Tutorial 6. Using a Non-Conformal Mesh

Introduction

Film cooling is a process that is used to protect turbine vanes in a gas turbine engine from exposure to hot combustion gases. This tutorial illustrates how to set up and solve a film cooling problem using a non-conformal mesh. The system that is modeled consists of three parts: a duct, a hole array, and a plenum. The duct is modeled with a hexahedral mesh, and the plenum and hole regions are modeled using a tetrahedral mesh. These two meshes are merged together to form a "hybrid" mesh, with a non-conformal interface boundary between them.

Due to symmetry of the hole array, only a portion of the geometry is modeled in FLUENT, with symmetry applied to the outer boundaries. The duct contains a high-velocity fluid in streamwise flow (Figure 6.1). An array of holes intersects the duct at an inclined angle, and a cooler fluid is injected into the holes from a plenum. The coolant that moves through the holes acts to cool the surface of the duct, downstream of the injection. Both fluids are air, and the flow is classified as turbulent. The velocity and temperature of the streamwise and cross-flow fluids are known, and FLUENT is used to predict the flow and temperature fields that result from convective heat transfer.

In this tutorial you will learn how to:

- Merge hexahedral and tetrahedral meshes to form a hybrid mesh
- Create a non-conformal grid interface
- Model heat transfer across a non-conformal interface with specified temperature and velocity boundary conditions
- Calculate a solution using the segregated solver
- Plot temperature profiles on specified isosurfaces

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed Tutorial 1 . Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

This problem considers a model of a 3D section of a film cooling test rig. A schematic is shown in Figures 6.1 and 6.2. The problem consists of a duct, 24.5 in long, with cross-sectional dimensions of 0.75 in × 5 in. An array of uniformly spaced holes is located at the bottom of the duct. Each hole has a diameter of 0.5 inches, is inclined at 35 degrees, and is spaced 1.5 inches apart laterally. Cooler injected air enters the system through the plenum, with cross-sectional dimensions of 3.3 in × 1.25 in.

Because of the symmetry of the geometry, only a portion of the domain needs to be modeled. The computational domain is shown in outline in Figure 6.2. The bulk temperature of the streamwise air (T_∞) is 273 K, and the velocity of the air stream is 20 m/s. The bottom wall of the duct that intersects the hole array is assumed to be a completely insulated (adiabatic) wall. The secondary (injected) air enters the plenum at a uniform velocity of 0.4559 m/s. The temperature of the injected air ($T_{\rm inject}$) is 136.6 K. The properties of air that are used in the model are shown in Figure 6.2.

