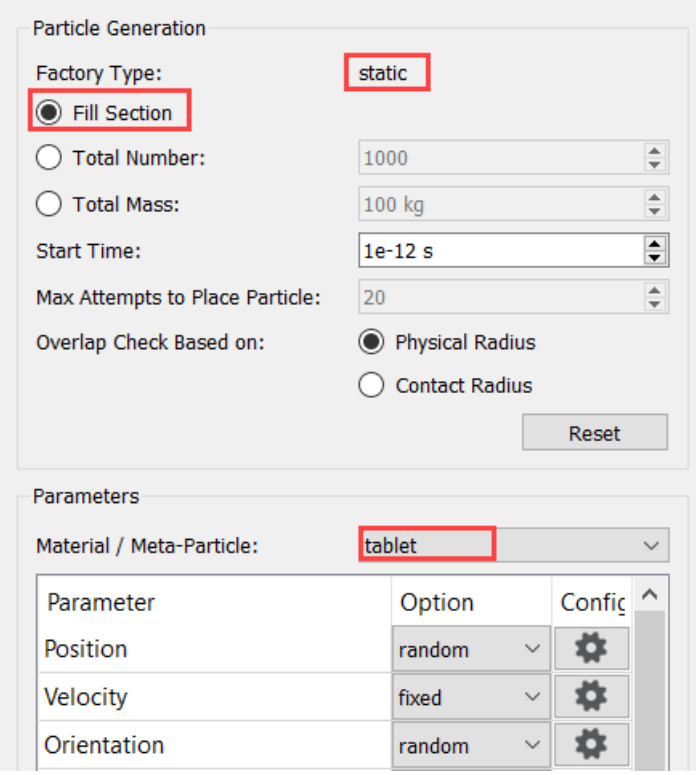


Figure 1642:

4. Under Parameters, set the Position option to **cubic** and then click ⚙.

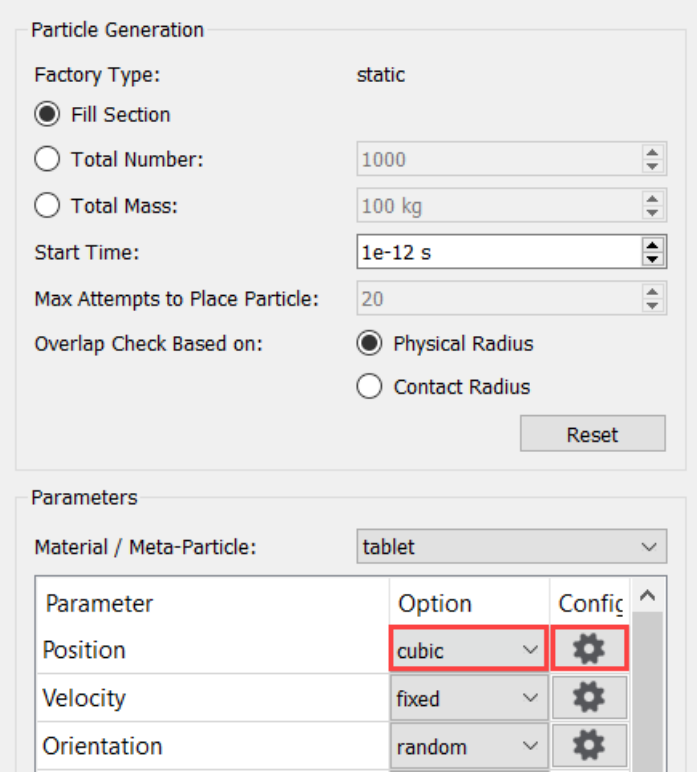


Figure 1643:

5. In the dialog, enter the position parameters as shown in the figure below and then click **OK**. This creates a row of tablet particles spaced evenly along the y-axis and located at the center of the box.

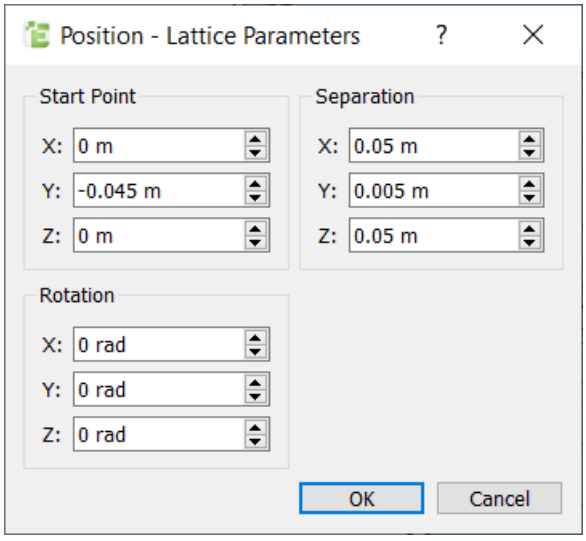


Figure 1644:

6. Similarly, click next to the Temperature setting, set the particle temperature to `350` K, and then click **OK**.

7. Under Geometries, click **spray** and then change the type to **Virtual**.
8. Right-click **spray** and select **Add Factory** > **Add Dynamic Factory**.
9. Set the particle generation parameters as shown in the figure below. Set the Material to **spray**.

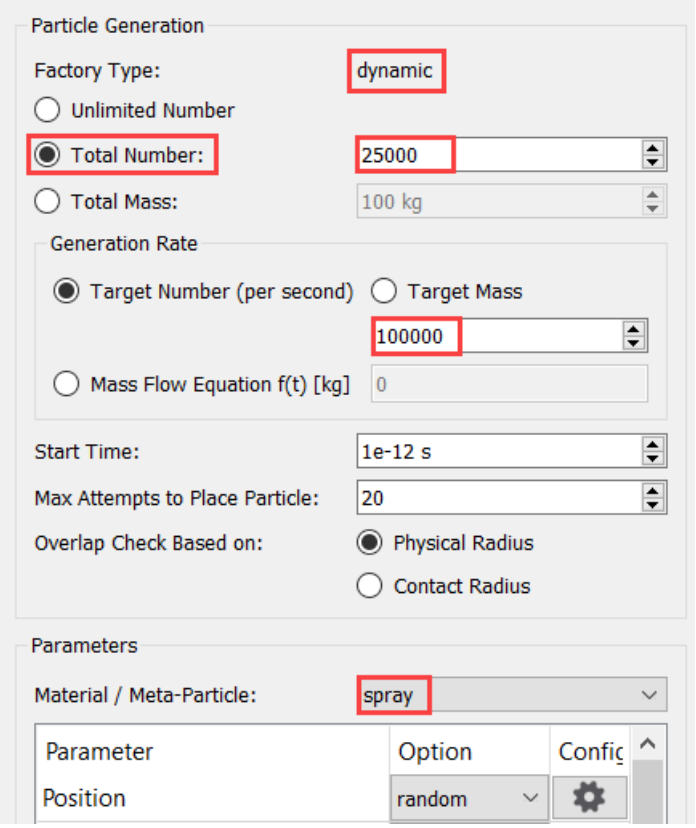


Figure 1645:

10. Under Parameters, set the Velocity option to **spray** and then click ⚙.
11. In the dialog, set the spray velocity parameters as shown in the figure below and then click **OK**.

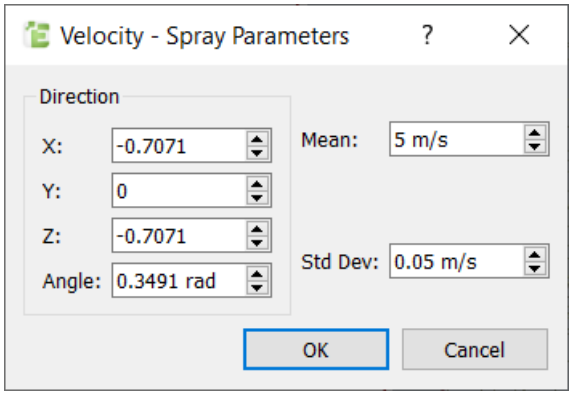


Figure 1646:

12. Similarly, click next to the Temperature setting, set the particle temperature to `350` K, and then click **OK**.
13. Save the model.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and de-activate the **Auto Time Step** checkbox.
3. Set the Fixed Time Step to `1e-5` s.

   Note: Generally, a value of 20-40% of the Rayleigh Time Step is recommended as the time step size to ensure stability of the DEM simulation. Spray particles do not impart force on either geometry or any other particles. This means you can run the simulation with the appropriate time step for the target material.
4. Set the Total Time to `0.5` s and the Target Save Interval to `0.01` s.
5. Set the Cell Size to `50` R min.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

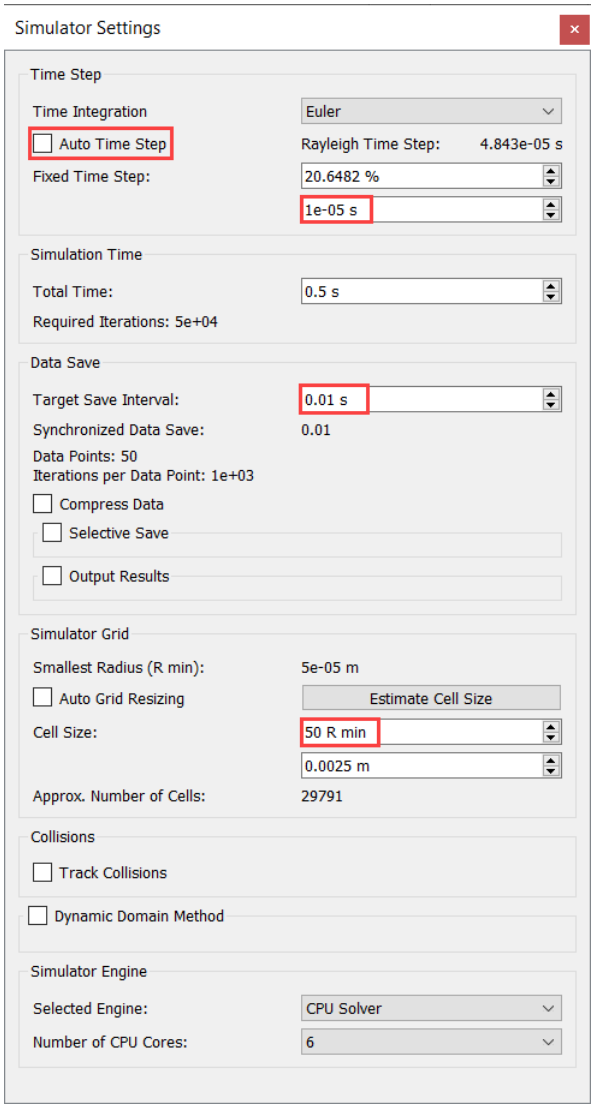


Figure 1647:

7. Once the simulation settings have been defined, save the model.

Part 2 - AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1648:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6106_tablet.hm` and click **Open**.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1649:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

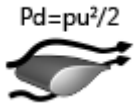


Figure 1650:

The **Setup** dialog opens.

2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Set Time step size and Final time to `0.001` and `0.5`, respectively. Select **Spalart-Allmaras** for the Turbulence model.
5. Set the Pressure scale to **Absolute**.

6. Activate **Heat transfer** and set the Temperature scale to **Absolute**.
7. Set the Number of passive species to `1`.

   This setting is necessary for the mass transfer feature to be active. The evaporated moisture is tracked using the species transport equation.

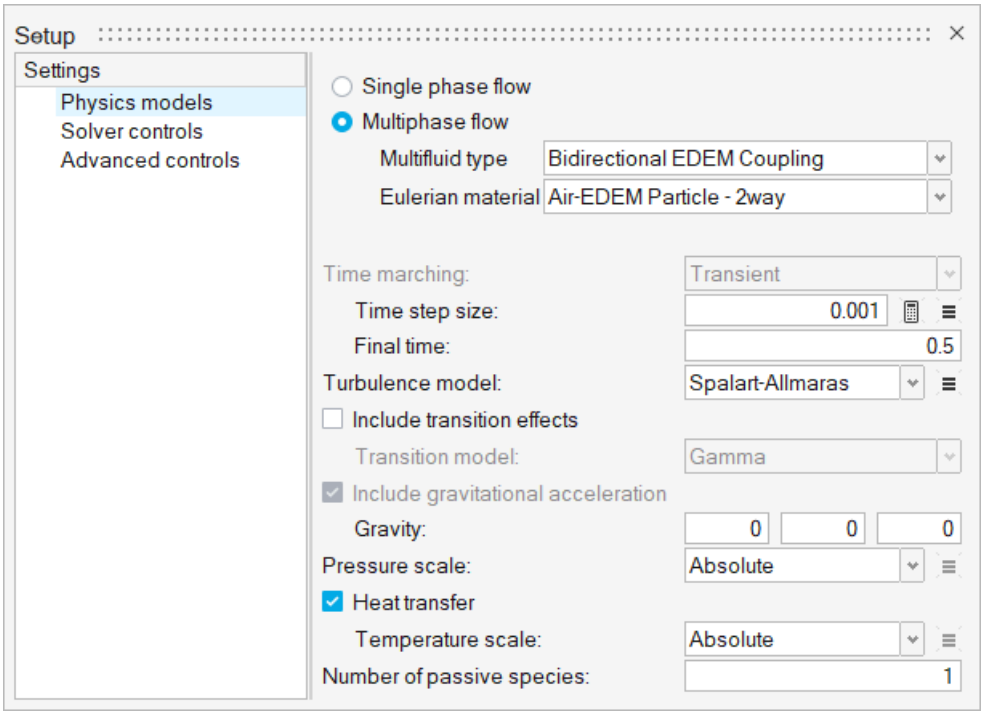


Figure 1651:

8. Click the **Solver controls** setting. Set the Minimum and Maximum stagger iterations to `2` and `2`, respectively.

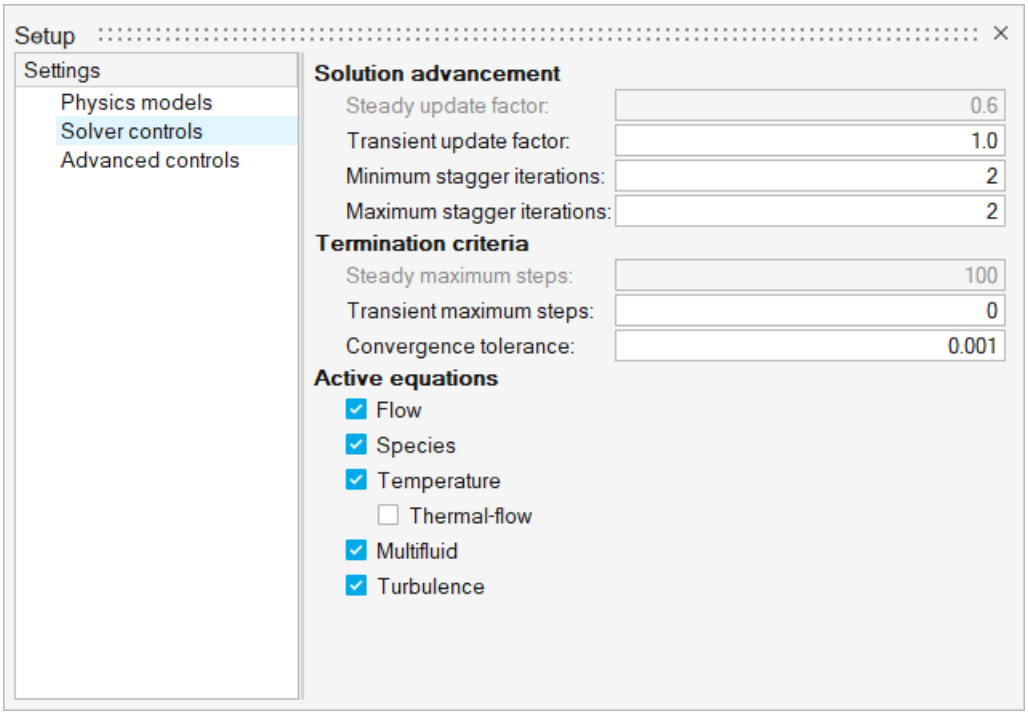


Figure 1652:

9. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1653:

2. Verify that Air-EDEM Particle – 2way has been assigned as the material.
   If not assigned, click the box geometry and select **Air-EDEM Particle – 2way** from the microdialog.
3. On the guide bar, click ✓ to exit the tool.
   Since you are applying an adiabatic condition on the box walls, you don't need to set any thermal boundary condition explicitly.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

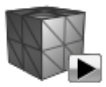


Figure 1654:

The **Meshing Operations** dialog opens.

2. Set the Mesh size to **Average size** and change the Maximum element size to `0.005`.
3. Deactivate **Curvature-based surface refinement** and then click **Mesh**.

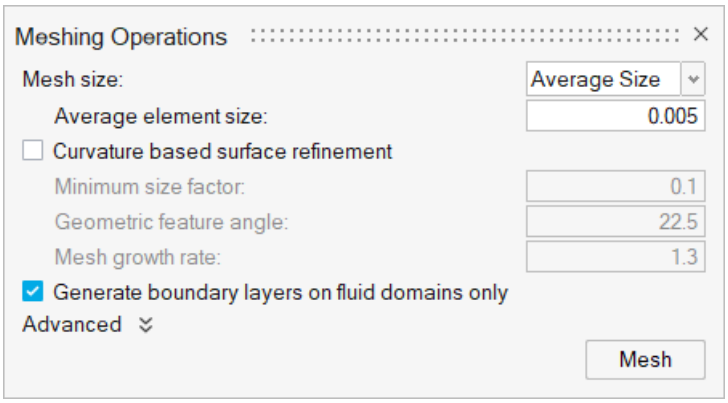


Figure 1655:

4. Once the meshing process is complete, save the model.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

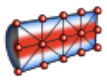


Figure 1656:

The **Field Output** dialog opens.

2. Check the box for **Write initial conditions**.
3. Uncheck the box for **Write results at time step interval**
4. Check the box for **Write results at time interval**.
5. Set the Time step interval to `0.01`.

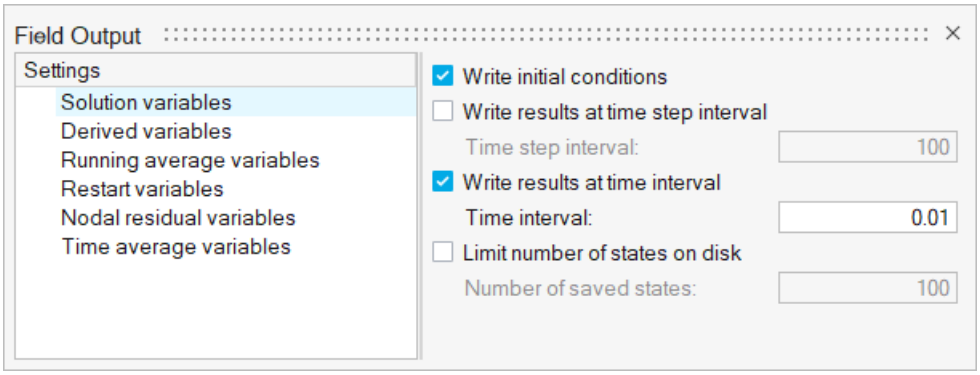


Figure 1657:

6. Close the dialog and save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1658:

Once the Coupling server is activated, the icon changes.

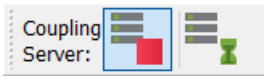


Figure 1659:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1660:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Set the Number of processors to `6`.
6. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
7. Set the Temperature to `363` K.

8. Click **Run** to launch AcuSolve.

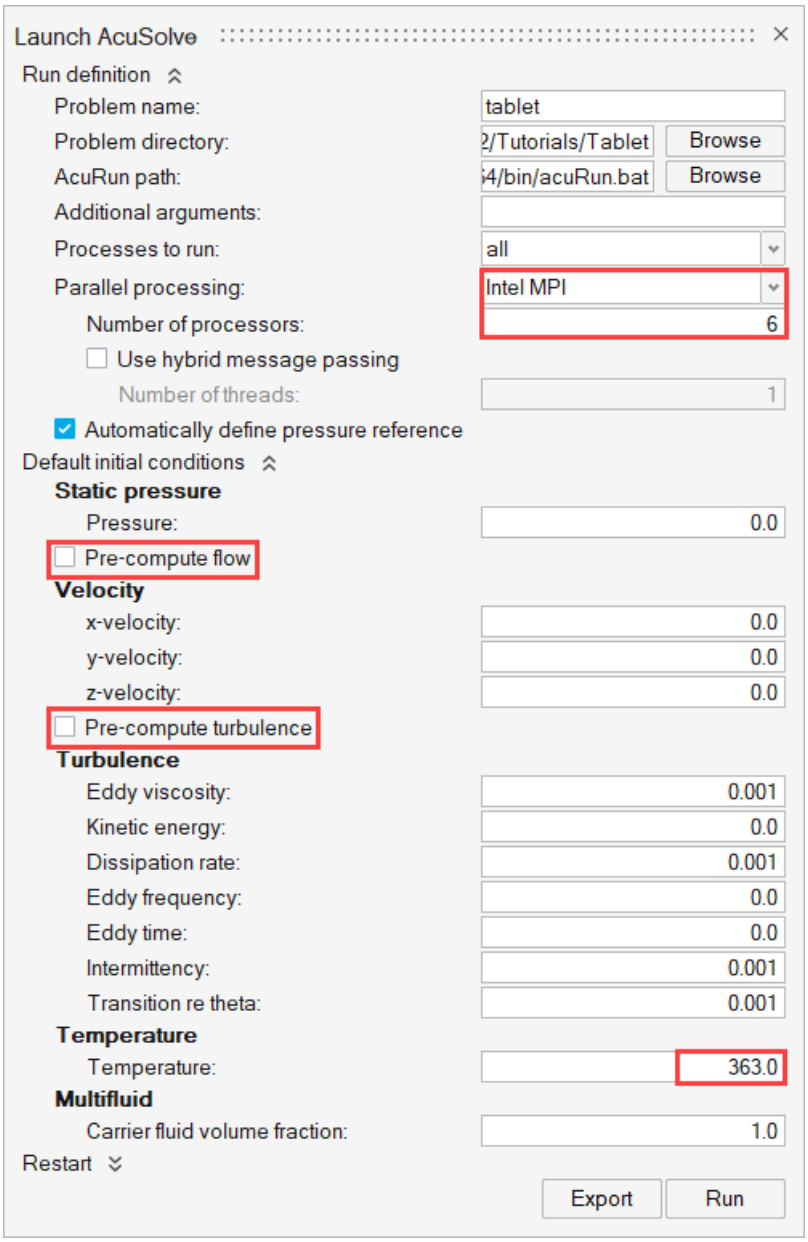


Figure 1661:

Once the AcuSolve simulation is launched, it connects with the EDEM coupling server. A warning message is displayed indicating that the base contact models have not been set. Click **Ignore**. Since the tablet particles are not in contact with each other or with the geometry, you don't need any contact models. The spray-tablet contacts are handled by the spray coating model.

9. In the dialog, right-click on the AcuSolve run and select **View log file**.

If the coupling with EDEM is successful, that information is printed in the log file.

```
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 2.468000e+000 1.525240e+002 Sec
acuSolve: Input Memory Usage        = 1.010000e+002 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Temperature in coupling client registered
acuSolve: VolumeAdded in coupling client registered
acuSolve: Interface thickness is 6.597853e-003
acuSolve: Init. CPU/Elapse time     = 4.700000e-001 1.950000e-001 Sec
acuSolve: ---------- Begin Time Step ----------
acuSolve: Time-Step=    1 ; timeInc= 1.000000e-003 ; time= 0.000000e+000
```

Figure 1662:

Once the simulation is complete, the summary of the run time is printed at the end of the log file.

```
acuSolve: ---------- End Time Step ----------
acuSolve: ---------- Offset Values ----------
acuSolve:          Pressure Offset = 0.000000e+000
acuSolve:       Temperature Offset = 0.000000e+000
acuSolve: ------------------------------------------------------------
acuSolve: Input      CPU/Elapse time= 2.938000e+000 1.527190e+002 Sec
acuSolve: Output     CPU/Elapse time= 7.327820e+002 1.245020e+002 Sec
acuSolve: Exchange   CPU/Elapse time= 9.265900e+001 1.984300e+001 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:  No of res/lhs formations =       1000       1000
acuSolve:  Elem.Form CPU/Elapse time= 5.604260e+002 9.499200e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 1.918390e+002 3.344200e+001 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:  No of res/lhs formations =       1000       1000
acuSolve:  Elem.Form CPU/Elapse time= 3.834400e+002 6.485400e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 5.808600e+001 1.007300e+001 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:  No of res/lhs formations =       1000       1000
acuSolve:  Elem.Form CPU/Elapse time= 3.961310e+002 6.705200e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 8.005700e+001 1.379900e+001 Sec
acuSolve: Species stagger "species_1":
acuSolve:  No of res/lhs formations =       1000       1000
acuSolve:  Elem.Form CPU/Elapse time= 3.642110e+002 6.245200e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 6.051100e+001 1.055900e+001 Sec
acuSolve: Edem stagger "edem":
acuSolve:  No of external code calls=        500
acuSolve:  EDEM ext. code CPU/Elapse time= 6.686970e+002 8.735990e+002 Sec
acuSolve: Field-Velocity stagger "field_velocity: EDEM Particle":
acuSolve:  No of res/lhs formations =        500        500
acuSolve:  Elem.Form CPU/Elapse time= 3.420670e+002 6.097900e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 4.405500e+001 7.762000e+000 Sec
acuSolve: Field-Velocity stagger "field_velocity: Air":
acuSolve:  No of res/lhs formations =        500          0
acuSolve:  Elem.Form CPU/Elapse time= 3.430000e-001 4.700000e-002 Sec
acuSolve:  Solution  CPU/Elapse time= 2.603500e+001 4.615000e+000 Sec
acuSolve: Fields stagger "EDEM Particle":
acuSolve:  No of res/lhs formations =        500        500
acuSolve:  Elem.Form CPU/Elapse time= 2.100420e+002 3.544400e+001 Sec
acuSolve:  Solution  CPU/Elapse time= 2.754100e+001 4.907000e+000 Sec
acuSolve: Fields stagger "Air":
acuSolve:  No of res/lhs formations =        500          0
acuSolve:  Elem.Form CPU/Elapse time= 1.870000e-001 2.700000e-002 Sec
acuSolve:  Solution  CPU/Elapse time= 2.351600e+001 4.029000e+000 Sec
acuSolve: LnSlv. (host)  Elapse time= 3.547300e+001 Sec
acuSolve: Total CPU/Elapse time     = 5.294125e+003 1.816717e+003 Sec
acuSolve: Total Memory Usage        = 1.400000e+002 Mbytes
acuRun: -*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
```

Figure 1663:

Post-Process the Results with EDEM

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. In the Analyst Tree, expand **Display** > **Geometries** and then select **box**.
3. Verify that the Display Mode is set to **Filled** and set the Opacity to `0.2`.

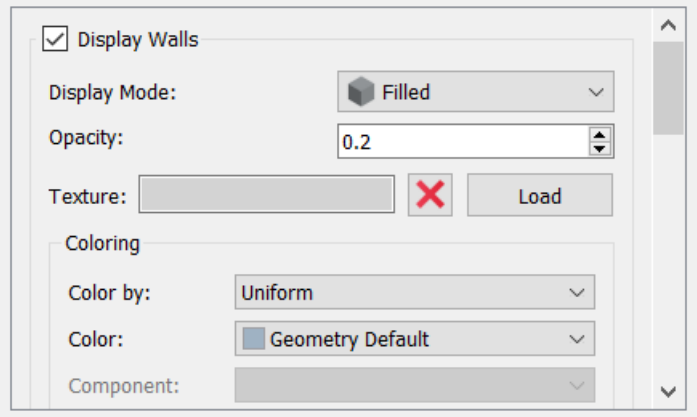


Figure 1664:

4. In the Analyst Tree, expand **Particles** and then click **tablet**.
5. Set the coloring to **Volume Added**.
6. Activate the **Auto Update** checkbox for both Min and Max Value.
7. Activate the **Show Legend** checkbox.

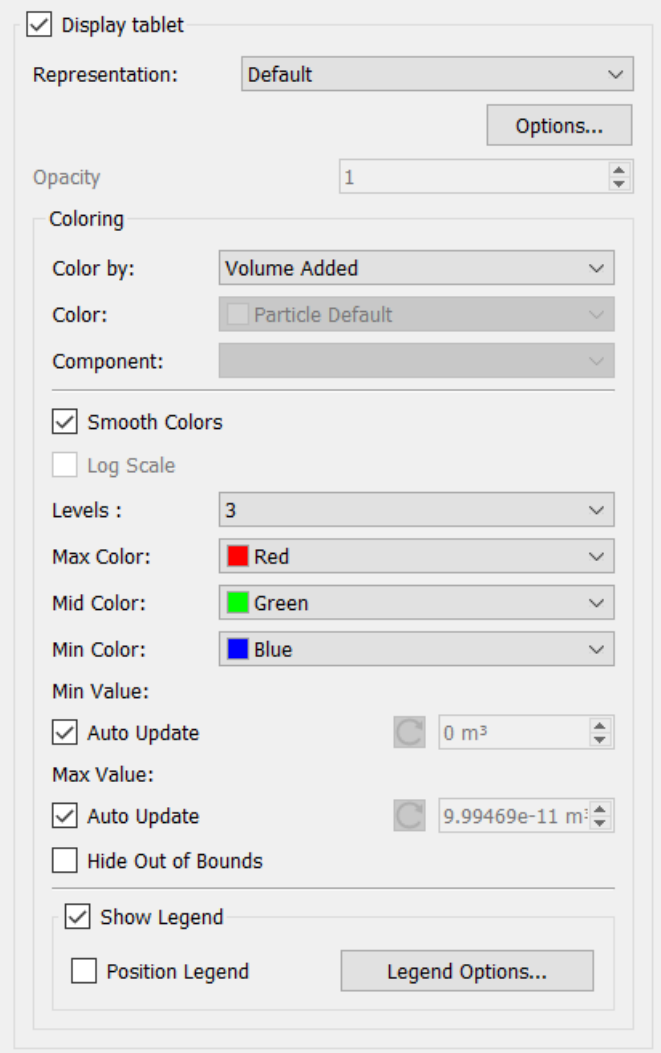


Figure 1665:

8. On the menu bar, set the time to `0` by clicking:

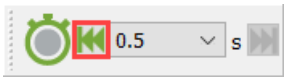


Figure 1666:

9. Set the View plane to **Default**.

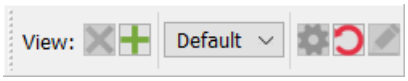


Figure 1667:

10. In the Viewer window, set the Playback Speed to `0.1x` and then click ▶ to play the particle flow animation.

Figure 1668:

Observe that the added volume increases in the beginning as the tablets receive the spray coating. Once the spray injection is stopped, the coating starts to dry up and the added volume decreases gradually.

Summary

In this tutorial, you have learned how to set up and run a basic AcuSolve-EDEM bidirectional (two-way) coupling problem with mass transfer. You learned how to create particles with specific position and internal spacing and also learned how to define a spray injection. Once the simulation was completed, you processed the results to view the coating variation over time.

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve.

This tutorial demonstrates the set up of a flow solution in SimLab. Before you begin, download the CAD geometry.

You will need an internet connection to watch the video tutorials and download the files.

Tutorial Video

https://fast.wistia.net/embed/iframe/2mjf69groa

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6107 AcuSolve - EDEM Bidirectional 3-Phase Coupling

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/pbx4v9otpb

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6108 AcuSolve - EDEM Bidirectional Coupling with User-Defined Drag Model

Perform this tutorial using the following user interface(s):

HyperMesh CFD

HyperMesh CFD

Tutorial Level: Advanced

Prerequisites

This tutorial covers the setup process for running a bidirectional AcuSolve EDEM coupled simulation using a custom user-defined drag model. An overview of the UDF code and the compilation process is provided in the beginning of the tutorial, followed by the simulation setup and post-processing. It is assumed that you have prior experience using HyperMesh CFD and EDEM, the basics of which are covered in the tutorials: ACU-T: 1000 UI Introduction, and ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM .

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6108_sphere.hm`

`sphere_EDEM.zip`

`usrStokesDrag.c.zip`

Problem Description

The problem to be solved is shown schematically in the figure below. In this tutorial, you will learn how to write a custom UDF for calculating the drag force on a solid particle in a creeping flow also known as Stokes flow. In Stokes flow, the fluid velocities are generally significantly low, or the fluid viscosities are significantly high and are characterized by very low particle Reynolds numbers (<<1) indicating that the inertial forces are negligible compared to the viscous forces. Stokes flow can be observed in microfluidic devices, blood flow in capillaries, solid particle sedimentation in quiescent fluid, and so on. According to the Stokes drag law, the total drag force (pressure drag plus shear stress drag) on a particle is given by:

$$F_D = 3\pi\varepsilon d_p \mu U \tag{6}$$

where, ε is the porosity; d_p is the particle diameter; μ is the fluid dynamic viscosity; U is the relative velocity.

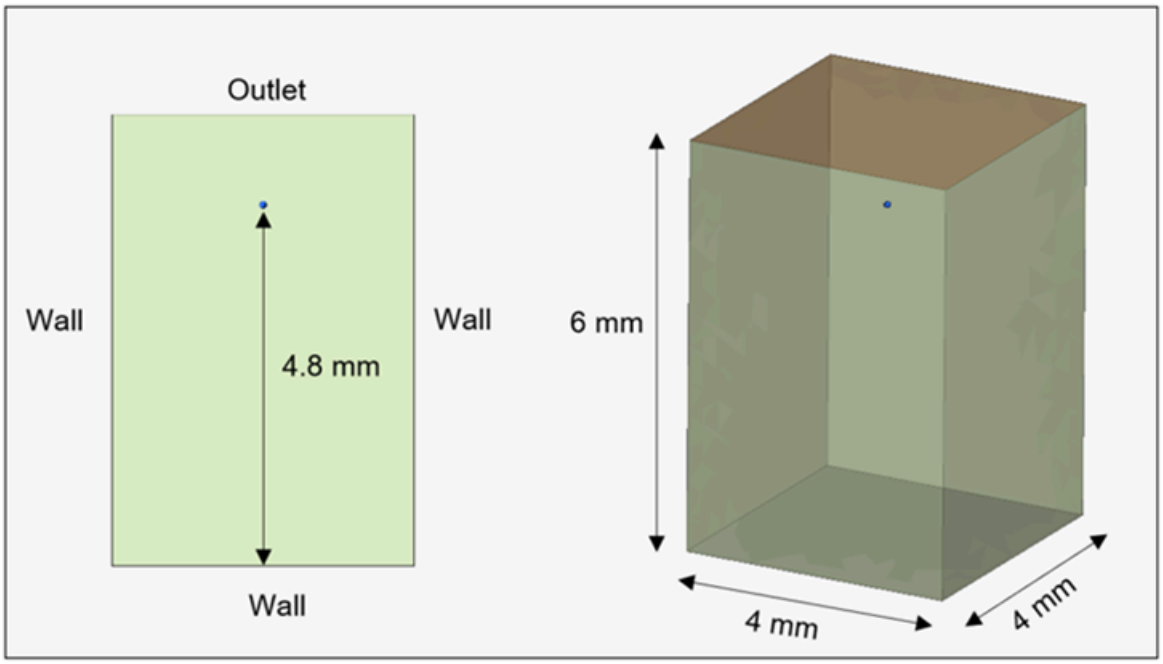


Figure 1669:

The objective of the simulation is to calculate the terminal velocity of the solid particle as it falls through the water column. The particle's initial location is 4.8 mm from the bottom of the water column. Under Stokes flow, the particle initially accelerates due to gravity and the fluid drag force is negligible. As the particle moves through the fluid, the frictional drag increases and the acceleration reduces to zero over time and thereafter the particle travels with a constant velocity, known as the terminal velocity.

Some important parameters used in the simulation are listed below:

Table 6:

Important Simulation Parameters	
Water density (kg/m^3)	1000
Water viscosity (Pa s)	0.001
Particle density (kg/m^3)	1500
Particle diameter (m)	0.0001

AcuSolve User-Defined Function (UDF) Overview

UDF Format

The general format of the UDF of a custom drag model for AcuSolve - EDEM bidirectional coupling is shown below.

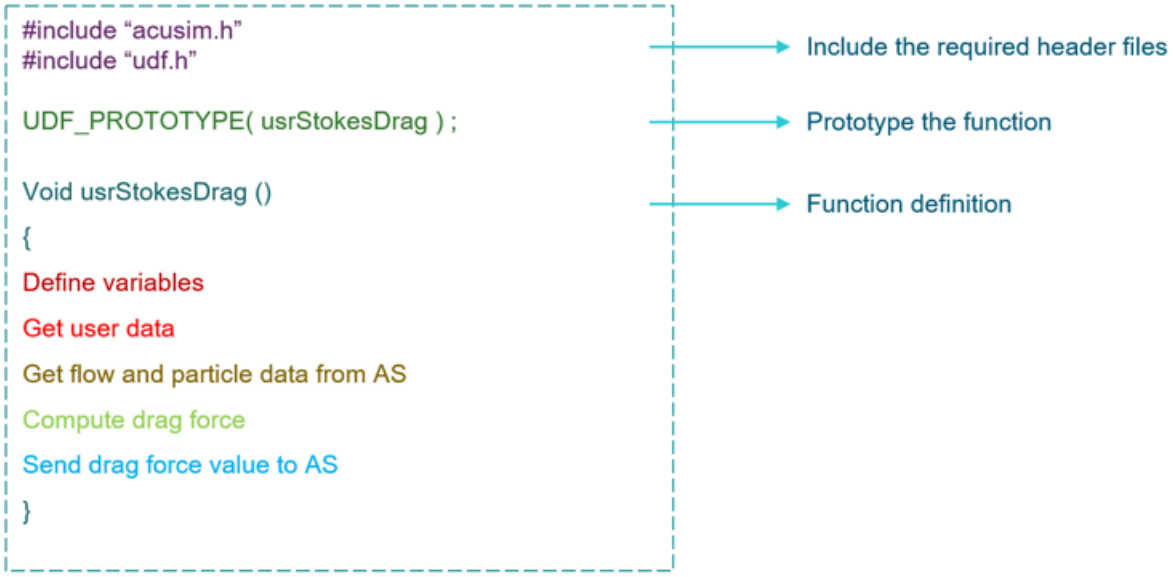


Figure 1670:

Header Files

The standard practice while writing C programs, or any other language for that matter, is to start with preprocessor directives such as including header files and defining macros. The following two header files should be included for all AcuSolve UDFs:

- `acusim.h`: This header file includes the main data types used by AcuSolve and the standard C header files such as `stdio.h`, `string.h` and `math.h`. Common data types used by AcuSolve:

a. Integer	Type integer
b. Real	Type floating point
c. String	Type string
d. Void	Type void

- `udf.h`: This header file contains all of the macros and declarations needed for accessing the data in the UDF.

Function Prototype

Once the header files are declared, you need to prototype the user function using the UDF_PROTOTYPE() macro and the name of the user function as the argument. For example, UDF_PROTOTYPE(usrStokesDrag).

This user function name should be the same as the one specified in the model setup. Otherwise, it can lead to run time errors. Although the user function names can be anything that is supported by C, it is good practice to use function names that start with usr to distinguish the user functions from other solver parameters.

Function Definition

A standard C function in AcuSolve UDF has the following syntax:

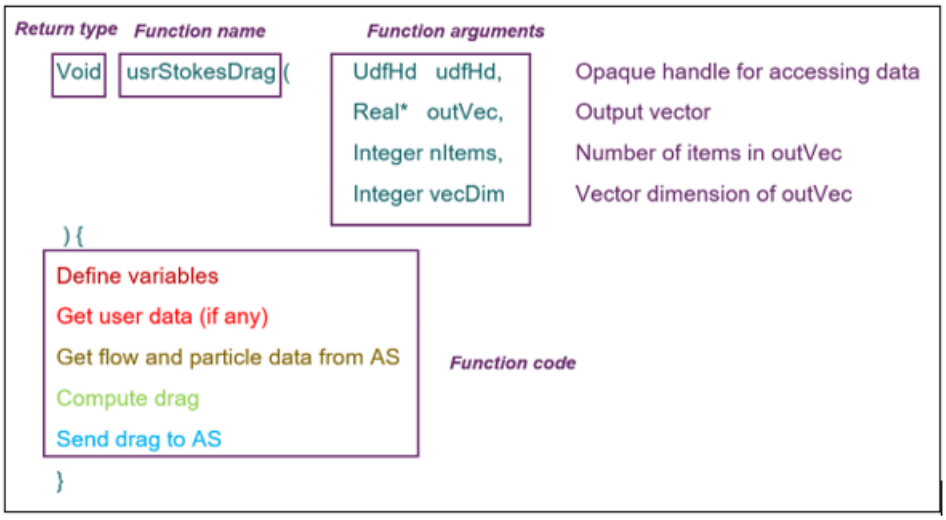


Figure 1671:

Return Type

All C functions should have a return type. Since the drag force values should be sent back to AcuSolve through outVec, the return value does not have any significance and the return type should be set to Void.

Function Arguments

The function arguments include:

udfHd	Pointer which contains the necessary information for accessing various data. All supporting routines require this argument.
outVec	Resulting vector of the user function. On input this array contains all zeros. This vector is used to send the force information back to AcuSolve.
nItems	This is the first dimension of outVec, which specifies the number of items that needs to be filled. For all drag model user functions,

	the value is 1 since you are calculating the drag force on 1 particle in each call.
vecDim	This is the second dimension of outVec which specifies the vector dimension of the particular data. This value depends on the type of data that is sent back.
Drag user function	Minimum three values (x, y ,z components) are required. Optionally a fourth value can also be sent back to AcuSolve (drag coefficient).
Lift user function	Three values (x, y, z components).
Torque user function	Three values (x, y, z components).

Function Code

Variables:

Variables are used to store certain types of data. This data can be a single value or an array of values. In C language, all variables need to be declared before they can be used in the program. Although the declaration can be done anywhere inside the program, declaring all the required variables upfront makes the code more organized and easily understandable. The syntax for declaring a variable in C is Datatype variable_name ;

Example

Real volume ;

where Real is the data type and volume is the name of the variable.

In many UDFs, including the current example, you may need to access some data from AcuSolve which has already been calculated and stored by the solver at a particular memory address. For this, you need to declare pointer variables which point to the memory address of the requested data. For instance, if you need to access the local fluid velocity, you need to declare a pointer variable that stores the memory address of the location where AcuSolve has already stored the value.

Example

Real* xVel ;

Accessing data from AcuSolve:

The routine to be used for accessing particle and fluid data from AcuSolve is udfGetEDEMData(udfHd, dataName). This returns an array of data and length of the array depends on the type of data that is requested.

For example,

To access the particle’s volume, first you need to declare a pointer variable and then use the following data routine.

Real* vol ;

vol = udfGetEDEMData(udfHd, UDF_EDEM_PARTICLE_VOLUME) ;

Note: vol is a pointer variable and the actual value of the particle volume is stored in vol[0].