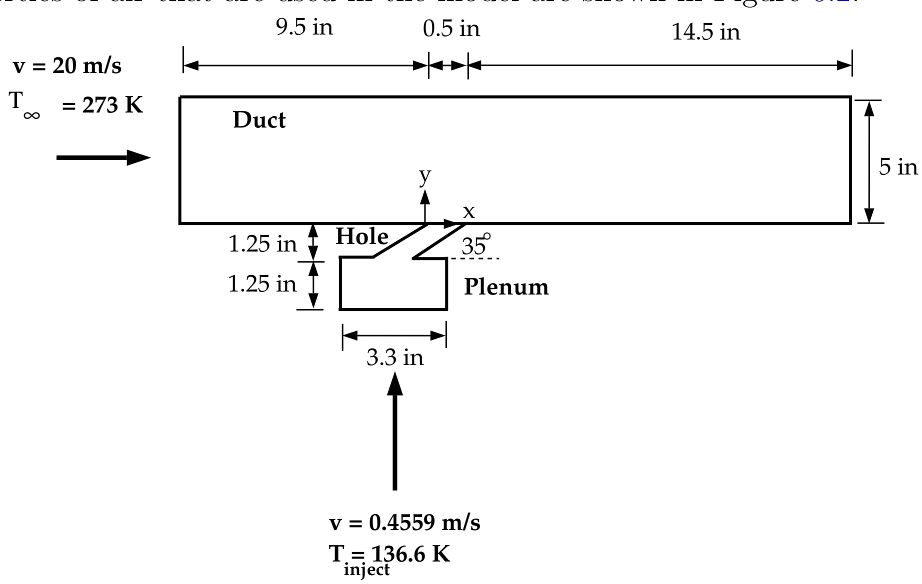


Figure 6.1: Schematic of the Problem, Front View

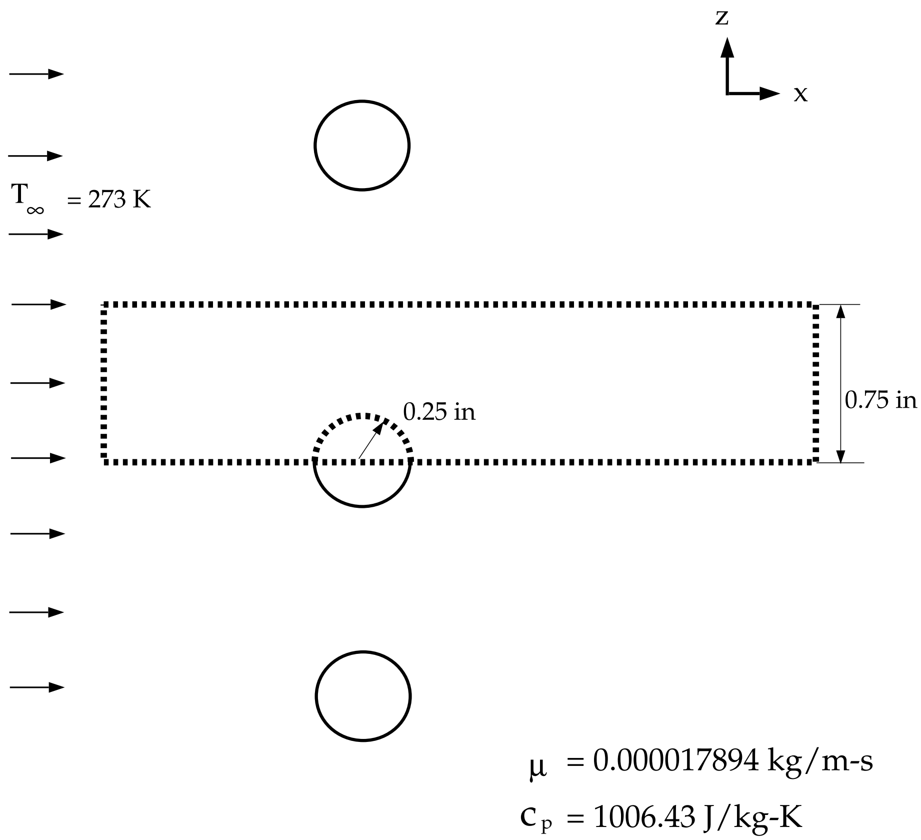


Figure 6.2: Schematic of the Problem, Top View

Setup and Solution

Preparation

1. Download `non_conformal_mesh.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working directory (as described in Tutorial 1).

2. Unzip `non_conformal_mesh.zip`.

   `film_hex.msh` *and* `film_tet.msh` *can be found in the* `/non_conformal_mesh` *folder created after unzipping the file.*

Step 1: Merging the Mesh Files

1. Start the 3D version of tmerge by typing `utility tmerge -3d` at the system prompt.

2. Provide the mesh file names `film_tet.msh` and `film_hex.msh` as prompted. Provide scaling of 1 and translations and rotations of zero for each mesh file. Save the new merged mesh file as `filmcool.msh`.

```
Append 3D grid files.
tmerge3D Fluent Inc, Version 2.1.11

 Enter name of grid file (ENTER to continue) : film_tet.msh

 x,y,z scaling factor, eg. 1 1 1         : 1 1 1

 x,y,z translation, eg. 0 1 0            : 0 0 0

 rot axis (0,1,2), angle (deg), eg. 1 45 : 0 0

 Enter name of grid file (ENTER to continue) : film_hex.msh

 x,y,z scaling factor, eg. 1 1 1         : 1 1 1

 x,y,z translation, eg. 0 1 0            : 0 0 0

 rot axis (0,1,2), angle (deg), eg. 1 45 : 0 0

 Enter name of grid file (ENTER to continue) : <ENTER>

 Enter name of output file               : filmcool.msh
```

⚠ The mesh files must be read into tmerge in this order for the tutorial to run as written. Otherwise, zone names and numbers will be assigned differently when the files are merged together. In general, however, you can specify files to be read into tmerge in any order.

Step 2: Grid

1. Start the 3D version of FLUENT.

2. Read in the mesh file `filmcool.msh`.

   File ⟶ Read ⟶ Case...

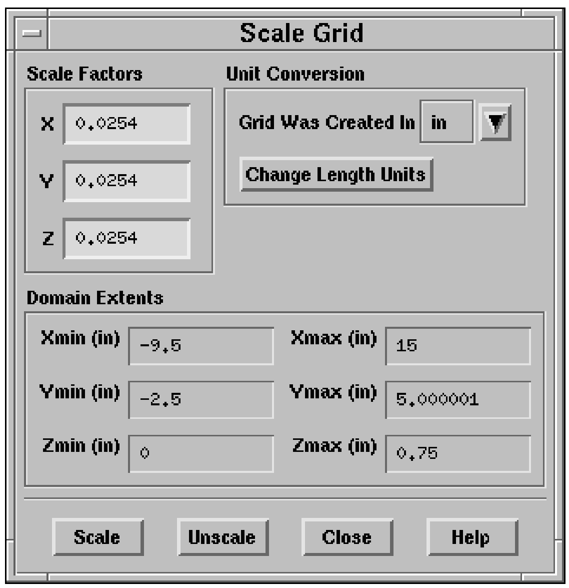
3. Check the grid.

   Grid ⟶ Check

   FLUENT *will perform various checks on the mesh and will report the progress in the console window. Pay particular attention to the reported minimum volume. Make sure this is a positive number.*

4. Scale the grid and change the unit of length to inches.

   Grid ⟶ Scale...



   (a) In the Unit Conversion drop-down list, select in to complete the phrase Grid Was Created In in (inches).

   (b) Click Scale to scale the grid.

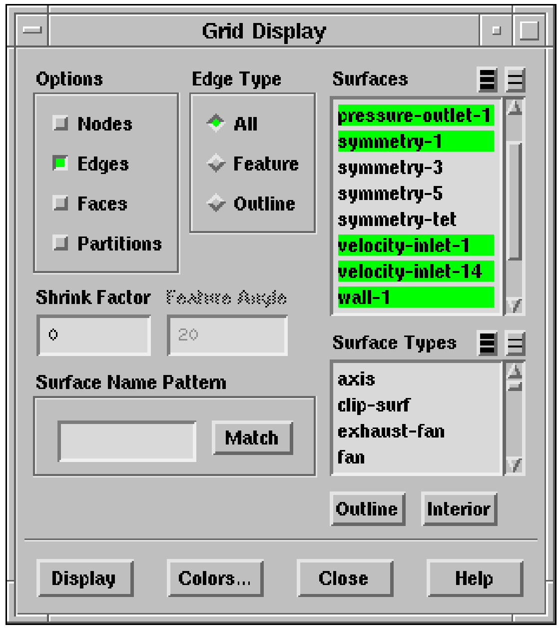
   (c) Click Change Length Units to set inches as the working units for length.

   The final Domain Extents *should appear as in the panel above.*

   (d) Close the panel.

5. Display an outline of the 3D grid (Figure 6.3).

   Display →Grid...



   (a) In the Surfaces list, deselect symmetry-3, symmetry-5 and symmetry-tet.

   (b) Click Display.

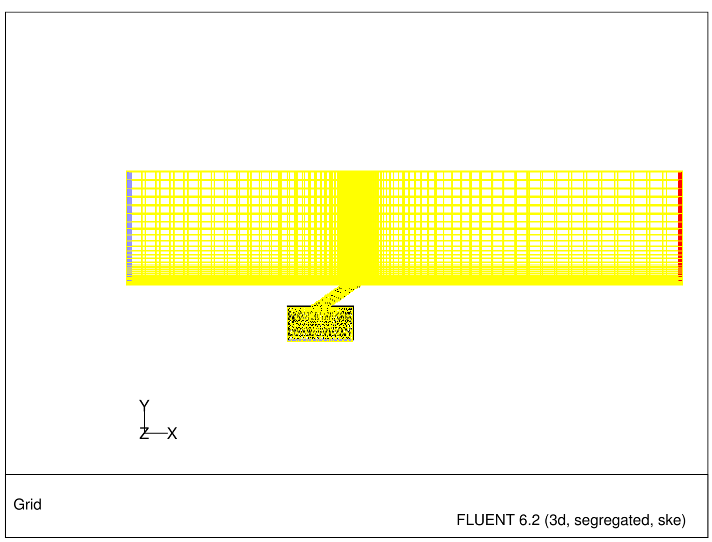


Figure 6.3: Hybrid Mesh for Film Cooling Problem

(c) Zoom in using your middle mouse button to get the view displayed in Figure 6.4.

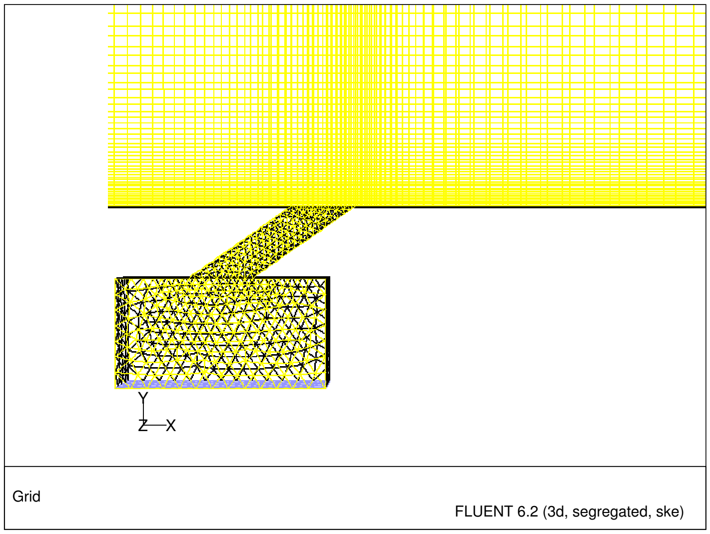


Figure 6.4: Hybrid Mesh (Zoomed-In View)