The list of all of the AS-EDEM data names is shown below.

```
/*============================================================================
 *
 * udfGetEDEMData dataName
 *
 *============================================================================
 */

#define UDF_EDEM_FLOW_DENSITY            1      /* flow density          */
#define UDF_EDEM_FLOW_VISCOSITY          2      /* flow viscosity        */
#define UDF_EDEM_FLOW_POROSITY           3      /* flow porosity         */
#define UDF_EDEM_FLOW_VELOCITY           4      /* flow velocity         */
#define UDF_EDEM_FLOW_VELOCITY_CURL      5      /* flow velocity curl    */
#define UDF_EDEM_PARTICLE_VOLUME         6      /* particle volume       */
#define UDF_EDEM_PARTICLE_ANGLE          7      /* particle angle        */
#define UDF_EDEM_PARTICLE_POSITION       8      /* particle position     */
#define UDF_EDEM_PARTICLE_VELOCITY       9      /* particle velocity     */
#define UDF_EDEM_PARTICLE_VELOCITY_CURL  10     /* particle velocity curl*/
#define UDF_EDEM_DRAG_COEF               11     /* drag coefficient       */
#define UDF_EDEM_PAXIS_VELOCITY          12     /* pAxis X slipVel X slipVel */
#define UDF_EDEM_DRAG                    13     /* drag                   */
#define UDF_EDEM_LIFT                    14     /* lift                   */
#define UDF_EDEM_PARTICLE_QUATERNION     15     /* quaternion             */
```

Since using a lot of pointer variables can be confusing and difficult to track for longer UDFs, it is better to limit them to as few as possible and assign the values to other variables. For example, for the stokes drag law $F_D = 3\pi\epsilon d_p \mu U$, you need to access the particle's volume, and fluid's porosity, viscosity and velocity. You can define a pointer variable named scalar and point that to the required scalar quantities such as volume, density, porosity and viscosity recursively while assigning the value to other variables in between.

```
Void usrStokesDrag( UdfHd       udfHd,
                    Real*       outVec,
                    Integer     nItems,
                    Integer     vecDim )
{

    Real    beta ;
    Real    vol ;
    Real    visc ;
    Real    diam ;
    Real*   scalar ;
    Real*   flowVel ;
    Real*   particleVel ;
    Real    slipVel[3] ;
    Real    poros ;

/*---------------------------------------------------------------------------
 * Get particle and flow data at current particle location
 *---------------------------------------------------------------------------
 */
    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_PARTICLE_VOLUME) ;
    vol     = scalar[0] ;

    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_VISCOSITY ) ;
    visc    = scalar[0] ;

    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_POROSITY  ) ;
    poros   = scalar[0] ;

    flowVel = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_VELOCITY  ) ;

    particleVel = udfGetEDEMData(   udfHd,  UDF_EDEM_PARTICLE_VELOCITY) ;
```

Figure 1672:

Once a pointer named scalar is declared, you first pointed it to the particle volume dataname. In the next line you assigned the value scalar[0] to a variable named vol. Next, you modified the pointer scalar to flow viscosity and assigned scalar[0] to visc.

> **Note:** This is only one out of many correct ways of writing a program.

- Calculate drag force values:

Once all the variables are declared and required data is accessed, you need to calculate the force values based on the drag law. Here, the 0,1,2 indices refer to the x, y and z components, respectively.

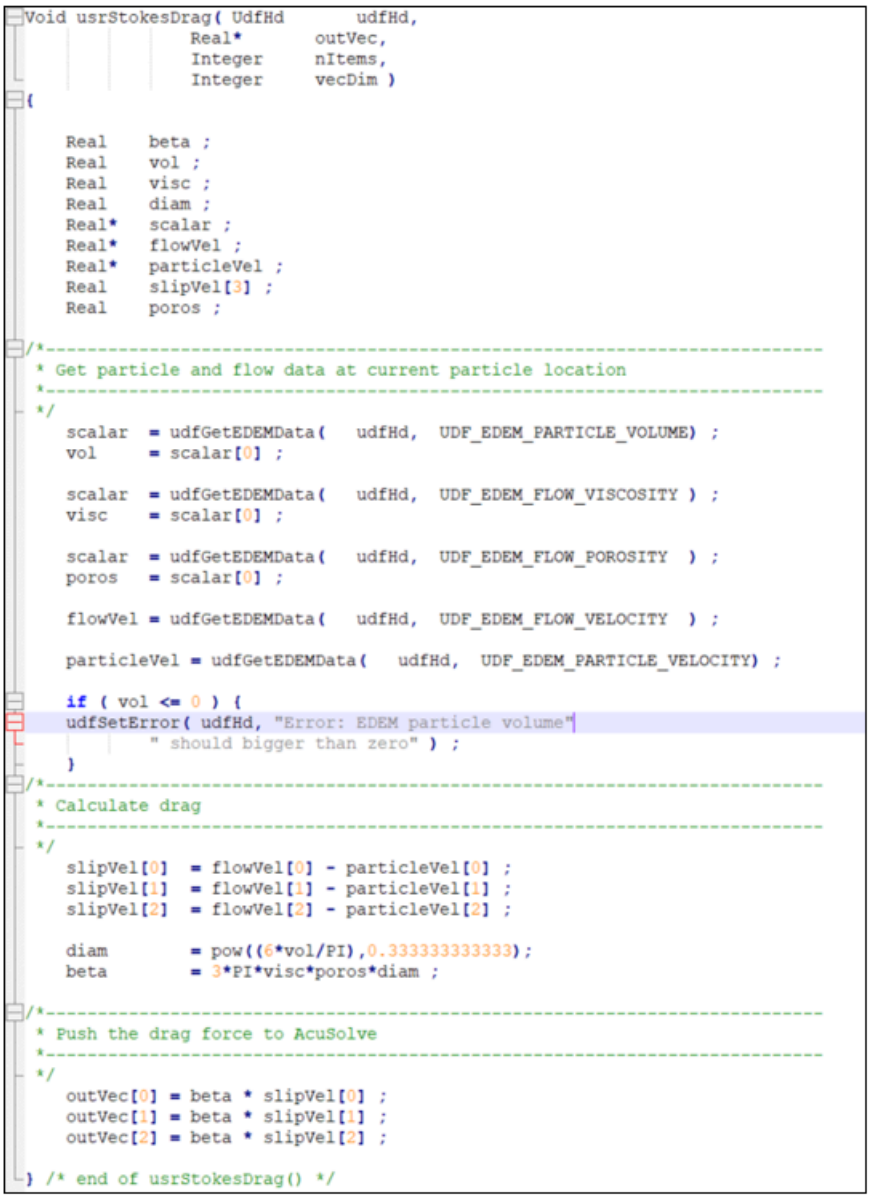
- Send force data to AcuSolve:

After the force has been calculated, the outVec array should be filled. Since this is a drag force user function, at least 3 values should be sent back. Optionally, the fourth value can also be set which refers to the drag coefficient. (Not needed in this example.)

```
/*-----------------------------------------------------------------------------
 * Calculate drag
 *-----------------------------------------------------------------------------
 */
    slipVel[0]  = flowVel[0] - particleVel[0] ;
    slipVel[1]  = flowVel[1] - particleVel[1] ;
    slipVel[2]  = flowVel[2] - particleVel[2] ;

    diam        = pow((6*vol/PI),0.333333333333);
    beta        = 3*PI*visc*poros*diam ;

/*-----------------------------------------------------------------------------
 * Push the drag force to AcuSolve
 *-----------------------------------------------------------------------------
 */
    outVec[0] = beta * slipVel[0] ;
    outVec[1] = beta * slipVel[1] ;
    outVec[2] = beta * slipVel[2] ;

} /* end of usrStokesDrag() */
```

Figure 1673:

The full Stokes drag function is shown below.



```
Void usrStokesDrag( UdfHd       udfHd,
                    Real*       outVec,
                    Integer     nItems,
                    Integer     vecDim )
{

    Real    beta ;
    Real    vol ;
    Real    visc ;
    Real    diam ;
    Real*   scalar ;
    Real*   flowVel ;
    Real*   particleVel ;
    Real    slipVel[3] ;
    Real    poros ;

/*---------------------------------------------------------------------------
 * Get particle and flow data at current particle location
 *---------------------------------------------------------------------------
 */
    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_PARTICLE_VOLUME) ;
    vol     = scalar[0] ;

    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_VISCOSITY ) ;
    visc    = scalar[0] ;

    scalar  = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_POROSITY  ) ;
    poros   = scalar[0] ;

    flowVel = udfGetEDEMData(   udfHd,  UDF_EDEM_FLOW_VELOCITY  ) ;

    particleVel = udfGetEDEMData(   udfHd,  UDF_EDEM_PARTICLE_VELOCITY) ;

    if ( vol <= 0 ) {
    udfSetError( udfHd, "Error: EDEM particle volume"
            " should bigger than zero" ) ;
    }
/*---------------------------------------------------------------------------
 * Calculate drag
 *---------------------------------------------------------------------------
 */
    slipVel[0]  = flowVel[0] - particleVel[0] ;
    slipVel[1]  = flowVel[1] - particleVel[1] ;
    slipVel[2]  = flowVel[2] - particleVel[2] ;

    diam        = pow((6*vol/PI),0.333333333333);
    beta        = 3*PI*visc*poros*diam ;

/*---------------------------------------------------------------------------
 * Push the drag force to AcuSolve
 *---------------------------------------------------------------------------
 */
    outVec[0] = beta * slipVel[0] ;
    outVec[1] = beta * slipVel[1] ;
    outVec[2] = beta * slipVel[2] ;

} /* end of usrStokesDrag() */
```

Figure 1674:

Full Code Snippet

```
#include "acusim.h"
#include "udf.h"
/*===========================================================================
 *
 * Prototype the function
 *
 *===========================================================================
 */
UDF_PROTOTYPE( usrStokesDrag ) ;
/*===========================================================================
 *
 * "stokesDrag":  calculate EDEM particle drag force based on the Stokes law
 *
```

```
 *  Arguments:
 *    udfHd     - opaque handle for accessing information
 *    outVec    - output vector
 *    nItems    - number of items in outVec (=1 in this case)
 *    vecDim    - vector dimension of outVec (=3 in this case)
 *
 *  Input file parameters:
 *    user_values = { }
 *    user_strings = { "" }
 *
 *=============================================================================
 */
Void usrStokesDrag( UdfHd       udfHd,
   Real*      outVec,
   Integer      nItems,
   Integer      vecDim )
{
    Real    beta ;
    Real    vol ;
    Real    visc ;
    Real    diam ;
    Real*   scalar ;
    Real*   flowVel ;
    Real*   particleVel ;
    Real    slipVel[3] ;
    Real    poros ;
/*-----------------------------------------------------------------------------
 * Get particle and flow data at current particle location
 *-----------------------------------------------------------------------------
 */
    scalar = udfGetEDEMData(  udfHd, UDF_EDEM_PARTICLE_VOLUME) ;
    vol = scalar[0] ;
    scalar = udfGetEDEMData(  udfHd, UDF_EDEM_FLOW_VISCOSITY ) ;
    visc = scalar[0] ;
    scalar = udfGetEDEMData(  udfHd, UDF_EDEM_FLOW_POROSITY ) ;
    poros = scalar[0] ;
    flowVel = udfGetEDEMData( udfHd, UDF_EDEM_FLOW_VELOCITY ) ;
    particleVel = udfGetEDEMData(   udfHd, UDF_EDEM_PARTICLE_VELOCITY) ;
    if ( vol <= 0 ) {
 udfSetError( udfHd, "Error: EDEM particle volume"
   " should bigger than zero" ) ;
    }
/*-----------------------------------------------------------------------------
 * Calculate drag
 *-----------------------------------------------------------------------------
 */
    slipVel[0] = flowVel[0] - particleVel[0] ;
    slipVel[1] = flowVel[1] - particleVel[1] ;
    slipVel[2] = flowVel[2] - particleVel[2] ;
    diam = pow((6*vol/PI),0.333333333333);
    beta = 3*PI*visc*poros*diam ;
/*-----------------------------------------------------------------------------
 * Push the drag force to AcuSolve
 *-----------------------------------------------------------------------------
 */
    outVec[0] = beta * slipVel[0] ;
    outVec[1] = beta * slipVel[1] ;
    outVec[2] = beta * slipVel[2] ;
} /* end of usrStokesDrag() */
```

For more information about the UDF routines available refer to the *AcuSolve User-Defined Functions Manual*.

Compiling the UDF

For the solver to access the user-defined functions, the functions must be compiled and linked into a shared library. These libraries should be placed in the problem directory along with the solver input files.

Windows

From the Start menu, expand Altair <solver version> and launch the AcuSolve Cmd Prompt.

In the terminal, cd to the directory where the model files and the UDF script file are located. Then execute the following command to compile the UDF:

acuMakeDll -src usrStokesDrag.c

Linux

Open a command line prompt and cd to the directory where the model files and the UDF script are located. Then execute the following command:

acuMakeLib -src usrStokesDrag.c

Part 1 – AcuSolve Simulation

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1675:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6108_sphere.hm` and click **Open**.

   This will be the working directory and all of the files related to the simulation will be stored in this location.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate

icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1676:

Set Up Flow

Set the General Simulation Parameters

1. From the Flow ribbon, click the **Physics** tool.

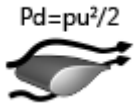
   

   Figure 1677:

   The **Setup** dialog opens.
2. Under the Physics models setting, select the **Multiphase flow** radio button.
3. Change the Multifluid type to **Bidirectional EDEM Coupling**.
4. Click the Eulerian material drop-down menu and select **Material Library** from the list.
   You can create new material models in the Material Library.
5. In the Material Library dialog, select **EDEM 2 Way Multiphase**, switch to the **My Material** tab, and then click + to add a new material model.
6. In the microdialog, click **EDEM Bidirectional Material** in the top-left corner and change the name to `Water-Particle`.
7. Set the Carrier field to **Water**.
8. Verify that the Particle Shape is set to `Spherical` and set the Drag model to `User Function`.
9. Click the table icon beside the drag model drop-down.

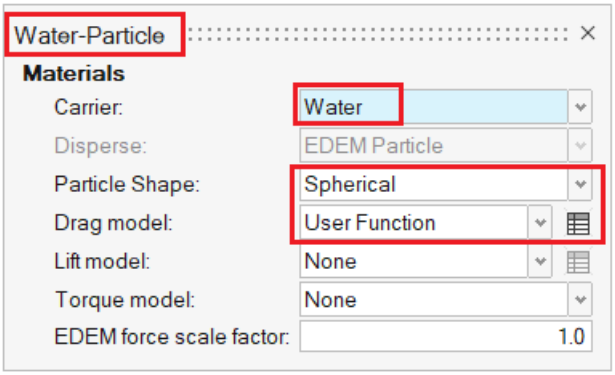


Figure 1678:

10. In the new dialog, set the Function name to `usrStokesDrag`.

> **Note:** The function name entered here should be the same as the name of the UDF_PROTOTPYE in the UDF. The user-function name can be any name supported in C that starts with the letters usr.

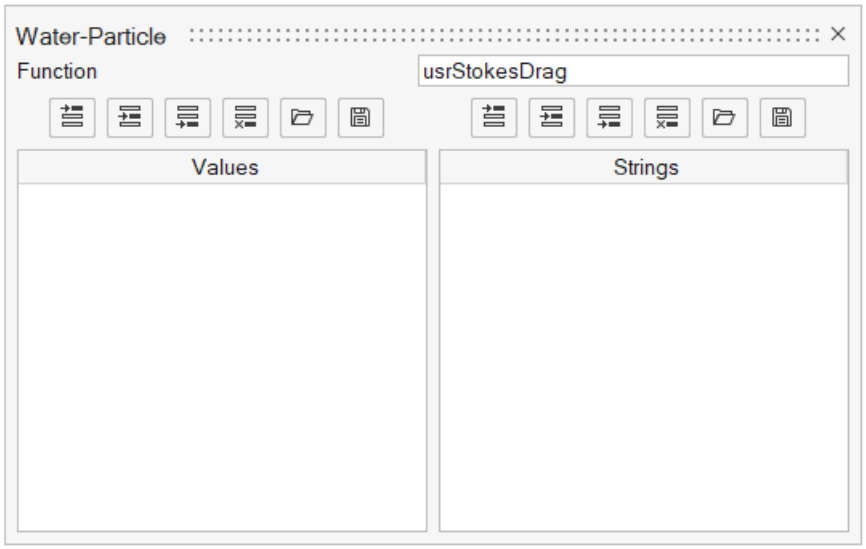


Figure 1679:

11. Press **Esc** to close the dialog.

12. Close the **Material Model** and **Material Library** dialogs.

13. In the **Setup** dialog, set the Eulerian material to `Water-Particle`.

14. Set Time step size and Final time to `5e-05` and `0.015`, respectively.

15. Set the Turbulence model to `Laminar`.

The calculated particle Reynolds number is less than 0.3 which is in the Stokes regime where the viscous forces dominate the inertial forces and the turbulence effects can be neglected.

16. Verify that Gravity is set to `0`, `0`, `-9.81`.

17. Set the Pressure scale to `Absolute`.

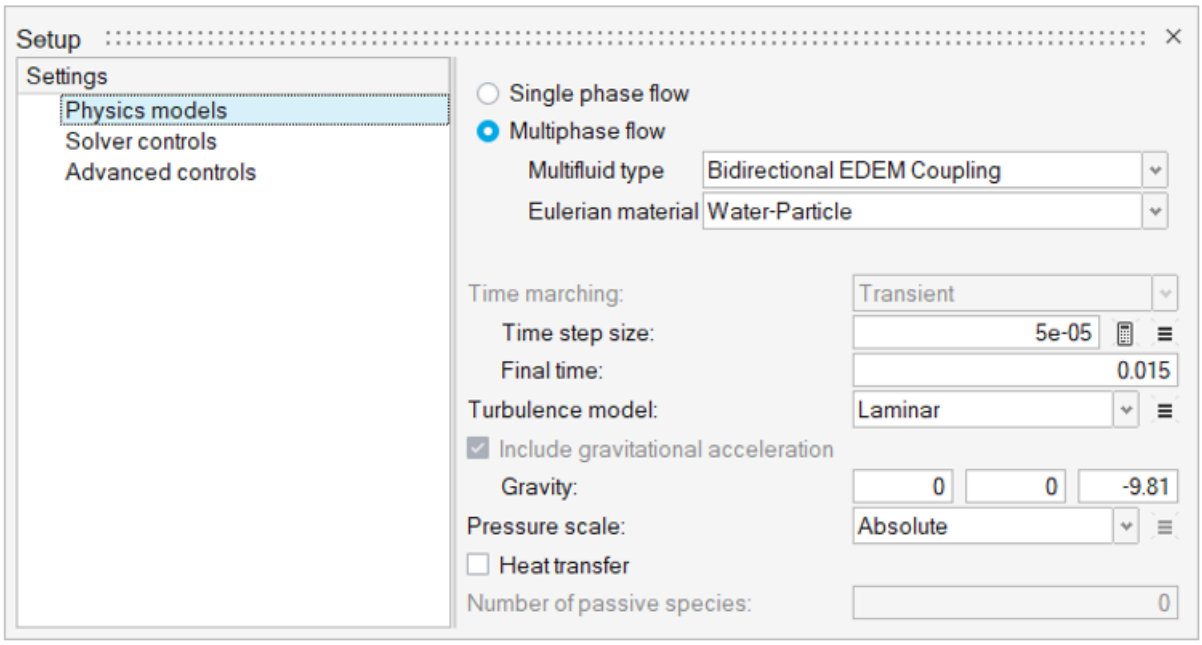


Figure 1680:

18. Click the **Solver controls** setting and set the Minimum/Maximum stagger iterations to **2**, respectively.

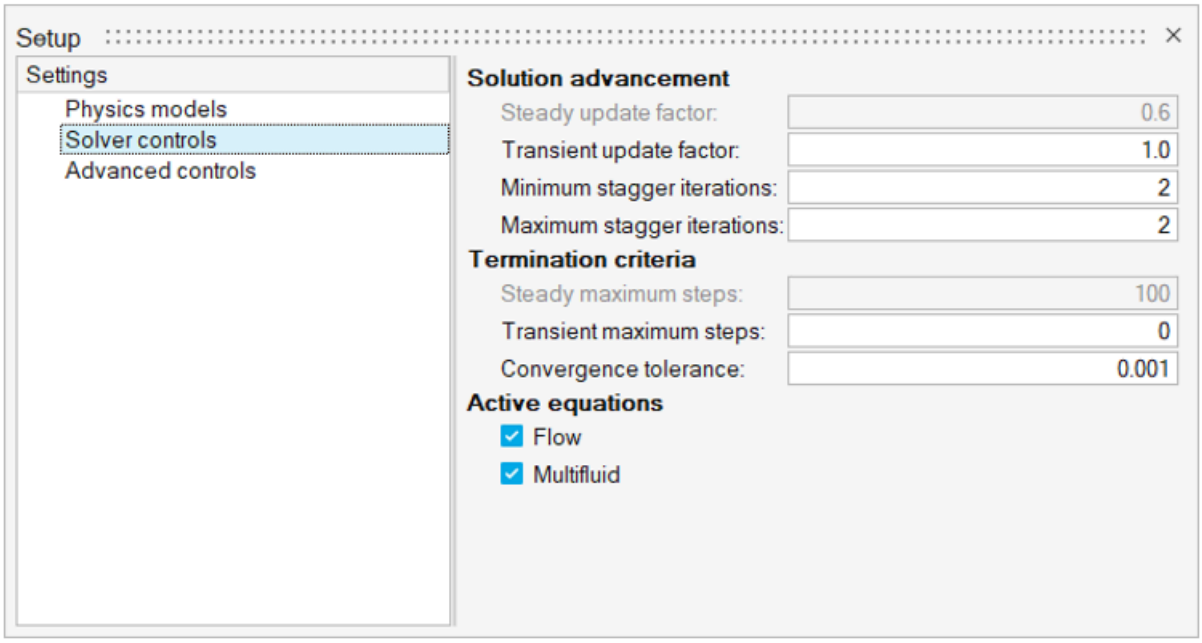


Figure 1681:

19. Close the dialog and save the model.

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1682:

2. Verify that Water-Particle has been assigned as the material.
   If not assigned, click the geometry and select **Water-Particle** from the microdialog.
3. On the guide bar, click ✓ to exit the tool.