In Figure 6.4 you can see the quadrilateral faces of the hexahedral cells that are used to model the duct region, and the triangular faces of the tetrahedral cells that are used to model the plenum and hole regions, resulting in a hybrid mesh.

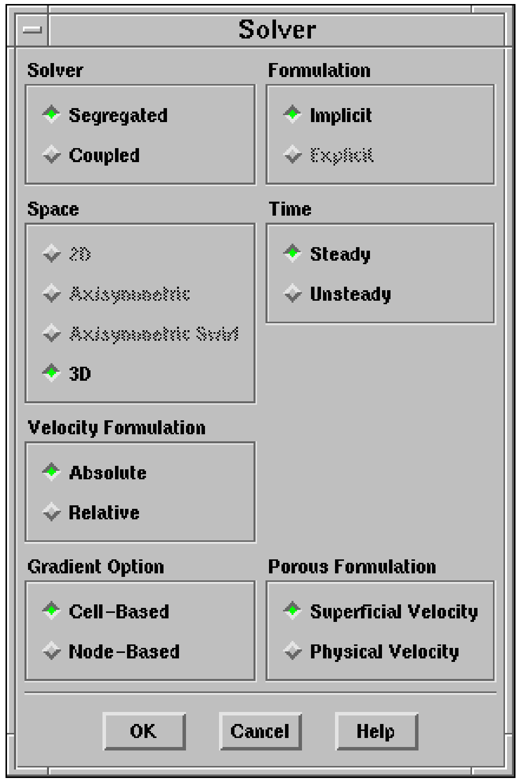
Extra: *You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the* FLUENT *console window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.*

6. Close the Grid Display panel.

Step 3: Models

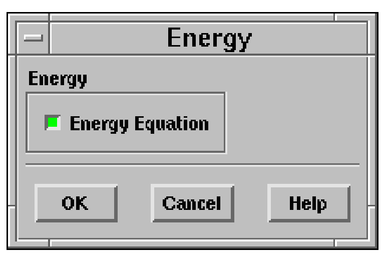
1. Keep the default solver settings.

   Define → Models → Solver...



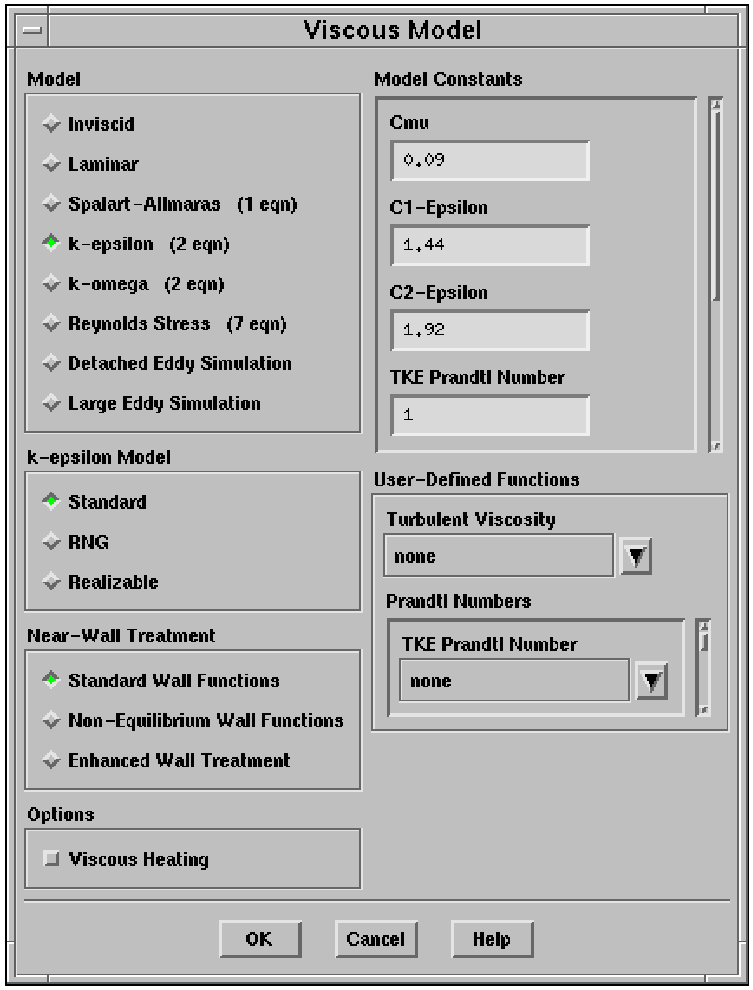
2. Enable heat transfer by activating the energy equation.