Define Flow Boundary Conditions

1. Click the **Outlet** tool in the Boundaries group.

Figure 1683:

2. In the modeling window, click the top surface of the model, which is highlighted in the figure below.

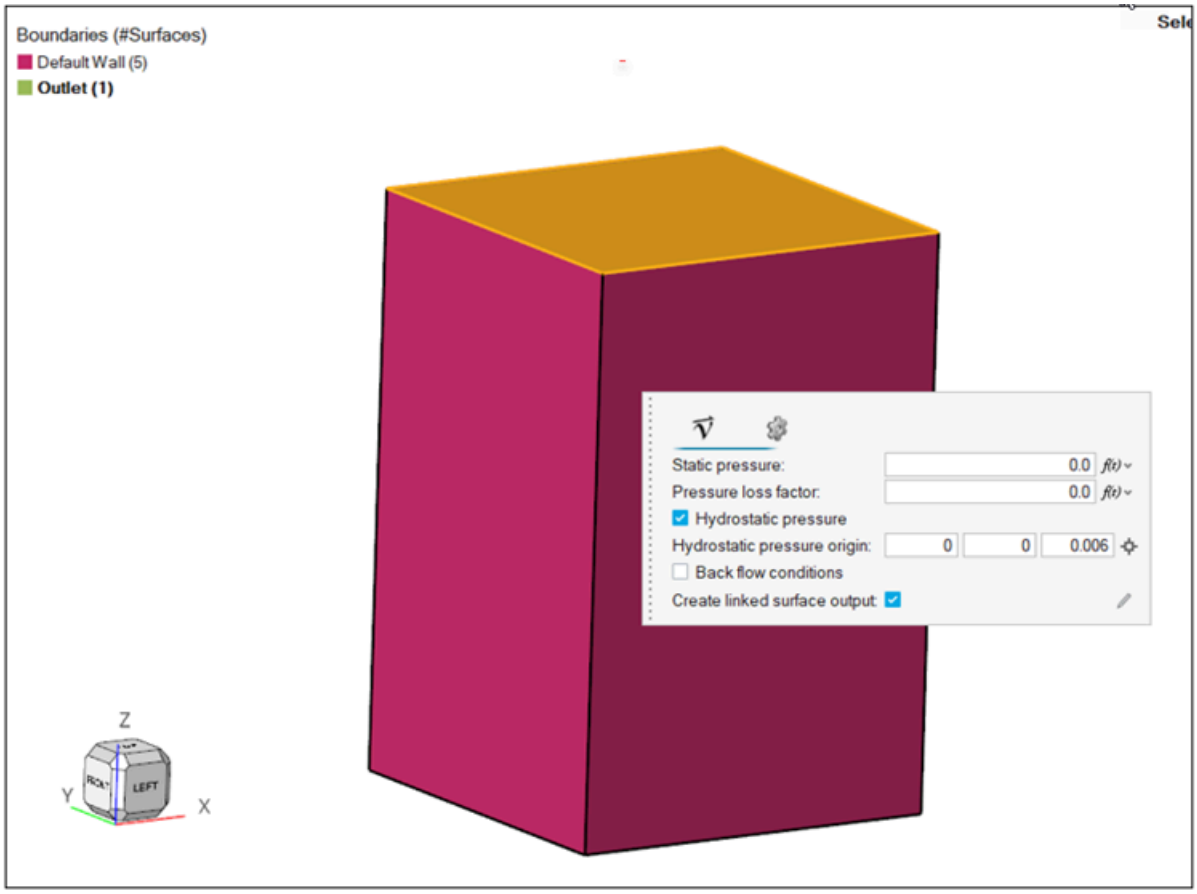


Figure 1684:

3. In the microdialog, activate the **Hydrostatic pressure** option and set the Hydrostatic pressure origin to (`0`, `0`, `0.006`.)
4. Click ✓ on the guide bar.

> **Note:** A no-slip wall condition is assigned to all surfaces by default.

5. Save the model.

Generate the Mesh

1. From the Mesh ribbon, click the **Volume** tool.

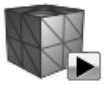


Figure 1685:

The **Meshing Operations** dialog opens.

2. In the **Meshing Operations** dialog, set the Mesh size to **Average Size** and set the Average element size to `0.0005`.
3. Deactivate **Curvature-based surface refinement** and then click **Mesh**.

Define Nodal Outputs

Once the meshing is complete, you are automatically taken to the Solution ribbon.

1. From the Solution ribbon, click the **Field** tool.

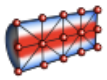


Figure 1686:

The **Field Output** dialog opens.

2. Check the box for **Write initial conditions**.
3. Check the box for **Write results at time step interval**.
4. Set the Time step interval to `5`.

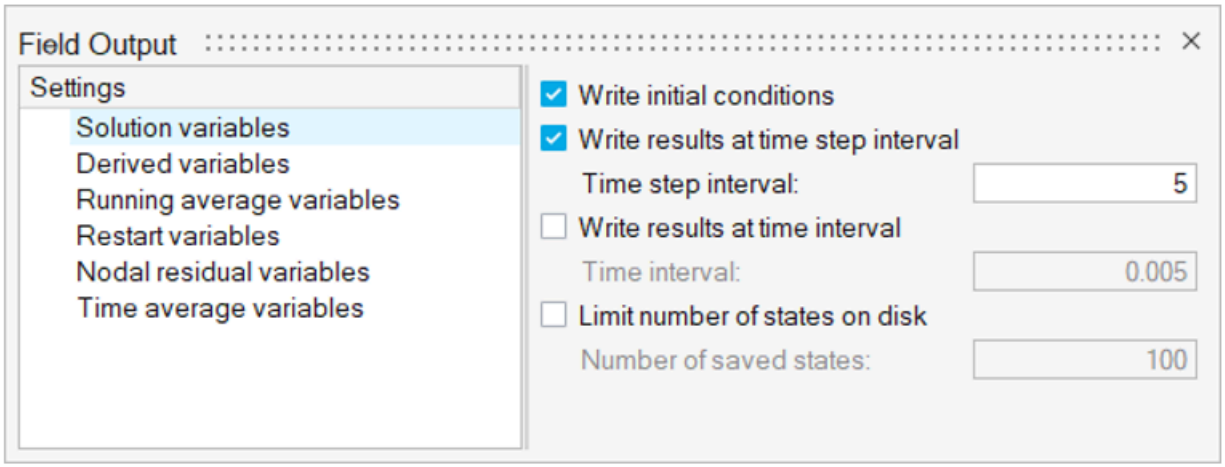


Figure 1687:

5. Close the dialog and save the model.

Part 2 - EDEM Simulation

Start Altair EDEM from the Windows start menu by clicking **Start** > **Altair 2025** > **EDEM 2025** .

Open the EDEM Input Deck

1. In the Creator tab in EDEM, go to **File** > **Open**.
2. Browse to the directory where you saved the `sphere.dem` file and open it.
   This loads the geometry and the materials.

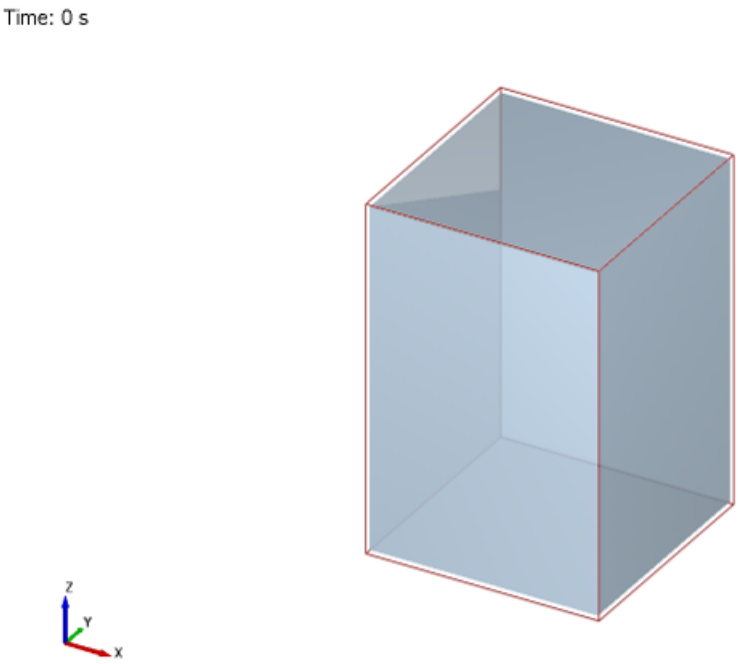


Figure 1688:

Review the EDEM Model Setup

The main area of focus for this tutorial is the user-defined drag model setup and compilation process for a bidirectional AcuSolve - EDEM coupling simulation. Therefore, for the sake of brevity, the EDEM database provided with the input files contains all of the required setup.

Summary of the EDEM Model

A spherical particle of diameter 0.0001 m is introduced in a water column at a point which is 0.0048 m from the bottom. Since the density of the particle is 1500 kg/m^3, that is, heavier than water, the particle falls freely through the water column.

Define the Simulation Settings

1. Click in the top-left corner to go to the EDEM Simulator tab.
2. In the Simulator Settings tab, set the Time Integration scheme to **Euler** and deactivate the **Auto Time Step** checkbox.

3. Enter a value of `1e-6`s for Fixed Time Step.

> **Note:** Generally, a value of 20-40% of the Rayleigh Time Step is recommended as the time step size to ensure stability of the DEM simulation. Since the DEM time steps are usually a few orders lower than the CFD time step, for every CFD time step the DEM solver performs multiple iterations to match with the CFD time. Therefore, the CFD time step should always be an integer multiple of DEM time step to ensure that the physical times in both AcuSolve and EDEM match.

4. Set the Total Time to `0.015` s and the Target Save Interval to `0.00015` s.
5. Set the Cell Size to `2.5` R min.
6. Set the Selected Engine to **CPU Solver** and set the Number of CPU Cores based on availability.

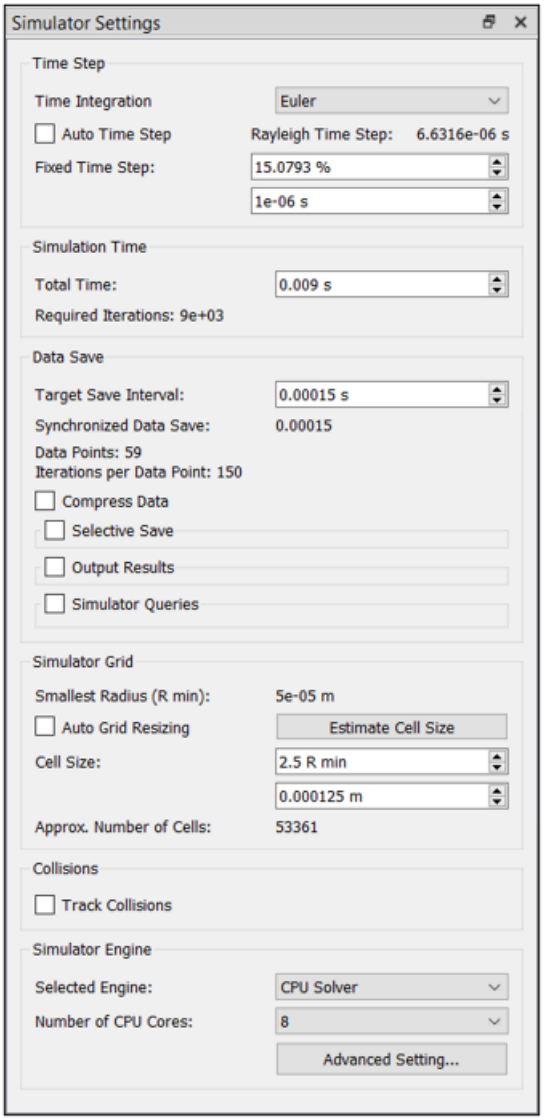


Figure 1689:

7. Once the simulation settings have been defined, save the model.

Submit the Coupled Simulation

1. Start the coupling server by clicking **Coupling Server** in EDEM.

Figure 1690:

Once the Coupling server is activated, the icon changes.

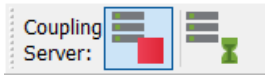


Figure 1691:

2. Return to HyperMesh CFD.
3. From the Solution ribbon, click the **Run** tool.

Figure 1692:

The **Launch AcuSolve** dialog opens.

4. Set the Parallel processing option to **Intel MPI**.
5. Set the Number of processors to `6`.
6. Uncheck **Automatically define pressure reference**.
7. Expand **Default initial conditions**, uncheck **Pre-compute flow** and set the velocity values to `0`. Uncheck **Pre-compute Turbulence**.
8. Click **Run** to launch AcuSolve.

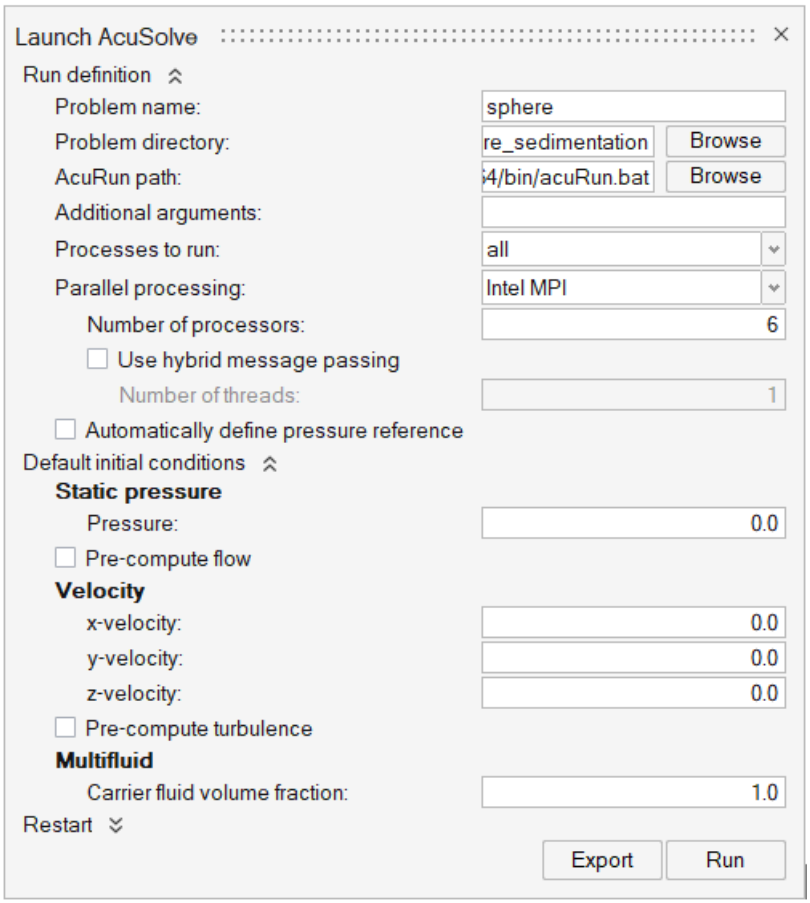


Figure 1693:

Once the AcuSolve run is launched, the **Run Status** dialog opens.

9. In the dialog, right-click the AcuSolve run and select **View log file**.

If the coupling with EDEM is successful, that information is printed in the `.log` file.

```
acuSolve:       Working directory = ACUSIM.DIR
acuSolve: ---------- Input Phase ----------
acuSolve: Input CPU/Elapse time     = 2.016000e+00 1.639000e+00 Sec
acuSolve: Input Memory Usage        = 7.400000e+01 Mbytes
acuSolve: EDEM coupling client initialized
acuSolve: Connecting to EDEM... successful
acuSolve: Interface thickness is 6.591507e-04
acuSolve: Init. CPU/Elapse time     = 2.340000e-01 2.800000e-02 Sec
acuSolve: ---------- Begin Time Step ----------
```

Figure 1694:

Once the simulation is complete, the summary of the run time is printed at the end of the `.log` file.

Analyze the Results

1. Once the EDEM simulation is complete, click in the top-left corner to go to the EDEM Analyst tab.
2. Click the **Create Graph** icon.

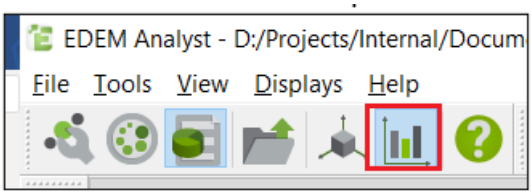


Figure 1695:

3. In the Analyst tree, select the **Line** plot and make sure that the parameters for X-axis are set as shown in the figure below.

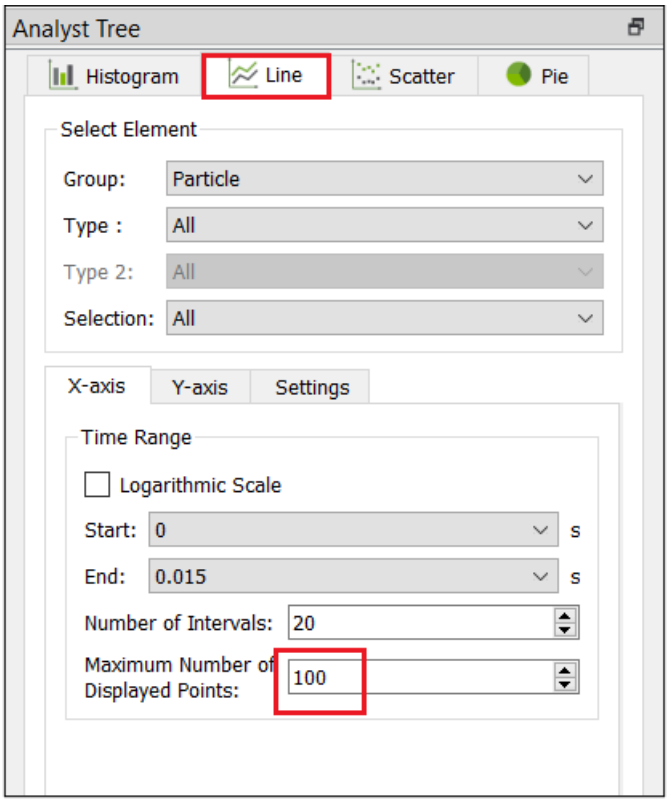


Figure 1696:

4. Click the **Y-axis** tab and set the Primary Attribute to **Velocity** and the Component to **Magnitude** (**Average**).
5. Click **Create Graph**.

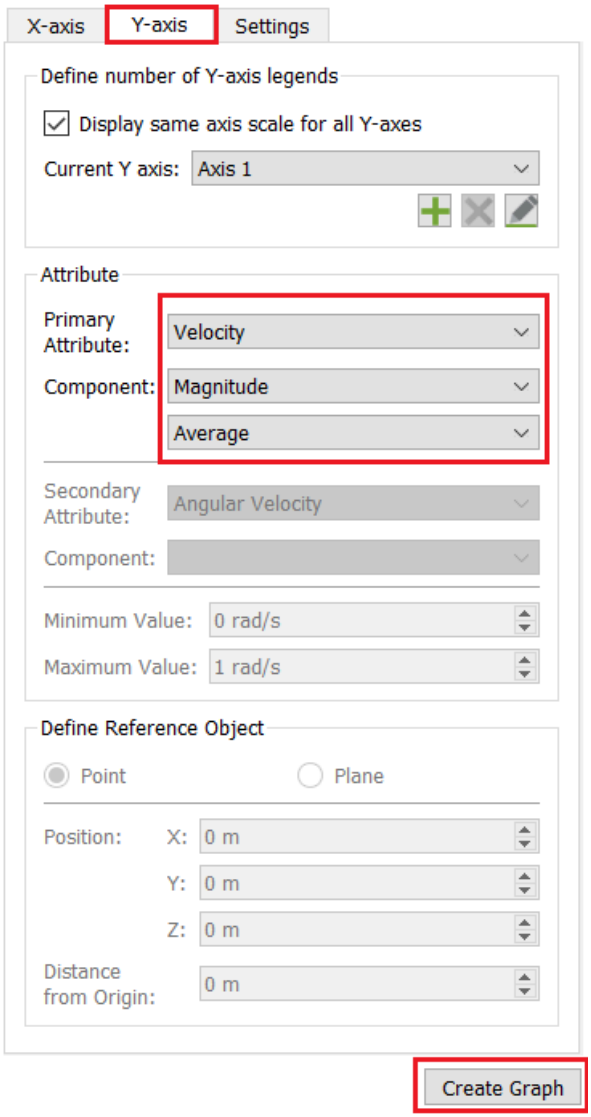


Figure 1697:

The particle velocity vs time plot displays in the plot window.

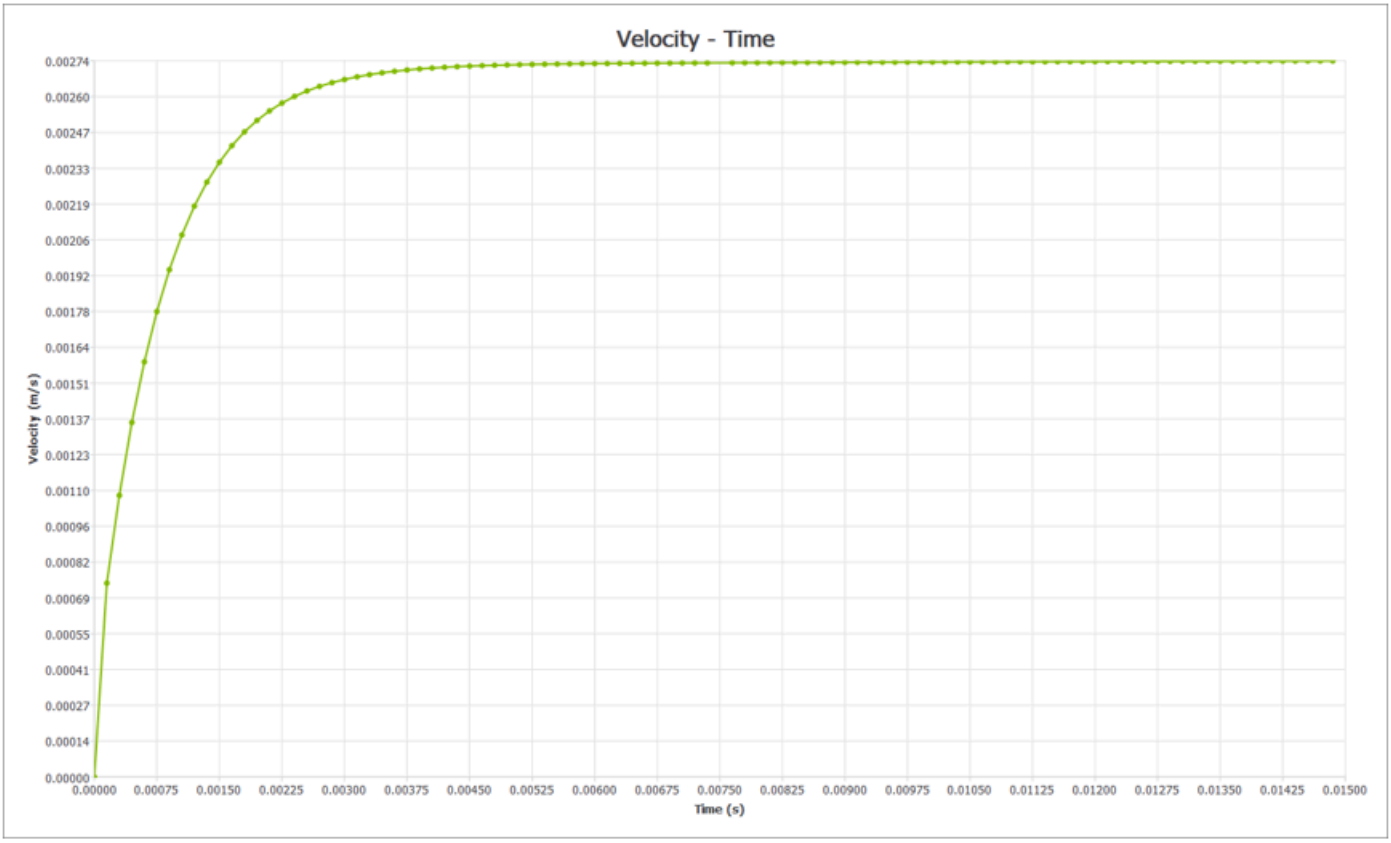


Figure 1698:

As expected, at the beginning of the simulation, the particle accelerates from zero-velocity because of gravity. As the particle moves and displaces the surrounding fluid, the frictional drag on the particle increases and due to the combined effect of buoyancy and drag force, the acceleration reduces to 0 and the particle moves with a constant velocity. This is known as the particle terminal velocity and in this case the predicted value is 0.00274 m/s which is very close (0.5% error) to the analytical solution (0.002725 m/s). This shows that the Stokes drag law is very accurate in predicting the fluid-induced drag forces for creeping flows.

Summary

In this tutorial, you learned how to write a user-defined function in C programming language for implementing a custom drag model for bidirectional AcuSolve-EDEM coupling. The Stokes drag law was used to calculate the terminal velocity of a spherical particle falling through a water column. The simulation results showed that the Stokes drag law is quite accurate in predicting the drag force on a particle in low Reynolds number flows.

Porous Medium

7

This chapter covers the following:

- ACU-T: 6500 Flow Through Porous Medium (p. 1016)
- ACU-T: 6501 Flow Through Porous Medium with Physical Velocity (p. 1032)

ACU-T: 6500 Flow Through Porous Medium

Perform this tutorial using the following user interface(s):

HyperMesh CFD

SimLab

HyperMesh CFD

Tutorial Level: Beginner

Prerequisites

This tutorial provides the instructions for setting up, solving and viewing results for a simulation of a flow through porous medium. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6500_PorousMedia.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a cylindrical channel with a porous medium in the flow section. As the flow passes through this section, a pressure drop is observed. In this simulation, an inlet velocity will be assigned to the flow and pressure drop across the porous medium will be calculated. The length of the porous section is 0.06 m and the fluid is air with a density of 1.225 kg/m^3 and a molecular viscosity of 1.781e-5 kg/m-s. The inlet velocity of the flow is 0.2 m/s.

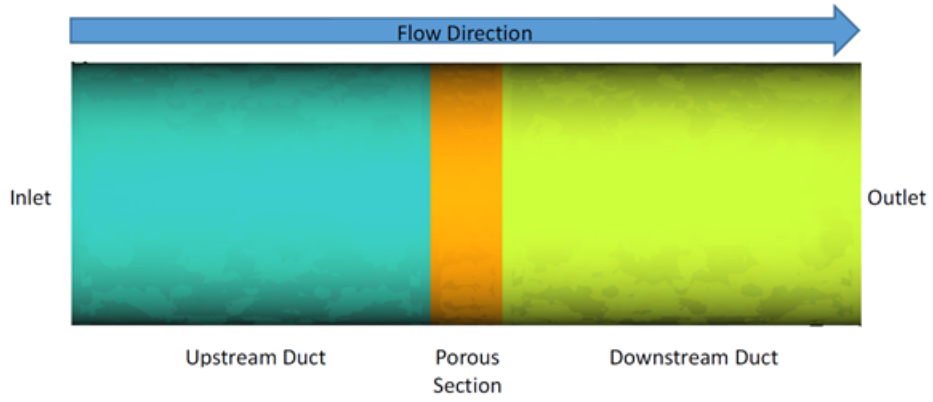


Figure 1699:

In AcuSolve, a porous media problem can be solved using one of two methods: the superficial velocity method and physical velocity method. In this tutorial, we adopt the superficial velocity method, which

provides good representation of the pressure drop through a porous media. However, it cannot predict the velocity increase in the porous region since the velocity in the porous region remains the same as those outside of the porous zones. The more accurate representation of velocity inside the porous region can be obtained using the physical velocity method, which solves the continuity and momentum equation inside the porous media using the intrinsic averaging method.

The pressure loss across the porous region using the superficial velocity method is modelled as:

$$\frac{\Delta P}{\Delta x_i} = C_D \frac{\mu}{k} V_i + C_p \frac{\rho}{\sqrt{k}} V_m V_i$$

where ΔP is the pressure drop (Pa) across the porous region, Δx_i is the porous region length (m), k is the permeability, C_D is the Darcy coefficient, C_p is the Forchheimer coefficient, μ is the viscosity (kg/m-sec), ρ is density (kg/m^3), V_m is velocity magnitude (m/sec), and V_i is velocity (m/sec).

In this tutorial, you will utilize the porous coefficient calculator to calculate the Darcy coefficient and the Forchheimer coefficient after curve fitting of the array data of pressure drop vs velocity. The pressure drop across the porous region can be simplified as the permeability is not included in the curve fitting.

$$\frac{\Delta P}{\Delta x} = C_D \mu V_x + C_p \rho V_x^2 \sim B V_x + A V_x^2$$

The figure below shows the Darcy and Forchheimer coefficient calculator after providing the velocity (m/s) and pressure drop (Pa) data in addition to the thickness (m) of the porous medium zone, density (kg/m^3) and viscosity (kg/m-s) of air.

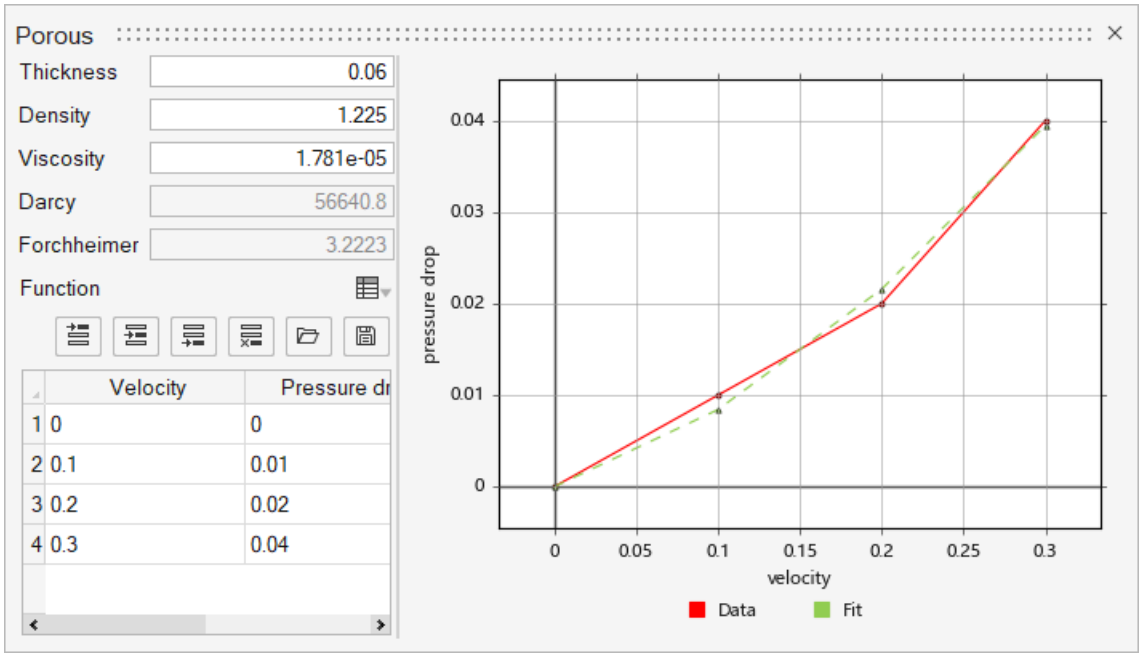


Figure 1700:

The Darcy coefficient C_D and Forchheimer coefficient C_p are then calculated based on the curve fit coefficients

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1701:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6500_PorousMedia.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `PorousMedia` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `PorousMedia` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1702:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

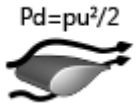


Figure 1703:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Steady**.
   b) Select **Laminar** as the Turbulence model.

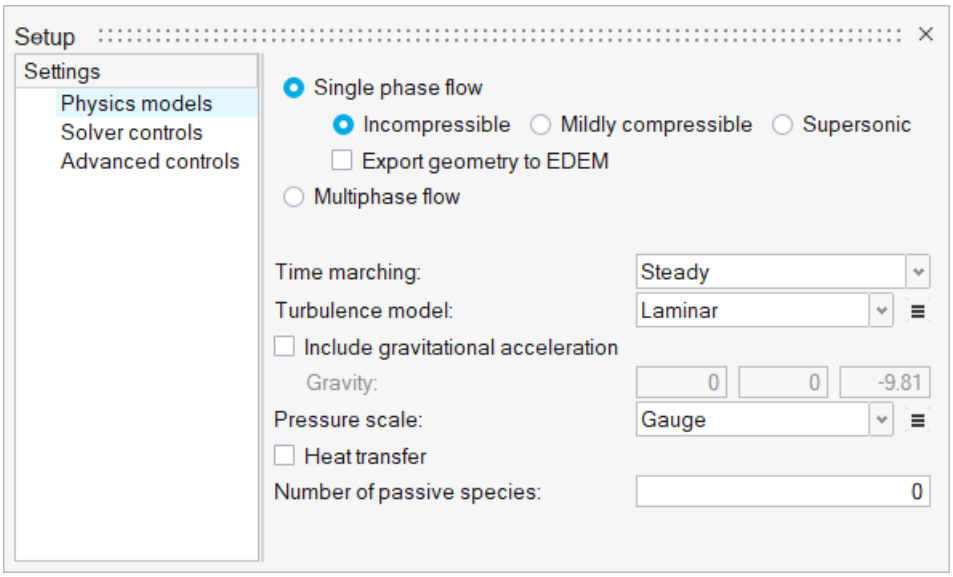


Figure 1704:

3. Click the **Solver controls** setting.
4. Confirm that the Steady update factor and the Steady maximum steps are set to `0.6` and `100`, respectively.

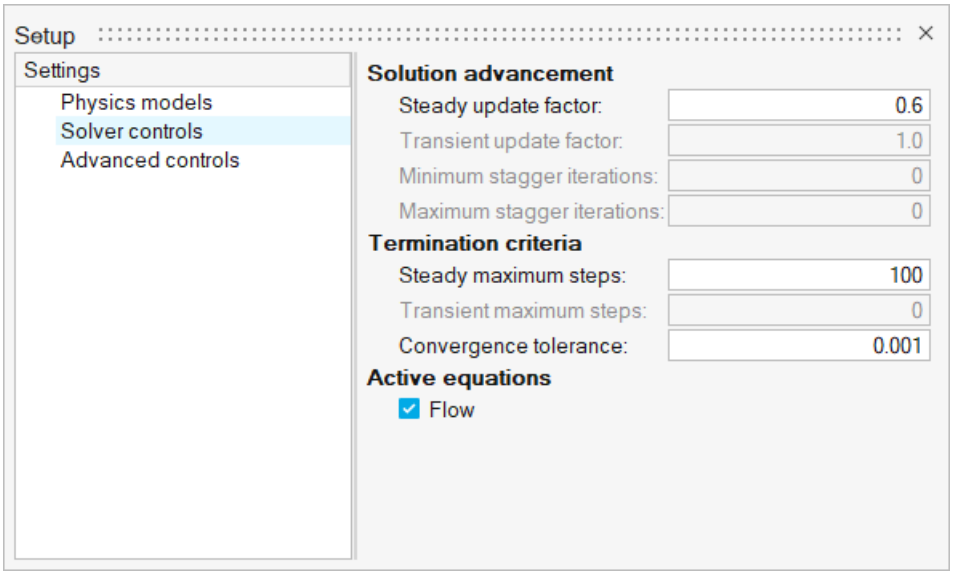


Figure 1705:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1706:

2. Verify that the material Air is assigned to the model's three solids.

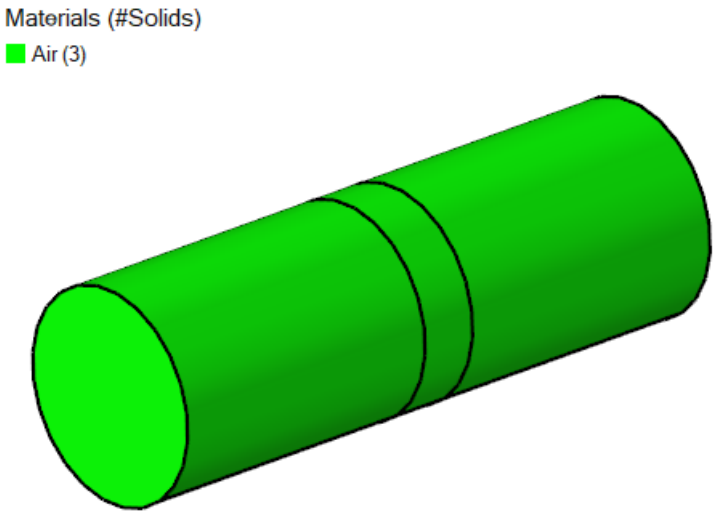


Figure 1707:

Define the Porous Medium

1. From the Flow ribbon, Porous tool group, click the **Cartesian Porous Media** tool.

Figure 1708:

2. Select the middle solid on the model.

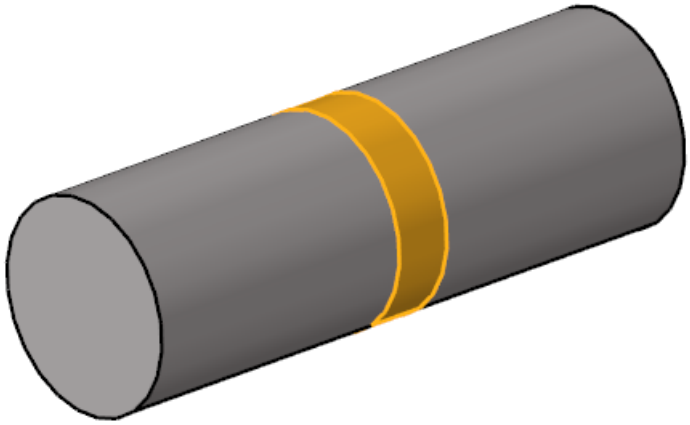


Figure 1709:

3. On the guide bar, click **Orientation**.
4. Left-click to place a point anywhere on the selected solid.
5. In the microdialog, click the coefficient calculator and enter the following values for the coefficient calculator.

   porous zone thickness = 0.06 (m)

   air density = 1.225 kg/m^3

   air viscosity = 1.781e-05 kg/m-sec

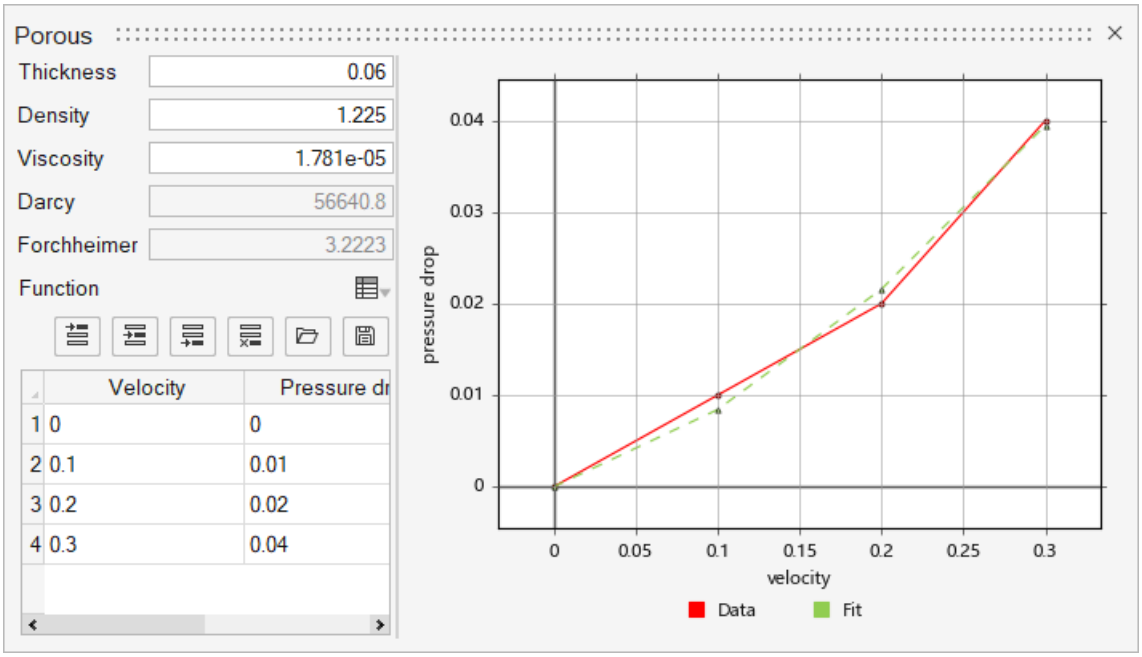


Figure 1710:

6. Close the coefficient calculator and confirm the values of the Darcy coefficient and Forchheimer coefficient. Enter permeability values for the three directions as shown in the figure below

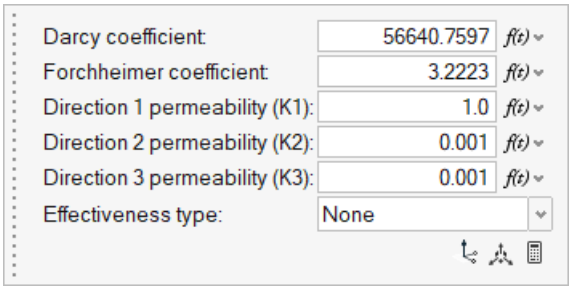


Figure 1711:

7. In the microdialog, click to open the Orient tool then verify that the direction is aligned to the global x-axis.

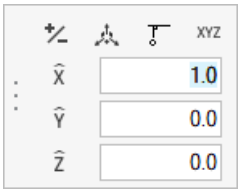


Figure 1712:

8. On the guide bar, click ✓ to execute the command and exit the tool.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, click the **Constant** tool.

Figure 1713:

2. Select the inlet face.

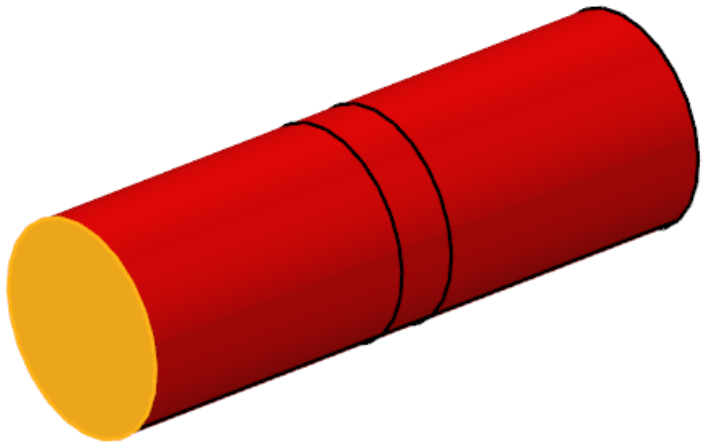


Figure 1714:

3. In the microdialog, set the velocity parameters as shown below.

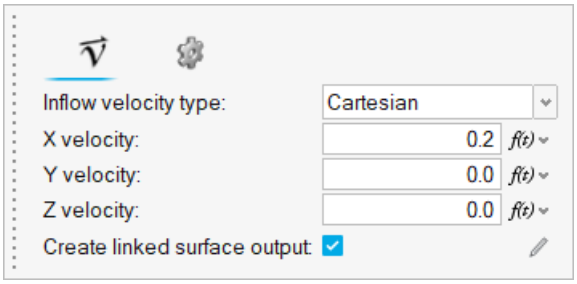


Figure 1715:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 1716:

6. Select the outlet face.

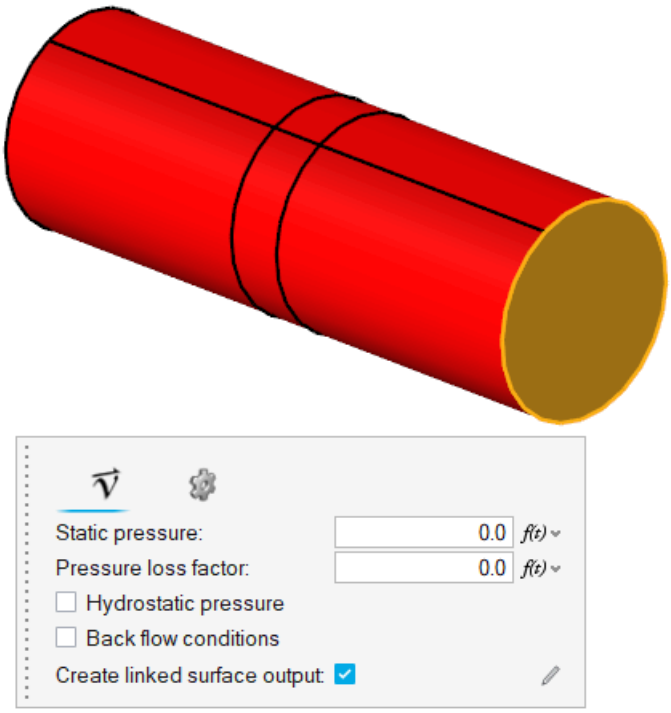


Figure 1717:

7. Accept the default parameters then click ✓ on the guide bar.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

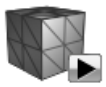


Figure 1718:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.01.
3. Accept all other default parameters.