   Define → Models → Energy...



3. Enable the standard k-ϵ turbulence model.

   Define ⟶ Models ⟶ Viscous...



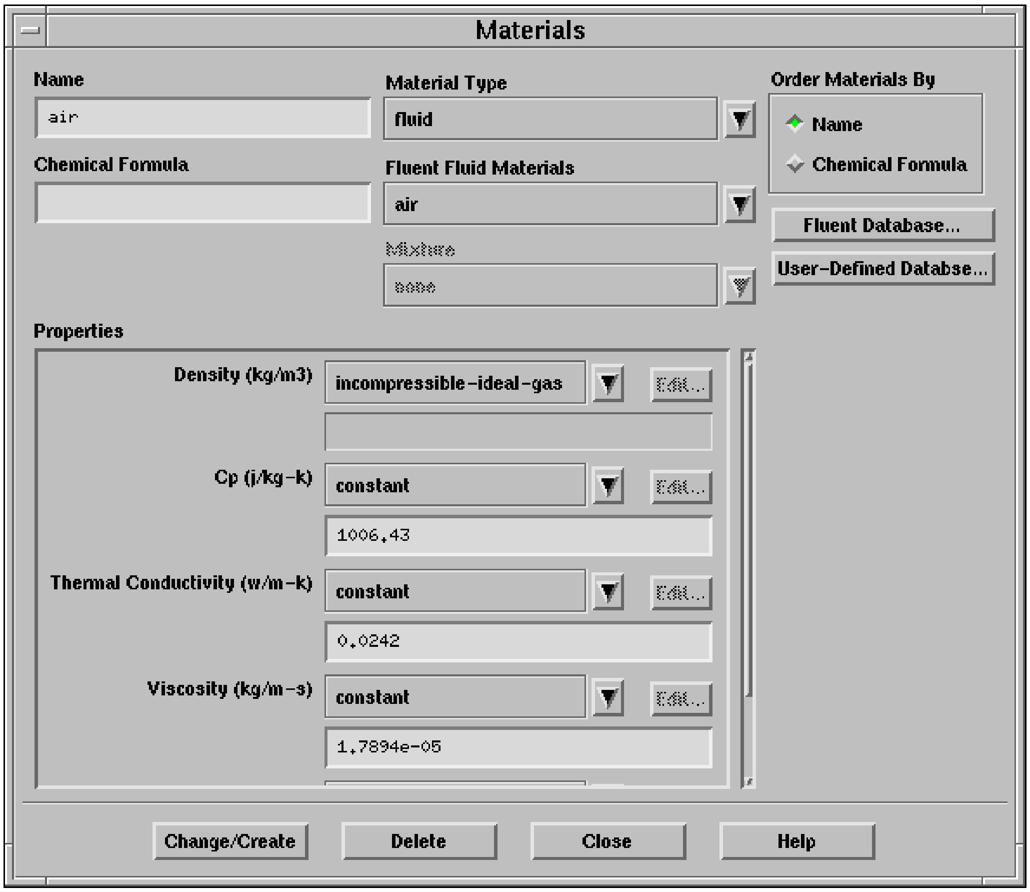
(a) Activate k-epsilon (2 eqn) under Model to expand the panel.

(b) Click OK.

Step 4: Materials

Define ⟶Materials...

1. Retain air as the Fluent Fluid Materials , and use the incompressible-ideal-gas law to compute density. Retain the default values for all other properties.

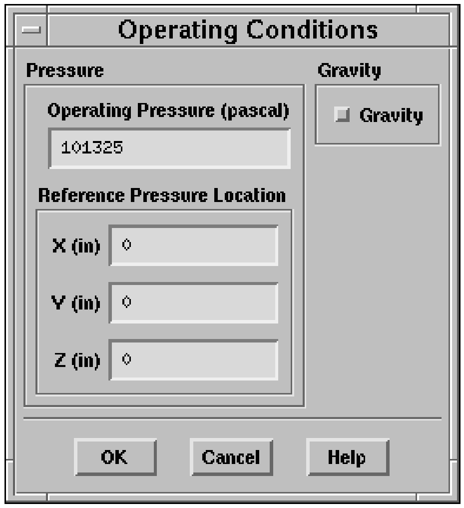


Do not forget to click the Change/Create button after selecting incompressible-ideal-gas in the drop-down list for Density.