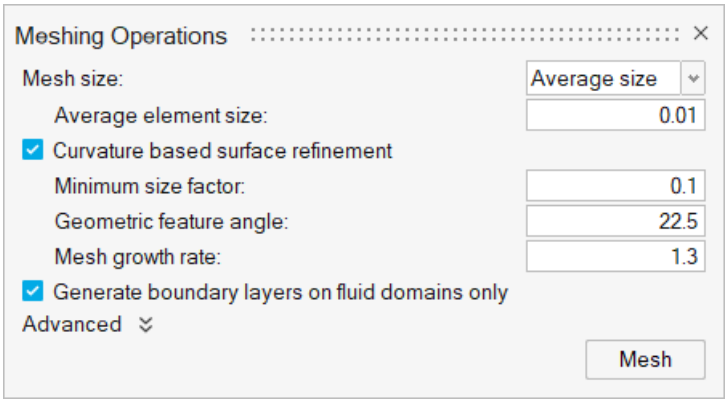


Figure 1719:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1720:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Leave the remaining options as default and click **Run** to launch AcuSolve.

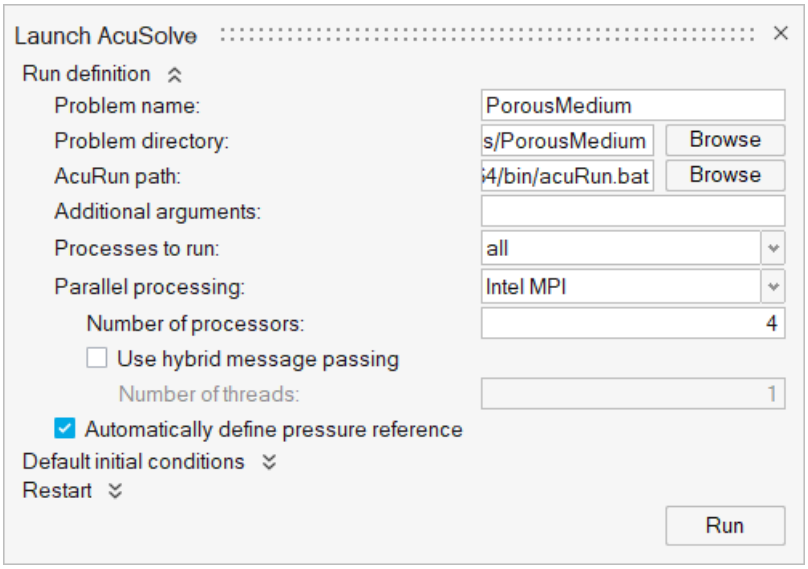


Figure 1721:

Post-Process with the Plot Tool

1. In the **Run Status** dialog, right-click on AcuSolve run and select **Plot time history** to launch the **Plot Manager**.

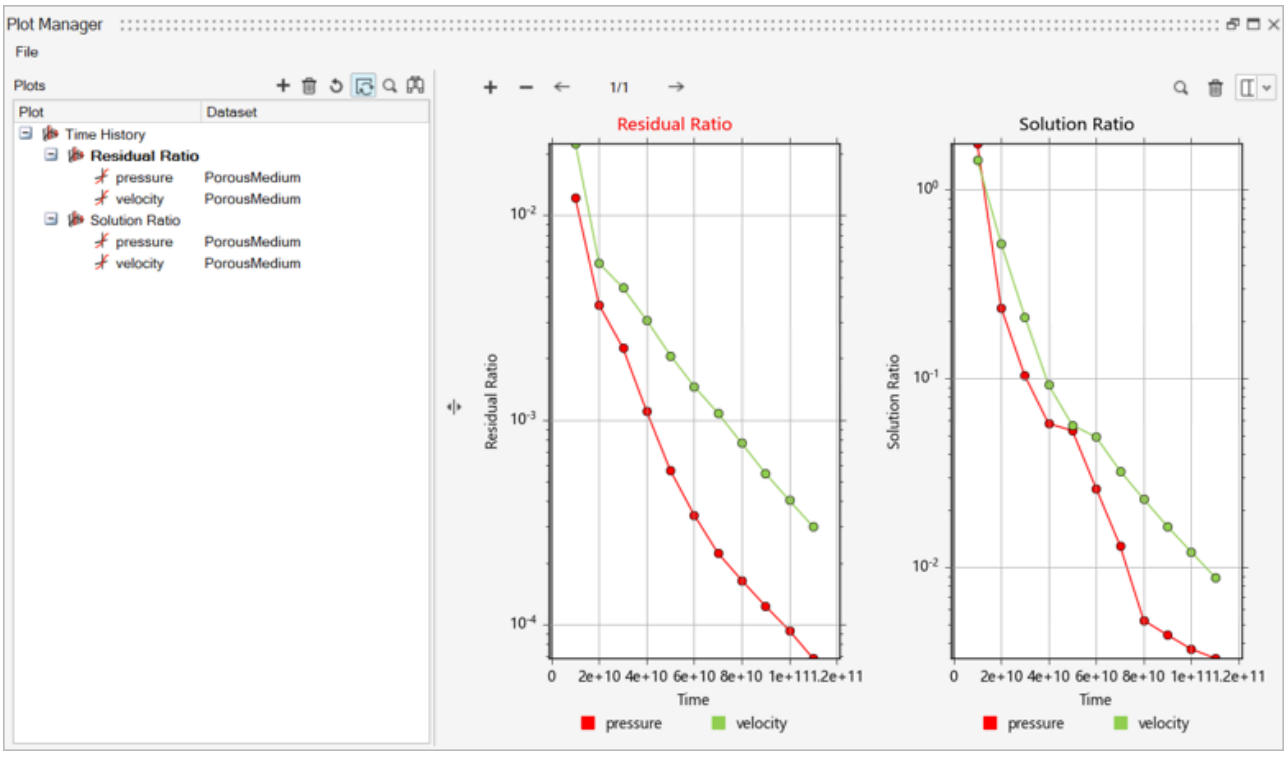


Figure 1722:

2. Click + to create a new plot.
3. Click under the Model panel to create a user defined function.
4. In the Data Tree, expand **Integrated Surface Output** > **Pressure** then select **pressure**.
5. In the Name field, enter `dP`.
6. In the Value field, type `value=`.
7. Under Parts, choose **AUTO Porous-1 SolidBody_2_1 internal** then click **Insert Field** to insert the field as part of the value, as shown below.

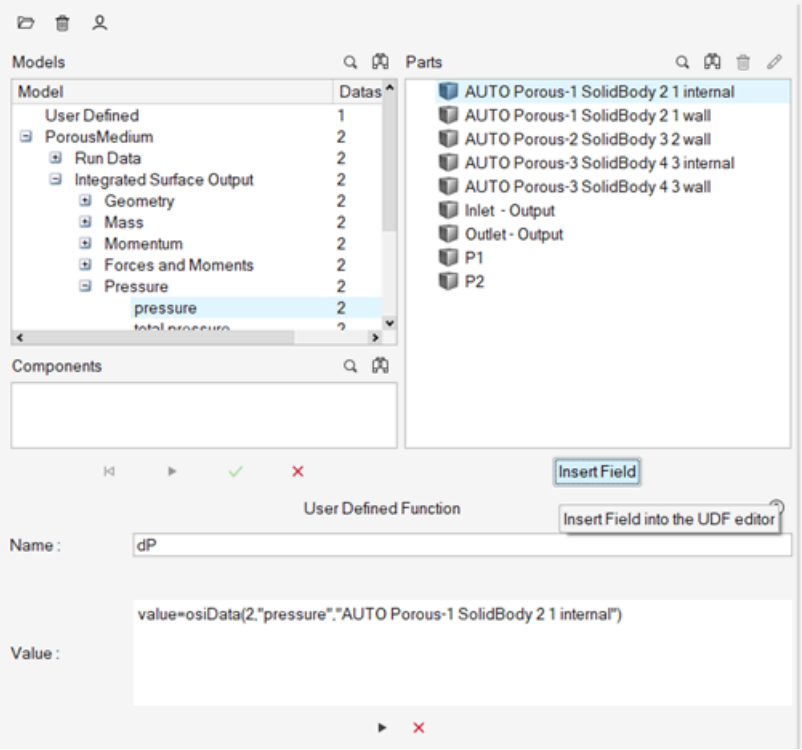


Figure 1723:

8. Type – at the end of the user defined function value.
9. Under Parts, choose **AUTO Porous-3 SolidBody_4_3 internal** then click **Insert Field** to insert the field as part of the value.
10. Click ▸ to add the user defined function.

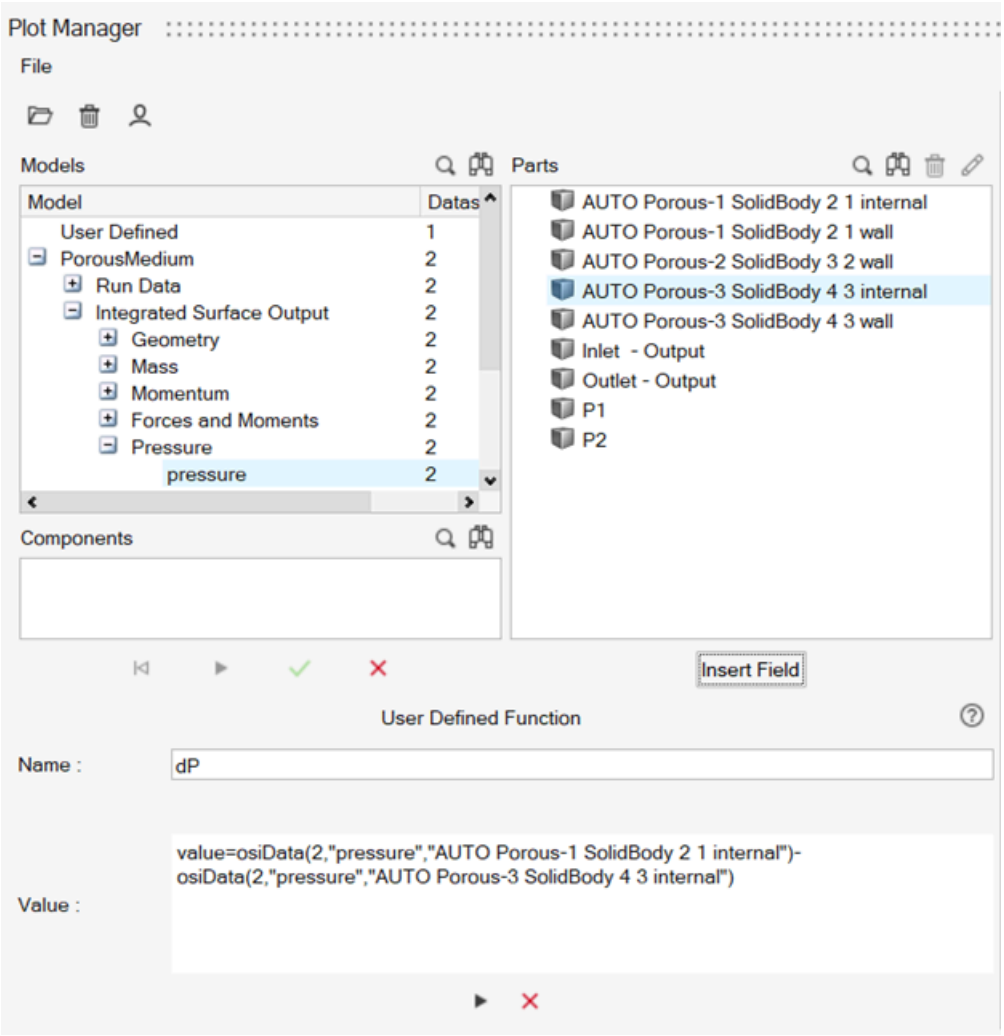


Figure 1724:

11. Click the **x axis** to switch to Time Steps.

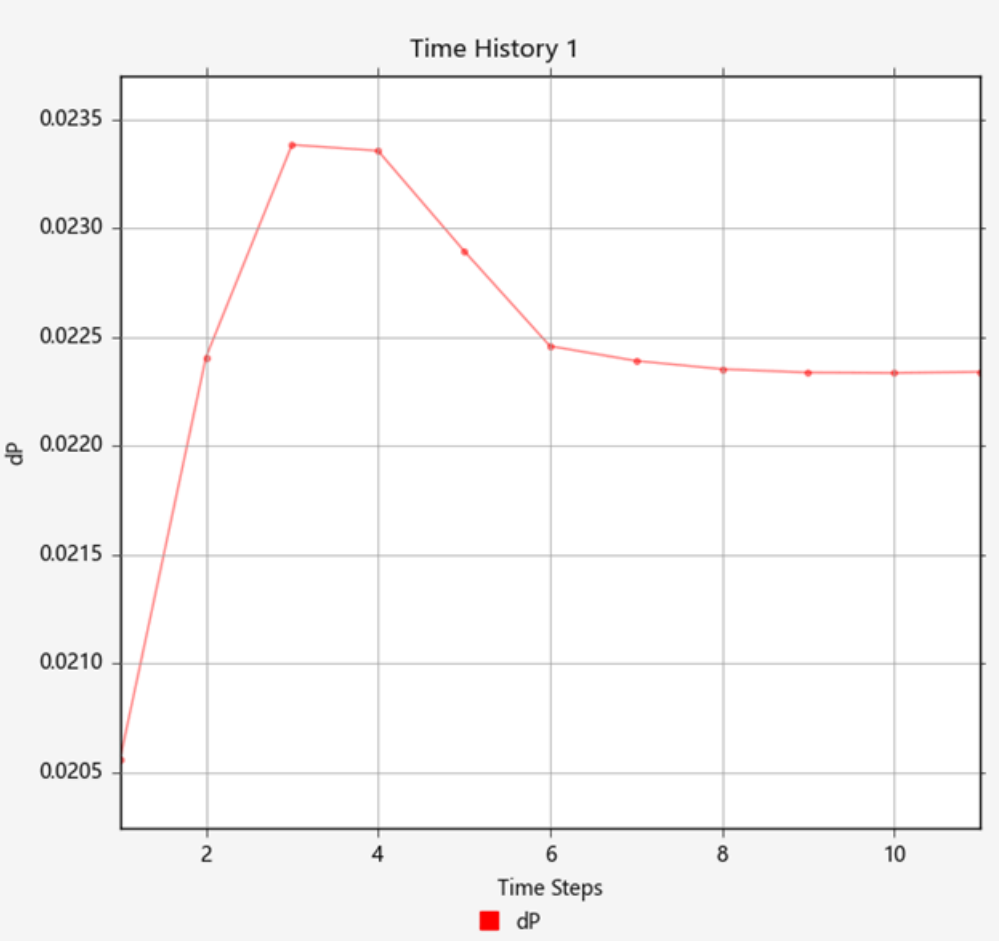


Figure 1725:

The pressure drop across the porous inlet (AUTO Porous-1 SolidBody_2_1 internal) and porous outlet (AUTO Porous-3 SolidBody_4_3 internal) surfaces is 0.0223 Pa.

Summary

In this tutorial, you learned how to set up and solve a flow simulation with porous medium. You started by importing the HyperMesh CFD input database and then you defined the porous medium. You also learned how to compute the Darcy and Forchheimer coefficients using the calculator available in HyperMesh CFD. Next, you assigned the flow boundary conditions and generated the mesh. Once the solution was computed, you created a plot of the pressure drop across the porous section using the Plot tool.

SimLab

Tutorial Level: Beginner

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.

2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/k047d19iw3

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 6501 Flow Through Porous Medium with Physical Velocity

Perform this tutorial using the following user interface(s):

HyperMesh CFD

HyperMesh CFD

Tutorial Level: Intermediate

Prerequisites

This tutorial provides the instructions for setting up, solving, and viewing results for a simulation of a flow through porous medium that is specified using the physical velocity input method. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction, and have a basic understanding of HyperMesh CFD and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh CFD and AcuSolve.

Before you begin, copy the file(s) used in this tutorial to your working directory.

`ACU-T6501_PorousMediaPhysicalVelocity.hm`

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a cylindrical channel with a porous medium in the flow section. As the flow passes through this section, a pressure drop is observed. In this simulation, an inlet velocity will be assigned to the flow and pressure drop across the porous medium will be calculated. The length of the porous section is 0.06 m and the fluid is defined as air fluid with a density of 1.225 kg/m^3 and a molecular viscosity of 1.781e-5 kg/m-s. The inlet velocity of the flow is 0.2 m/s.

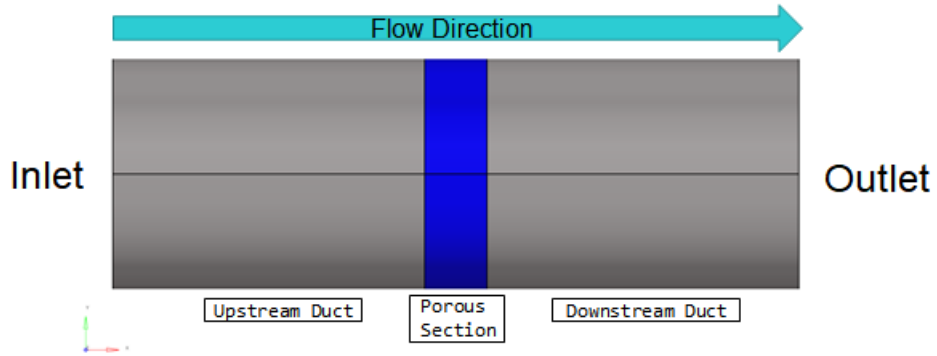


Figure 1726:

The porosity is defined as the ratio of the volume of fluid to the total volume. With the superficial velocity approach, the porosity is set to 1 and the flow velocity is calculated as if there is no obstruction to the flow. When using the porous medium with physical velocity, the porosity of the volume is

considered when calculating the flow velocity. This allows for a more physical representation of the flow within these components. In this tutorial, you will use a porosity value of 0.5.

Start HyperMesh CFD and Open the HyperMesh Database

1. Start HyperMesh CFD from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh CFD**.
2. From the Home tools, Files tool group, click the **Open Model** tool.

Figure 1727:

The **Open File** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6501_PorousMediaPhysicalVelocity.hm` and click **Open**.
4. Click **File** > **Save As**.
5. Create a new directory named `PorousMediaPhysical` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `PorousMediaPhysical` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Validate the Geometry

The Validate tool scans through the entire model, performs checks on the surfaces and solids, and flags any defects in the geometry, such as free edges, closed shells, intersections, duplicates, and slivers.

To focus on the physics part of the simulation, this tutorial input file contains geometry which has already been validated. Observe that a blue check mark appears on the top-left corner of the Validate icon on the Geometry ribbon. This indicates that the geometry is valid, and you can go to the flow set up.

Figure 1728:

Set Up Flow

Set Up the Simulation Parameters and Solver Settings

1. From the Flow ribbon, click the **Physics** tool.

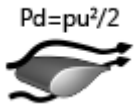


Figure 1729:

The **Setup** dialog opens.

2. Under the Physics models setting:
   a) Set Time marching to **Steady**.
   b) Select **Laminar** as the Turbulence model.

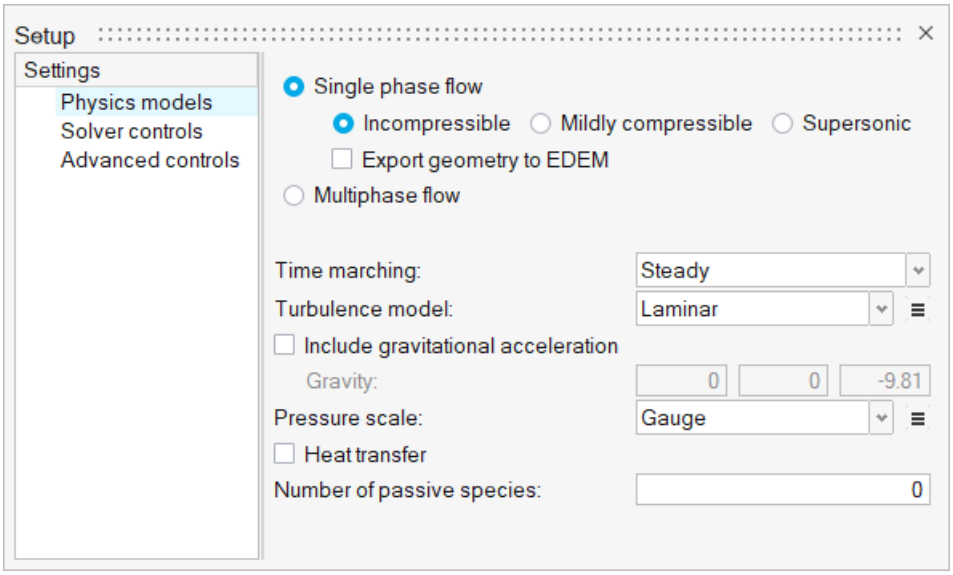


Figure 1730:

3. Click the **Solver controls** setting.
4. Confirm that the Steady update factor and the Steady maximum steps are set to `0.6` and `100`, respectively.