Step 5: Operating Conditions

1. Keep the default operating conditions.

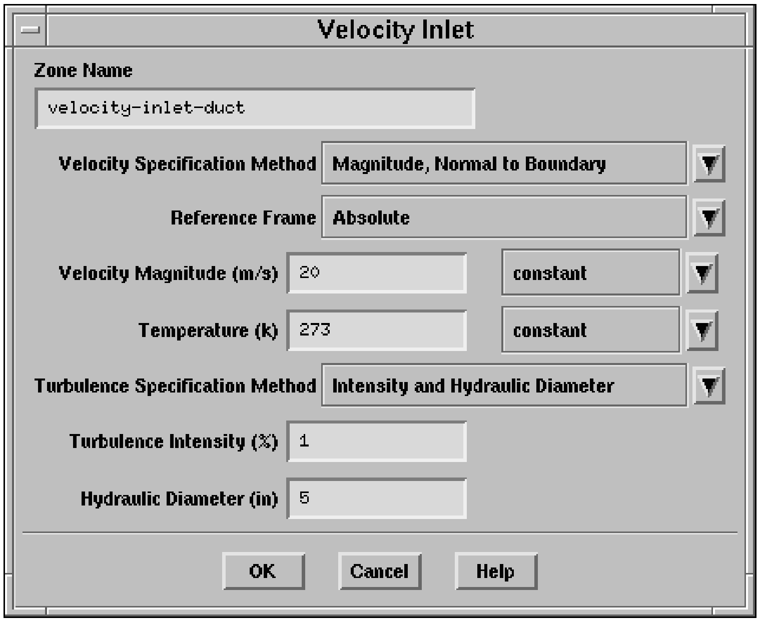
   Define →Operating Conditions...



Step 6: Boundary Conditions

Define ⟶Boundary Conditions...

1. Set the boundary conditions for the streamwise flow inlet (velocity-inlet-1).



   (a) Change the Zone Name from `velocity-inlet-1` to `velocity-inlet-duct`.

   (b) Set the Velocity Magnitude to 20 m/s.

   (c) Set the Temperature to 273 K.

   (d) In the Turbulence Specification Method drop-down list, select Intensity and Hydraulic Diameter.

   (e) Set the Turbulence Intensity to 1% and the Hydraulic Diameter to 5 in.

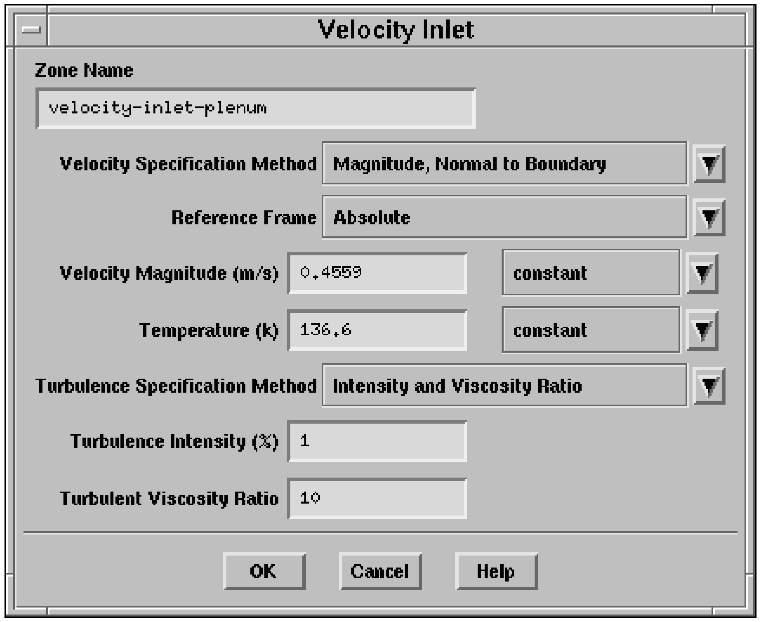
2. Set the boundary conditions for the injected stream inlet (velocity-inlet-14).

   (a) Change the Zone Name from `velocity-inlet-14` to `velocity-inlet-plenum`.

   (b) Set the Velocity Magnitude to `0.4559` m/s.

   (c) Set the Temperature to `136.6` K.

   (d) In the Turbulence Specification Method drop-down list, select Intensity and Viscosity Ratio.