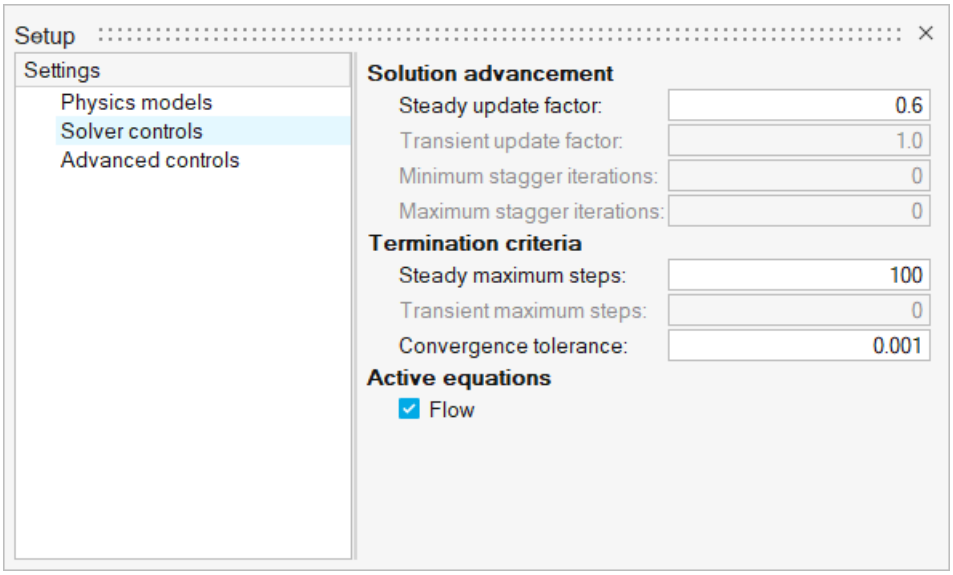


Figure 1731:

5. Click the **Advanced controls** setting.
6. Confirm that the Porous media velocity type is set to **Physical**.

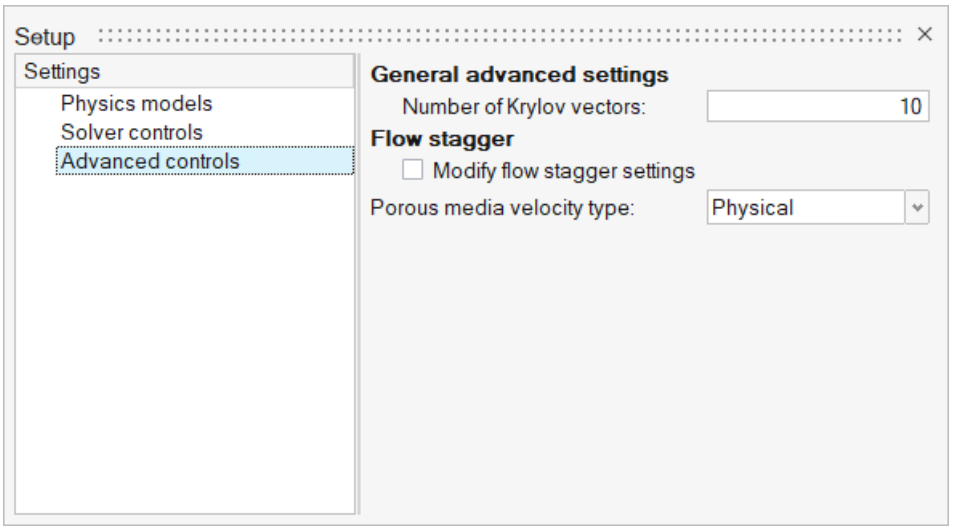


Figure 1732:

Assign Material Properties

1. From the Flow ribbon, click the **Material** tool.

Figure 1733:

2. Verify that the material Air is assigned to the model's three solids.

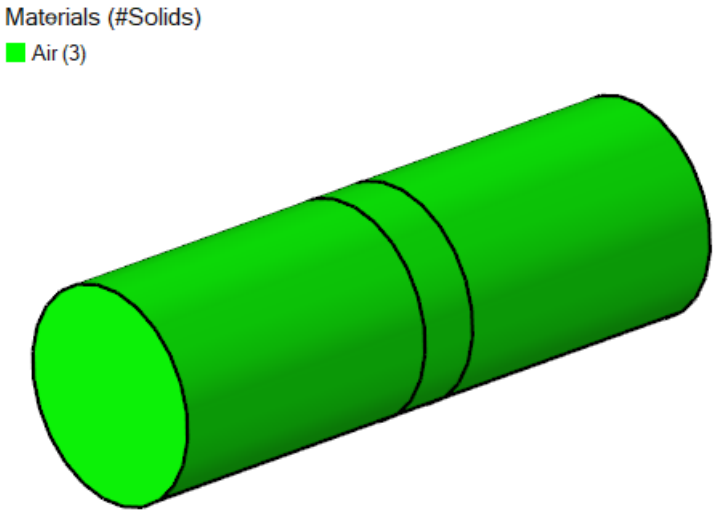


Figure 1734:

Define the Porous Medium

1. From the Flow ribbon, Porous tool group, click the **Cartesian Porous Media** tool.

Figure 1735:

2. Select the middle solid on the model.

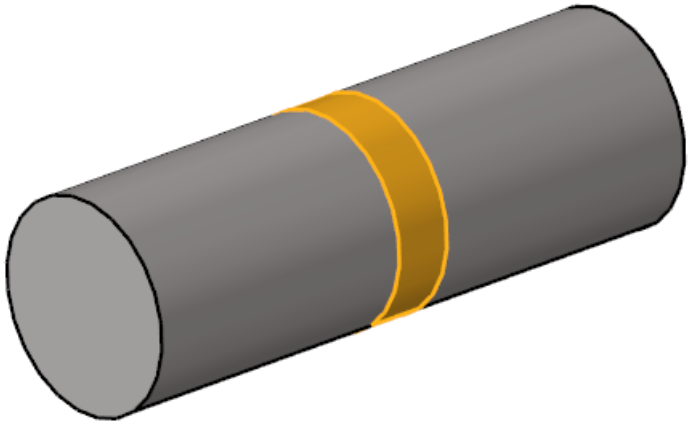


Figure 1736:

3. On the guide bar, click **Orientation**.
4. Left-click to place a point anywhere on the selected solid.
5. In the microdialog, enter the following values for the coefficients.

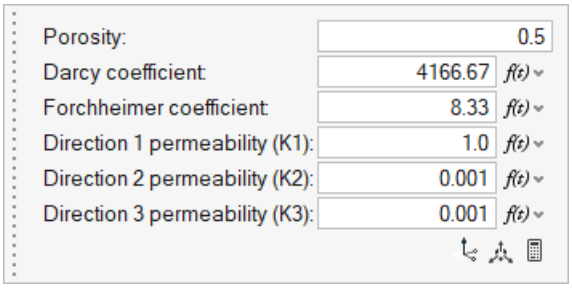


Figure 1737:

6. In the microdialog, click to open the Orient tool then verify that the direction is aligned to the global x-axis.

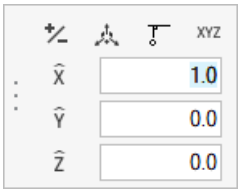


Figure 1738:

7. On the guide bar, click ✓ to execute the command and exit the tool.

Assign the Flow Boundary Conditions

1. From the Flow ribbon, Profiled tool group, click the **Profiled Inlet** tool.

Figure 1739:

2. Select the inlet face.

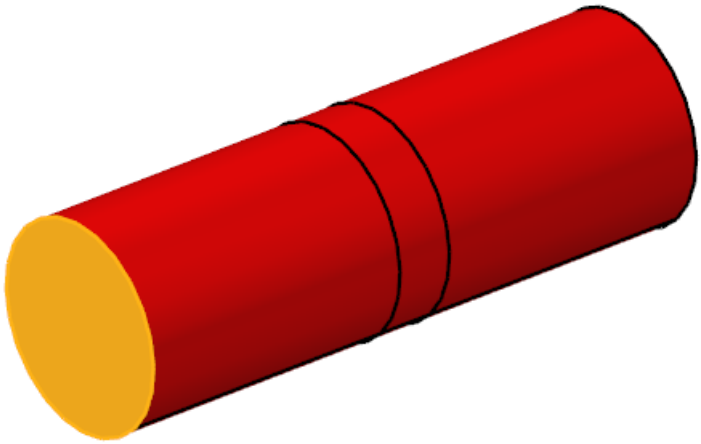


Figure 1740:

3. In the microdialog, set the average velocity to `0.2`.

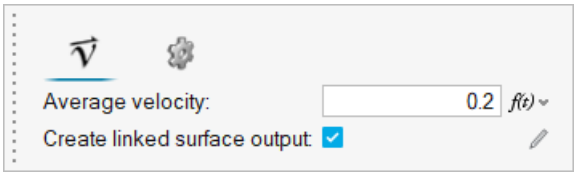


Figure 1741:

4. On the guide bar, click ✓ to execute the command and exit the tool.
5. Click the **Outlet** tool.

Figure 1742:

6. Select the outlet face.

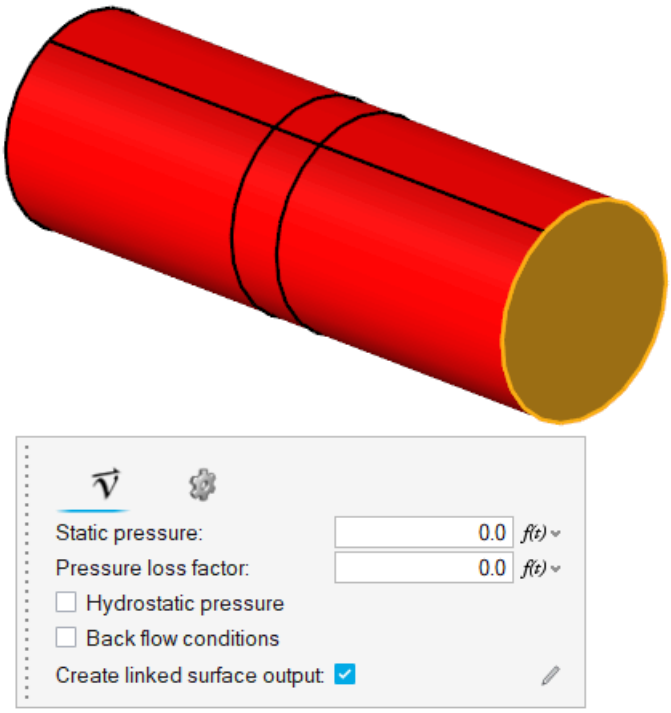


Figure 1743:

7. Accept the default parameters then click ✓ on the guide bar.

Generate the Mesh

The meshing parameters for this tutorial are already set in the input file.

1. From the Mesh ribbon, click the **Volume** tool.

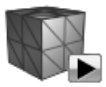


Figure 1744:

The **Meshing Operations** dialog opens.

> **Note:** If the model has not been validated, you are prompted to create the simulation model before running the batch mesh.

2. Check that the Average element size is 0.01.
3. Accept all other default parameters.

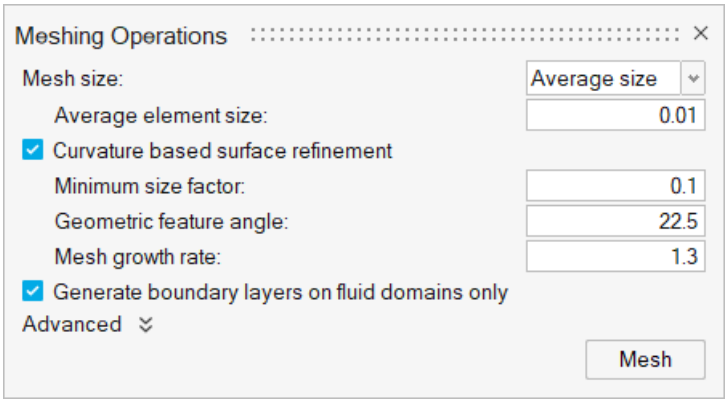


Figure 1745:

4. Click **Mesh**.
   The **Run Status** dialog opens. Once the run is complete, the status is updated and you can close the dialog.

   Tip: Right-click on the mesh job and select **View log file** to view a summary of the meshing process.

Run AcuSolve

1. From the Solution ribbon, click the **Run** tool.

Figure 1746:

2. Set the Parallel processing option to **Intel MPI**.
3. Optional: Set the number of processors to **4** or **8** based on availability.
4. Leave the remaining options as default and click **Run** to launch AcuSolve.

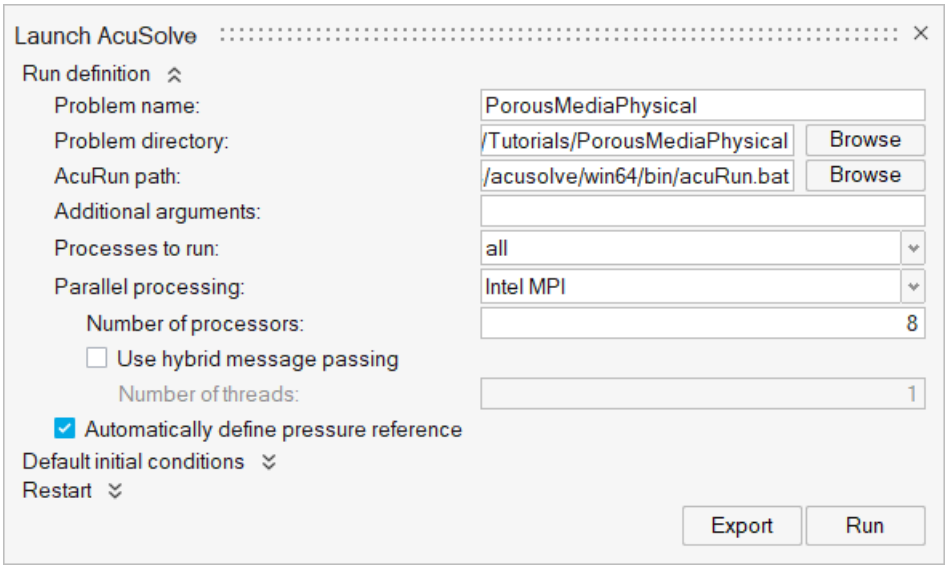


Figure 1747:

Post-Process the Results

1. In the **Run Status** dialog, right-click on AcuSolve run and select **Plot time history** to launch the **Plot Manager**.

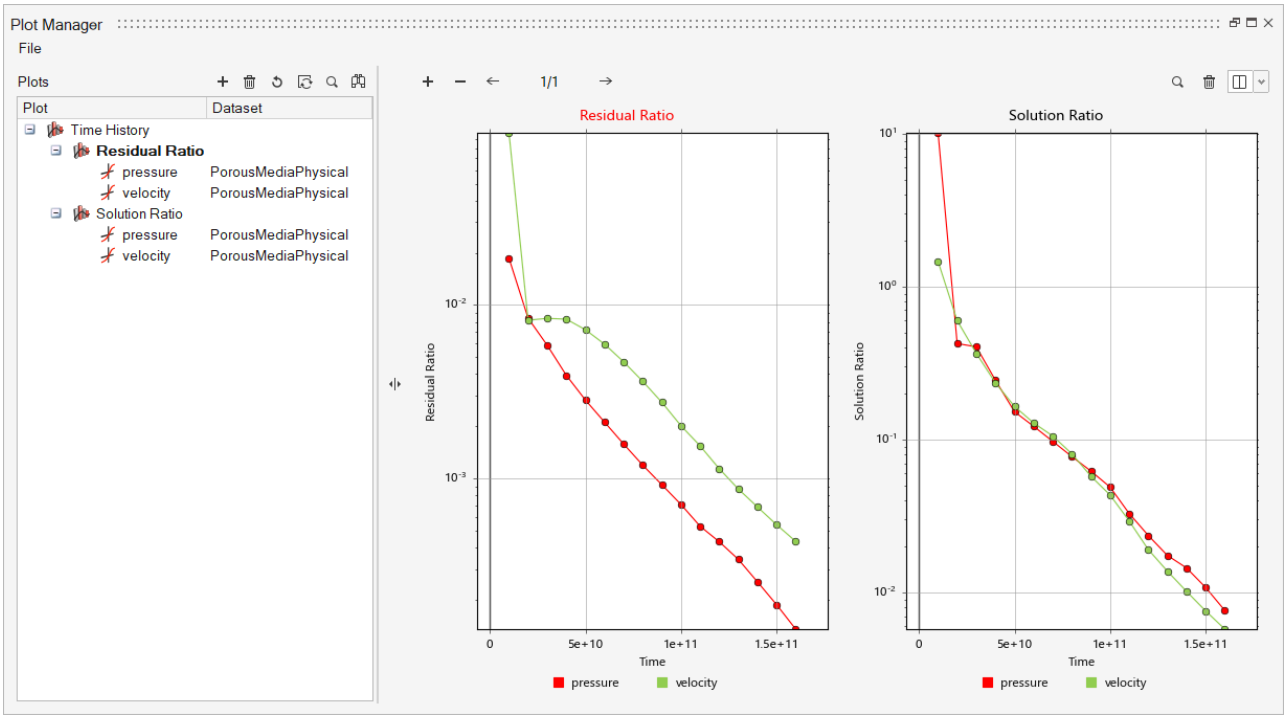


Figure 1748:

2. Click + to create a new plot.
3. Right-click **User Defined** under the Model panel to create a user-defined function.
4. In the Data Tree, expand **Integrated Surface Output** > **Pressure** and then select **pressure**.
5. In the Name field, enter `dP`.
6. In the Value field, type `value=`.
7. Under Parts, choose **AUTO Porous-1 SolidBody_2_1 internal** and then click to insert the field as part of the value, as shown below.

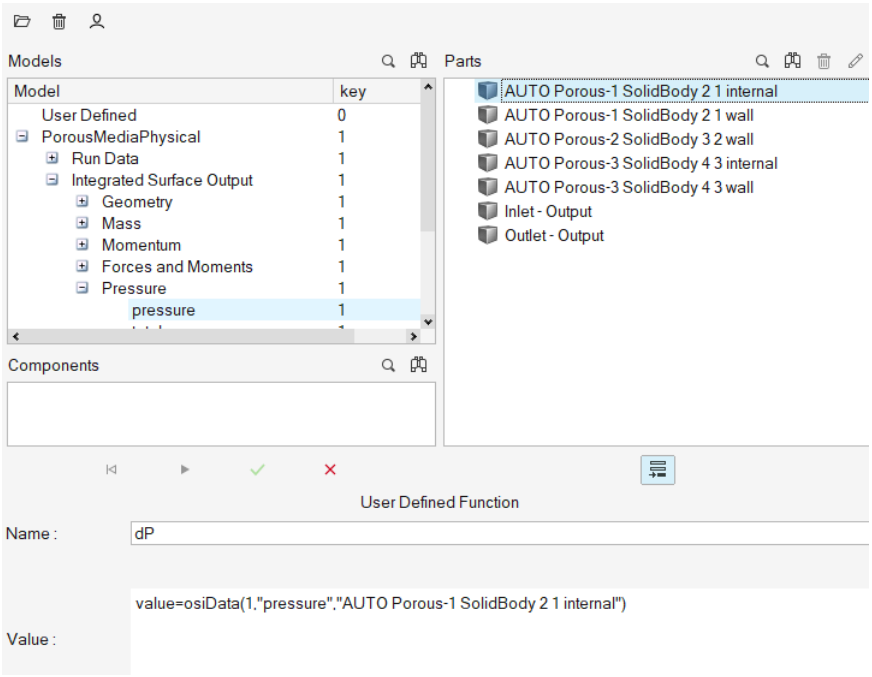


Figure 1749:

8. Type – at the end of the user-defined function value.
9. Under Parts, choose **AUTO Porous-3 SolidBody_4_3 internal** and then click to insert the field as part of the value.
10. Click ▸ to add the user-defined function.

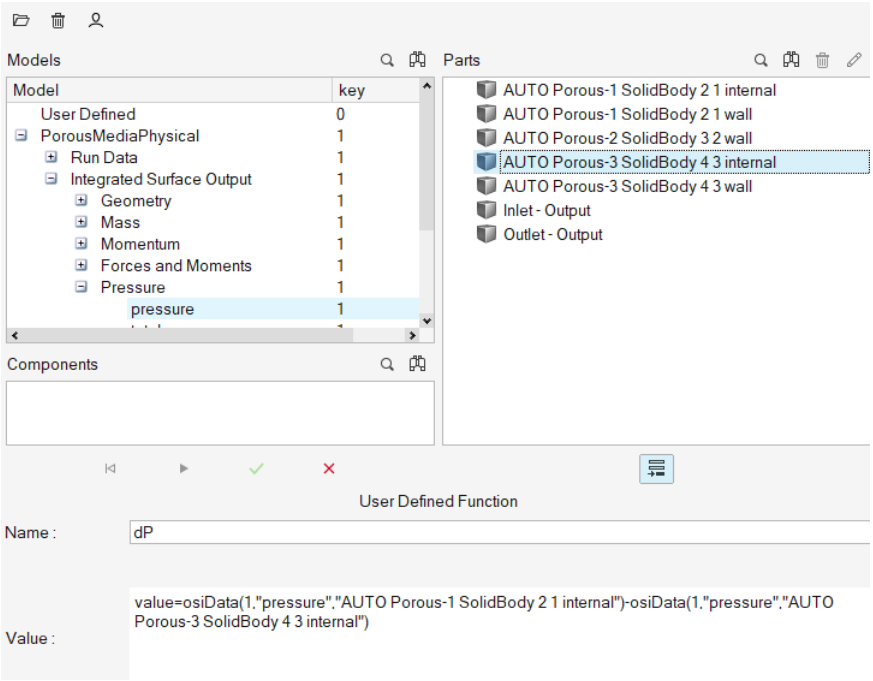


Figure 1750:

11. Click the **x axis** to switch to Time Steps.

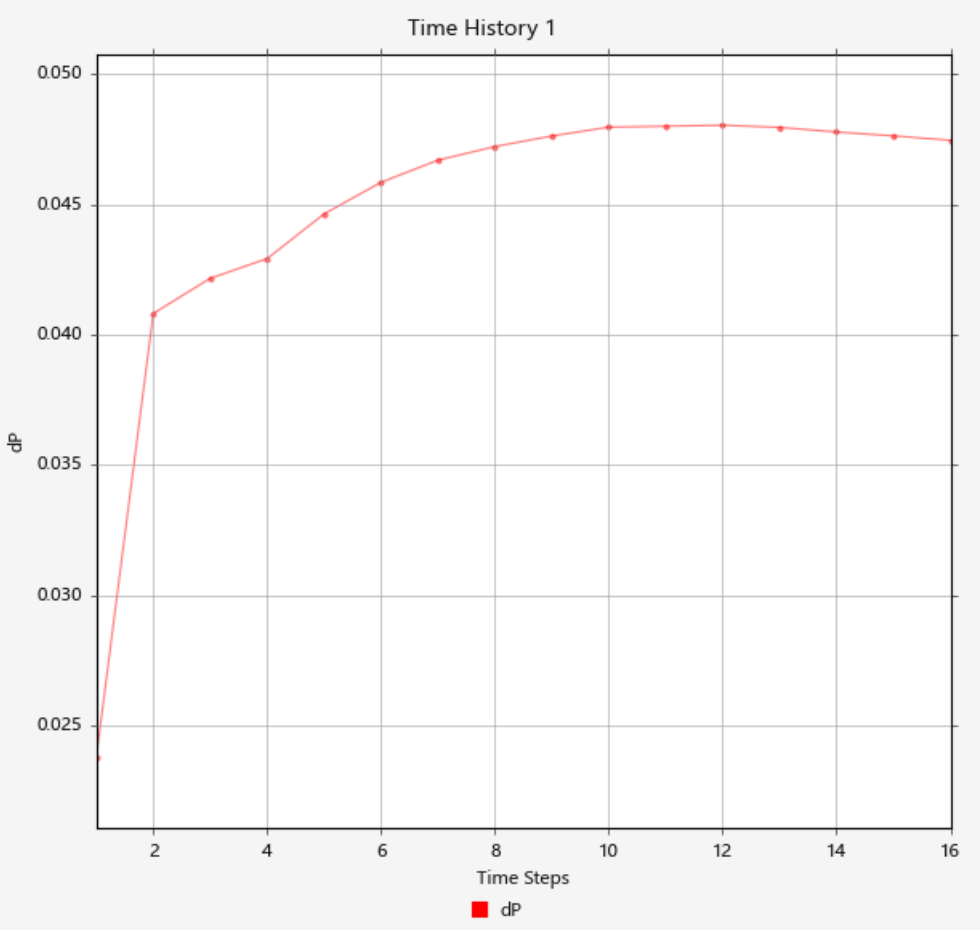


Figure 1751:

The pressure drop across the porous inlet (AUTO Porous-1 SolidBody_2_1 internal) and porous outlet (AUTO Porous-3 SolidBody_4_3 internal) surfaces is 0.047 Pa.