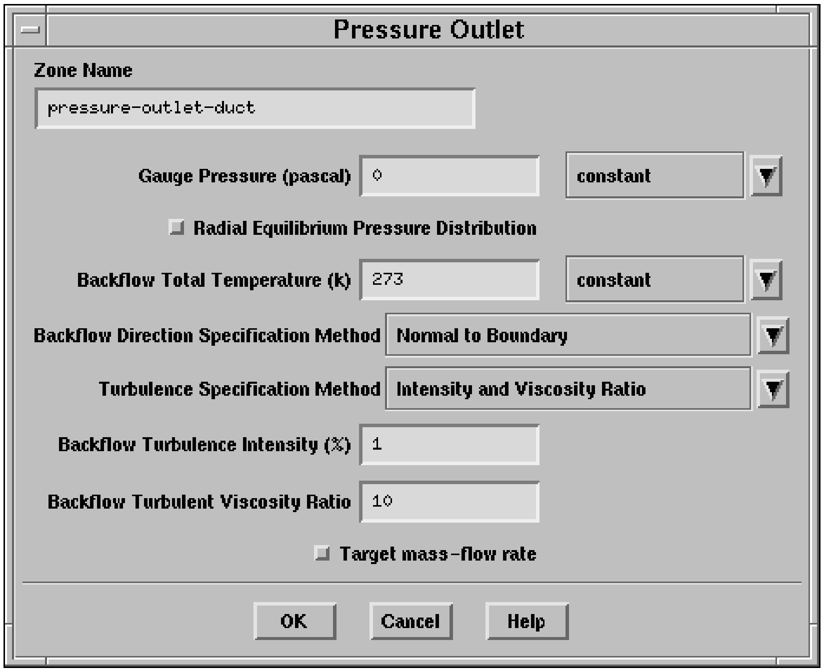


(e) Set the Turbulence Intensity to 1% and keep the Turbulent Viscosity Ratio default of 10.

In the absence of any identifiable length scale for turbulence, the Intensity and Viscosity Ratio *method should be used.*

See Chapter 11 of the User's Guide for more information on how to set the boundary conditions for turbulence.

3. Set the boundary conditions for the flow exit (pressure-outlet-1).



(a) Change the Zone Name from `pressure-outlet-1` to `pressure-outlet-duct`.

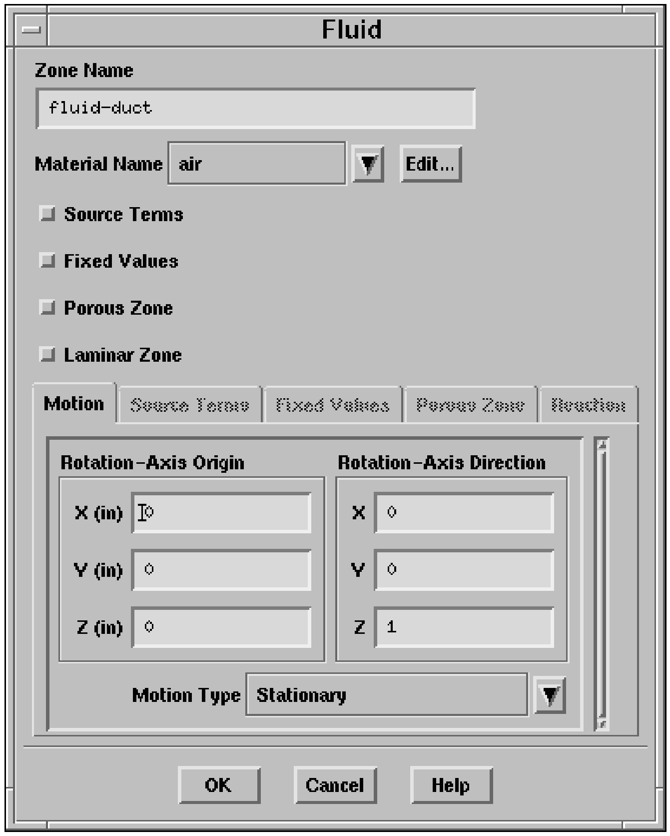
(b) Keep the default setting of 0 Pa for Gauge Pressure.

(c) Set the Backflow Total Temperature to 273 K.

(d) In the Turbulence Specification Method drop-down list, select Intensity and Viscosity Ratio.

(e) Set the Backflow Turbulence Intensity to 1% and keep the Backflow Turbulent Viscosity Ratio default of 10.