Summary

In this tutorial, you learned how to set up and solve a flow simulation with porous medium using the physical velocity formulation. This implementation allows you to specify values of porosity in the fluid to emulate the fluid packing within a porous media. You started by importing the HyperMesh CFD input database and then you defined the porous medium. Next, you assigned the flow boundary conditions and generated the mesh. Once the solution was computed, you created a plot of the pressure drop across the porous section using HyperMesh CFD.

Optimization

8

This chapter covers the following:

- ACU-T: 7200 Topology Optimization (p. 1046)
- ACU-T: 7201 Topology Optimization with Flow Distribution (p. 1047)
- ACU-T: 7202 CFD Topology Optimization of Multiple Flow Paths (p. 1048)
- ACU-T: 7300 DOE using Inspire, SimLab and HyperStudy (p. 1049)
- ACU-T: 7301 Solution DOE (p. 1050)

ACU-T: 7200 Topology Optimization

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/k9j843mgyw

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 7201 Topology Optimization with Flow Distribution

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Intermediate

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/q39ahm8g8d

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 7202 CFD Topology Optimization of Multiple Flow Paths

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 3 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.
3. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/e5lpaxz5cs

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 7300 DOE using Inspire, SimLab and HyperStudy

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in 2 different ways:

1. Complete tutorial - CAD to Results

   This provides a complete workflow starting from importing the CAD, generating mesh, and setting up the simulation to visualizing the results. To begin here, download the CAD geometry.
2. Mesh Generation - CAD to Mesh

   You can navigate directly to the Mesh Generation section and work from importing the CAD to generating the mesh. Afterwards you can continue with the solution setup. To begin here, download the CAD geometry.

Tutorial Video

https://fast.wistia.net/embed/iframe/slysr9cyr3

Additional Resources

For more tutorials, visit the SimLab Learning Center.

ACU-T: 7301 Solution DOE

Perform this tutorial using the following user interface(s):

SimLab

SimLab

Tutorial Level: Advanced

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 UI Introduction. To run this simulation, you will need access to a licensed version of SimLab and AcuSolve. You will need an internet connection to watch the video tutorials and download the files.

You can work through this tutorial in the following way:

1. Solution Setup - Mesh to Results

   You can navigate to the Solution Setup section and work from setting up the simulation to visualizing the results. To begin here, download the SimLab database.

Tutorial Video

https://fast.wistia.net/embed/iframe/1uq8yy7kid

Additional Resources

For more tutorials, visit the SimLab Learning Center.

Intellectual Property Rights Notice

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

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

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

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

Altair® HyperWorks®, a Design & Simulation Platform

Altair® AcuSolve® ©1997-2024

Altair® Activate® ©1989-2024

Altair® Automated Reporting Director™ ©2008-2022

Altair® Battery Damage Identifier™ ©2019-2024

Altair® CFD™ ©1990-2024

Altair Compose® ©2007-2024

Altair® ConnectMe™ ©2014-2024

Altair® DesignAI™ ©2022-2024

Altair® DSim™ ©2024

Altair® EDEM™ ©2005-2024

Altair® EEvision™ ©2018-2024

Altair® ElectroFlo™ ©1992-2024

Altair Embed® ©1989-2024

Altair Embed® SE ©1989-2024

Altair Embed®/Digital Power Designer ©2012-2024

Altair Embed®/eDrives ©2012-2024

Altair Embed® Viewer©1996-2024

Altair® e-Motor Director™©2019-2024

Altair® ESAComp® ©1992-2024

Altair® expertAI™ ©2020-2024

Altair® Feko® ©1999-2024

Altair® FlightStream® ©2017-2024

Altair® Flow Simulator™ ©2016-2024

Altair® Flux® ©1983-2024

Altair® FluxMotor® ©2017-2024

Altair® GateVision PRO™ ©2002-2024

Altair® Geomechanics Director™ ©2011-2022

Altair® HyperCrash® ©2001-2023

Altair® HyperGraph® ©1995-2024

Altair® HyperLife® ©1990-2024

Altair® HyperMesh® ©1990-2024

Altair® HyperMesh® CFD ©1990-2024

Altair® HyperMesh ® NVH ©1990-2024

Altair® HyperSpice™ ©2017-2024

Altair® HyperStudy® ©1999-2024

Altair® HyperView® ©1999-2024

Altair® HyperView Player® ©2022-2024

Altair® HyperWorks® ©1990-2024

Altair® HyperWorks® Design Explorer ©1990-2024

Altair® HyperXtrude® ©1999-2024

Altair® Impact Simulation Director™ ©2010-2022

Altair® Inspire™ ©2009-2024

Altair® Inspire™ Cast ©2011-2024

Altair® Inspire™ Extrude Metal ©1996-2024

Altair® Inspire™ Extrude Polymer ©1996-2024

Altair® Inspire™ Form ©1998-2024

Altair® Inspire™ Mold ©2009-2024

Altair® Inspire™ PolyFoam ©2009-2024

Altair® Inspire™ Print3D ©2021-2024

Altair® Inspire™ Render ©1993-2024

Altair® Inspire™ Studio ©1993-2024

Altair® Material Data Center™ ©2019-2024

Altair® Material Modeler™ ©2019-2024

Altair® Model Mesher Director™ ©2010-2024

Altair® MotionSolve® ©2002-2024

Altair® MotionView® ©1993-2024

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

Altair® Multiscale Designer® ©2011-2024

Altair® newFASANT™ ©2010-2020

Altair® nanoFluidX® ©2013-2024

Altair® NLVIEW® ©2018-2024

Altair® NVH Director™ ©2010-2024

Altair® NVH Full Vehicle™ ©2022-2024

Altair® NVH Standard™ ©2022-2024

Altair® OmniV™ ©2015-2024

Altair® OptiStruct® ©1996-2024

Altair® physicsAI™ ©2021-2024

Altair® PollEx™ ©2003-2024

Altair® PollEx™ for ECAD ©2003-2024

Altair® PSIM™ ©1994-2024

Altair® Pulse™ ©2020-2024

Altair® Radioss® ©1986-2024

Altair® romAI™ ©2022-2024

Altair® RTLvision PRO™ ©2002-2024

Altair® S-CALC™ ©1995-2024

Altair® S-CONCRETE™ ©1995-2024

Altair® S-FRAME® ©1995-2024

Altair® S-FOUNDATION™ ©1995-2024

Altair® S-LINE™ ©1995-2024

Altair® S-PAD™ ©1995-2024

Altair® S-STEEL™ ©1995-2024

Altair® S-TIMBER™ ©1995-2024

Altair® S-VIEW™ ©1995-2024

Altair® SEAM® ©1985-2024

Altair® shapeAI™ ©2021-2024

Altair® signalAI™ ©2020-2024

Altair® Silicon Debug Tools™ ©2018-2024

Altair® SimLab® ©2004-2024

Altair® SimLab® ST ©2019-2024

Altair® SimSolid® ©2015-2024

Altair® SpiceVision PRO™ ©2002-2024

Altair® Squeak and Rattle Director™ ©2012-2024

Altair® StarVision PRO™ ©2002-2024

Altair® Structural Office™ ©2022-2024

Altair® Sulis™ ©2018-2024

Altair® Twin Activate® ©1989-2024

Altair® UDE™ ©2015-2024

Altair® ultraFluidX® ©2010-2024

Altair® Virtual Gauge Director™ ©2012-2024

Altair® Virtual Wind Tunnel™ ©2012-2024

Altair® Weight Analytics™ ©2013-2022

Altair® Weld Certification Director™ ©2014-2024

Altair® WinProp™ ©2000-2024

Altair® WRAP™ ©1998-2024

Altair® HPCWorks®, a HPC & Cloud Platform

Altair® Allocator™ ©1995-2024

Altair® Access™ ©2008-2024

Altair® Accelerator™ ©1995-2024

Altair® Accelerator™ Plus ©1995-2024

Altair® Breeze™ ©2022-2024

Altair® Cassini™ ©2015-2024

Altair® Control™ ©2008-2024

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

Altair® FlowTracer™ ©1995-2024

Altair® Grid Engine® ©2001, 2011-2024

Altair® InsightPro™ ©2023-2024

Altair® Hero™ ©1995-2024

Altair® Liquid Scheduling™ ©2023-2024

Altair® Mistral™ ©2022-2024

Altair® Monitor™ ©1995-2024

Altair® NavOps® ©2022-2024

Altair® PBS Professional® ©1994-2024

Altair® PBS Works™ ©2022-2024

Altair® Simulation Cloud Suite (SCS) ©2024

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

Altair® Unlimited™ ©2022-2024

Altair® Unlimited Data Analytics Appliance™ ©2022-2024

Altair® Unlimited Virtual Appliance™ ©2022-2024

Altair® RapidMiner®, a Data Analytics & AI Platform

Altair® AI Hub ©2023-2024

Altair® AI Edge™ ©2023-2024

Altair® AI Cloud ©2022-2024

Altair® AI Studio ©2023-2024

Altair® Analytics Workbench™ ©2002-2024

Altair® Graph Lakehouse™ ©2013-2024

Altair® Graph Studio™ ©2007-2024

Altair® Knowledge Hub™ ©2017-2024

Altair® Knowledge Studio® ©1994-2024

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

Altair® Knowledge Seeker™ ©1994-2024

Altair® IoT Studio™ ©2002-2024

Altair® Monarch® ©1996-2024

Altair® Monarch® Classic ©1996-2024

Altair® Monarch® Complete™ ©1996-2024

Altair® Monarch® Data Prep Studio ©2015-2024

Altair® Monarch Server™ ©1996-2024

Altair® Panopticon™ ©2004-2024

Altair® Panopticon™ BI ©2011-2024

Altair® SLC™ ©2002-2024

Altair® SLC Hub™ ©2002-2024

Altair® SmartWorks™ ©2002-2024

Altair® RapidMiner® ©2001-2024

Altair One® ©1994-2024

Altair® CoPilot™ ©2023-2024

Altair® License Utility™ ©2010-2024

Altair® TheaRender® ©2010-2024

OpenMatrix™ ©2007-2024

OpenPBS® ©1994-2024

OpenRadioss™ ©1986-2024

October 7, 2024

Technical Support

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

Visit Customer Support to learn more about our support offerings.

Index

A

ACU-T: 1000 / SL-2000 SimLab UI Introduction (SimLab) *43*
ACU-T: 1000 HyperMesh CFD UI Introduction (HyperMesh CFD) *15*
ACU-T: 2000 / SL-2010 Turbulent Flow in a Mixing Elbow (SimLab) *60*
ACU-T: 2000 Turbulent Flow in a Mixing Elbow (HyperMesh CFD) *45*
ACU-T: 2100 / SL-2020 Turbulent Flow Over an Airfoil Using the SST Turbulence Model (SimLab) *94*
ACU-T: 2100 Turbulent Flow Over an Airfoil Using the SST Turbulence Model (HyperMesh CFD) *61*
ACU-T: 2300 / SL 2060 Atmospheric Boundary Layer Problem – Flow Over Building (SimLab) *111*
ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building (HyperMesh CFD) *96*
ACU-T: 2400 / SL 2030 Supersonic Flow in a Converging – Diverging Nozzle (SimLab) *126*
ACU-T: 2400 Supersonic Flow in a Converging-Diverging Nozzle (HyperMesh CFD) *112*
ACU-T: 2500 / SL-2040 Floating Surface Monitor (SimLab) *127*
ACU-T: 3000 / SL-2100 Enclosed Hot Cylinder: Natural Convection (SimLab) *129*
ACU-T: 3100 / SL-2110 Conjugate Heat Transfer in a Mixing Elbow (SimLab) *148*
ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow (HyperMesh CFD) *130*
ACU-T: 3101 / SL-2111 Transient Conjugate Heat Transfer in a Mixing Elbow (SimLab) *163*
ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow (HyperMesh CFD) *149*
ACU-T: 3110 / SL-2400 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping (SimLab) *187*
ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping (HyperMesh CFD) *164*
ACU-T: 3200 / SL-2120 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model (SimLab) *212*
ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model (HyperMesh CFD) *189*
ACU-T: 3201 / SL-2130 Solar Radiation and Thermal Shell Tutorial (SimLab) *241*
ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial (HyperMesh CFD) *213*
ACU-T: 3202 / SL-2140 Heat Transfer Between Concentric Spheres – P1 Radiation Model (SimLab) *255*
ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model (HyperMesh CFD) *242*
ACU-T: 3203 / SL-2141 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model (SimLab) *270*
ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model (HyperMesh CFD) *257*
ACU-T: 3204 / SL-2121 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model (SimLab) *304*
ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model (HyperMesh CFD) *272*
ACU-T: 3300 / SL-2150 Modeling of a Heat Exchanger Component (SimLab) *320*
ACU-T: 3300 Modeling of a Heat Exchanger Component (HyperMesh CFD) *305*
ACU-T: 3310 / SL 2160 Single Phase Nucleate Boiling (SimLab) *334*
ACU-T: 3310 Single Phase Nucleate Boiling (HyperMesh CFD) *321*
ACU-T: 3311 / SL 2161 Multiphase Nucleate Boiling (SimLab) *350*
ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model (HyperMesh CFD) *335*
ACU-T: 3400 / SL-2410 AcuSolve-Flux Integration (SimLab) *352*
ACU-T: 3500 / SL 2170 Electric Potential – Automotive Fuse (SimLab) *354*
ACU-T: 3510 / SL-2171 Battery Thermo - Electric: 2S2P Module (SimLab) *355*
ACU-T: 3511 / SL-2172 Battery Thermo-Electric Pouch Cell (SimLab) *356*

ACU-T: 3512 / SL-2173 Battery Thermal Runaway (SimLab) *357*
ACU-T: 3513 / SL-2176 ARC-Based Thermal Runaway: 1S1P Module (SimLab) *358*
ACU-T: 3600 Melting of Diesel Exhaust Additive within an Enclosed Tank (HyperMesh CFD) *359*
ACU-T: 4000 / SL-2200 Dam Break Simulation (SimLab) *392*
ACU-T: 4000 Transient Dam Break Simulation (HyperMesh CFD) *379*
ACU-T: 4001 / SL-2210 Water Filling in a Tank (SimLab) *407*
ACU-T: 4001 Water Filling in a Tank (HyperMesh CFD) *393*
ACU-T: 4002 / SL 2220 Sloshing of Water in a Tank (SimLab) *421*
ACU-T: 4002 Sloshing of Water in a Tank (HyperMesh CFD) *409*
ACU-T: 4003 / SL 2260 Freely Falling Water Droplet (SimLab) *440*
ACU-T: 4003 Freely Falling Water Droplet (HyperMesh CFD) *423*
ACU-T: 4100 / SL-2230 Multiphase Flow using Algebraic Eulerian Model (SimLab) *453*
ACU-T: 4100 Disperse Case – LPipe (HyperMesh CFD) *442*
ACU-T: 4101 T-Junction Flow using the Eulerian Multiphase Model (HyperMesh CFD) *455*
ACU-T: 4102 Fluidized Bed using the Granular Multiphase Model (HyperMesh CFD) *470*
ACU-T: 4200 / SL-2240 Humidity – Pipe Junction (SimLab) *501*
ACU-T: 4200 Humidity – Pipe Junction (HyperMesh CFD) *487*
ACU-T: 4201 / SL-2250 Condensation & Evaporation - Air Box (SimLab) *521*
ACU-T: 4201 Condensation & Evaporation - Air Box (HyperMesh CFD) *502*
ACU-T: 4300 Species Transport Modeling (HyperMesh CFD) *523*
ACU-T: 5000 / SL-2300 Centrifugal Air Blower with Moving Reference Frame (Steady) (SimLab) *558*
ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady) (HyperMesh CFD) *542*
ACU-T: 5001 / SL-2301 Blower - Transient (Sliding Mesh) (SimLab) *573*
ACU-T: 5001 Blower - Transient (Sliding Mesh) (HyperMesh CFD) *560*
ACU-T: 5100 / SL-2310 Modeling of a Fan Component Using the Fan Component - Coefficient Method (SimLab) *598*
ACU-T: 5100 Modeling of a Fan Component Using the Fan Component - Coefficient Method (HyperMesh CFD) *576*
ACU-T: 5101 Modeling of a Fan Component Using the Fan Component - PQ Method (HyperMesh CFD) *599*
ACU-T: 5200 / SL-2330 Rigid-Body Dynamics of a Check Valve (SimLab) *650*
ACU-T: 5200 Rigid-Body Dynamics of a Check Valve (HyperMesh CFD) *619*
ACU-T: 5201 / SL-2331 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve (SimLab) *683*
ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve (HyperMesh CFD) *651*
ACU-T: 5202 / SL-2320 Flow Closing Valve (SimLab) *684*
ACU-T: 5400 / SL-2420 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (P-FSI) (SimLab) *685*
ACU-T: 5403 Piezoelectric Flow Energy Harvester: A Fluid-Structure Interaction (HyperMesh CFD) *686*
ACU-T: 6000 Static Mixer Simulation – AcuTrace (HyperMesh CFD) *713*
ACU-T: 6010 Flow Through Porous Medium (HyperMesh CFD) *1016*
ACU-T: 6100 / SL 2430 Particle Separation in a Windshifter using Altair EDEM (SimLab) *775*
ACU-T: 6100 Particle Separation in a Windshifter using AcuSolve - EDEM Bidirectional Coupling (HyperMesh CFD) *818*
ACU-T: 6100 Particle Separation in a Windshifter using Altair EDEM (HyperMesh CFD) *743*
ACU-T: 6101 / SL 2431 Particle Separation in a Windshifter using AcuSolve - EDEM Unidirectional Coupling (SimLab) *817*
ACU-T: 6101 Particle Separation in a Windshifter using AcuSolve - EDEM Unidirectional Coupling (HyperMesh CFD) *776*
ACU-T: 6102 / SL 2432 Particle Separation in a Windshifter using AcuSolve - EDEM Bidirectional Coupling (SimLab) *862*
ACU-T: 6103 / SL 2433 AcuSolve – EDEM Bidirectional Coupling with Heat Transfer (SimLab) *900*

ACU-T: 6103 AcuSolve - EDEM Bidirectional Coupling with Heat Transfer (HyperMesh CFD) *863*
ACU-T: 6104 / SL 2440 AcuSolve – EDEM Bidirectional Coupling with Non-Spherical Particles (SimLab) *936*
ACU-T: 6104 AcuSolve - EDEM Bidirectional Coupling with Non-Spherical Particles (HyperMesh CFD) *901*
ACU-T: 6105 / SL 2441 - Single Particle Sedimentation – Effect of Lift and Torque (SimLab) *961*
ACU-T: 6105 Single Particle Sedimentation – Effect of Lift and Torque (HyperMesh CFD) *937*
ACU-T: 6106 / SL 2442 - AcuSolve - EDEM Bidirectional Coupling with Mass Transfer (SimLab) *989*
ACU-T: 6106 AcuSolve - EDEM Bidreictional Coupling with Mass Transfer (HyperMesh CFD) *962*
ACU-T: 6107 / SL-2450 AcuSolve - EDEM Bidirectional 3-Phase Coupling (SimLab) *990*
ACU-T: 6108 AcuSolve - EDEM Bidreictional Coupling with User-Defined Drag Model (HyperMesh CFD) *991*
ACU-T: 6500 / SL-2050 Flow Through Porous Medium (SimLab) *1031*
ACU-T: 6501 Flow Through Porous Medium with Physical Velocity (HyperMesh CFD) *1032*
ACU-T: 7200 / SL 2500 Topology Optimization (SimLab) *1046*
ACU-T: 7201 / SL 2501 – Topology Optimization with Flow Distribution (SimLab) *1047*
ACU-T: 7202 / SL 2502 – CFD Topology Optimization of Multiple Flow Paths (SimLab) *1048*
ACU-T: 7300 / SL-2510 DOE using Inspire, SimLab and HyperStudy *1049*
ACU-T: 7301 /SL-2515 Solution DOE *1050*
ACU-T:3401 / SL-2411 Electric Motor Analysis using AcuSolve-Flux One-Way Coupling (SimLab) *353*
ACU-T:4101 / SL 2231 T-junction Flow using the Eulerian Multiphase Model (SimLab) *469*
ACU-T:4102 / SL 2232 Fluidized Bed using the Eulerian Multiphase Model (SimLab) *486*
ACU-T:4300 / SL 2240 Species Transport Modeling (SimLab) *538*
ACU-T:4400 / SL 2250 Mixing Flows using the Gas Kinetics Model (SimLab) *540*
ACU-T:5002 / SL 2302 Centrifugal Blower – Hybrid (MRF + MM) (SimLab) *575*
ACU-T:5403 / SL 2421 Piezoelectric Flow Energy Harvester DC-FSI (SimLab) *710*
AcuSolve user-defined function overview (HyperMesh CFD) *993*

B

basic workflow *v*

I

Introduction to AcuSolve Tutorials *ii*

O

objectives *iii*
overview, AcuSolve user-defined function (HyperMesh CFD) *993*

S

supporting files *vii*

T

tutorial prerequisites *iv*
typographical conventions used in this manual *viii*