4. Set the conditions for the fluid in the duct (fluid-9).



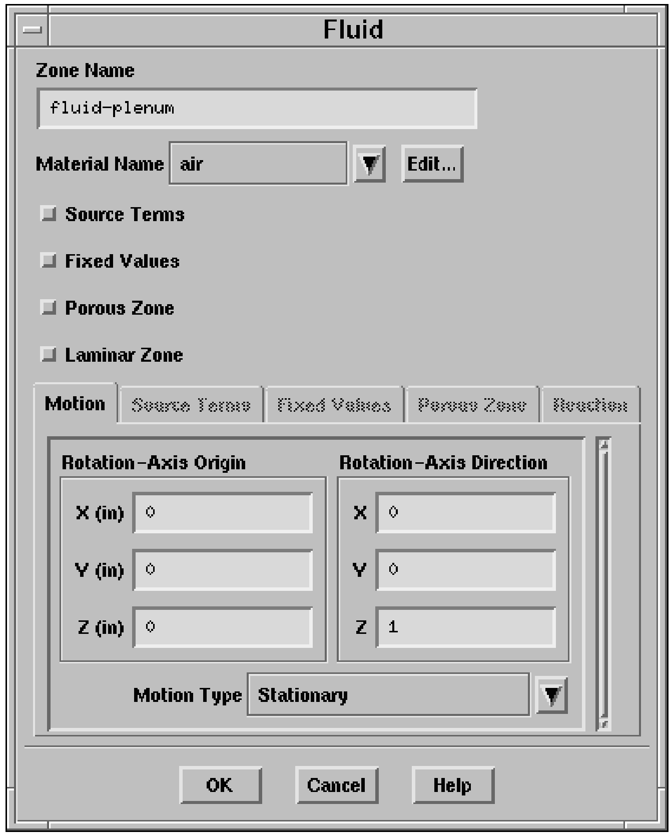
(a) Change the Zone Name from `fluid-9` to `fluid-duct`.

(b) Keep the default selection of air as the Material Name.

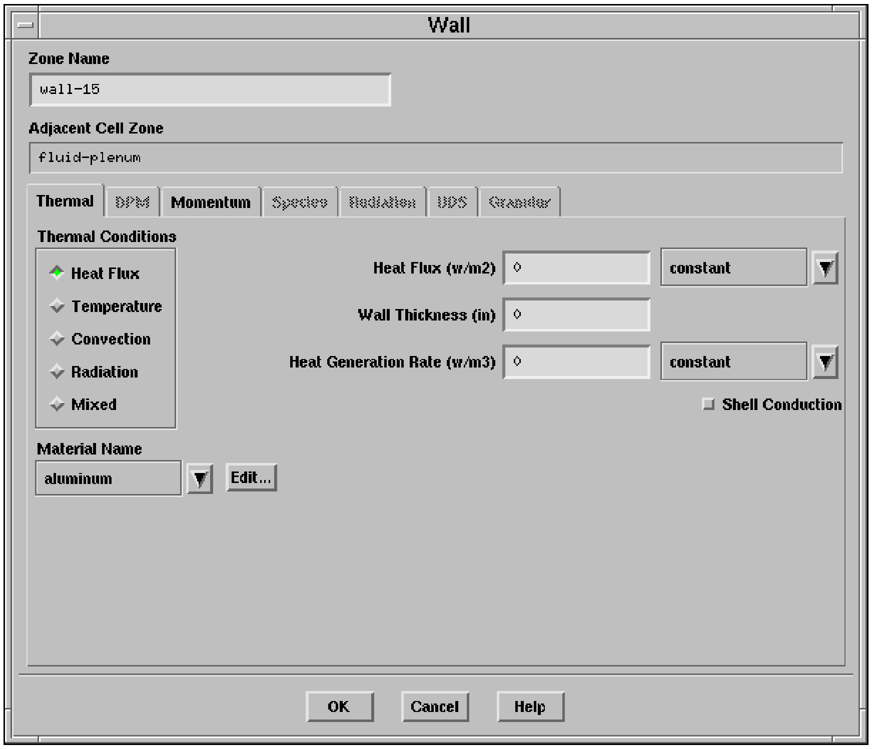
5. Set the conditions for the fluid in the plenum and hole (fluid-17).

   (a) Change the Zone Name from `fluid-17` to `fluid-plenum`.

   (b) Keep the default selection of air as the Material Name.



6. Keep the default boundary conditions for the plenum and hole wall (wall-15).



7. Define the zones on the non-conformal boundary as interface zones.

   The non-conformal grid interface contains two boundary zones: wall-1 *and* wall-12*.* wall-1 *is the bottom surface of the duct, and* wall-12 *represents the hole through which the cool air is injected from the plenum (Figure 6.5). These boundaries were defined as walls in the original mesh files,* ***film_hex.msh*** *and* ***film_tet.msh****, and must be redefined as* interface *boundary types.*

   (a) Open the Grid Display panel.

   (b) Select wall-1 and wall-12 under Surfaces.

   (c) Click Display.

      i. Activate bottom view.

         Display ⟶Views...

         A. Select bottom under Views and click Restore.

            Zoom in using middle mouse button to get the view as shown below.

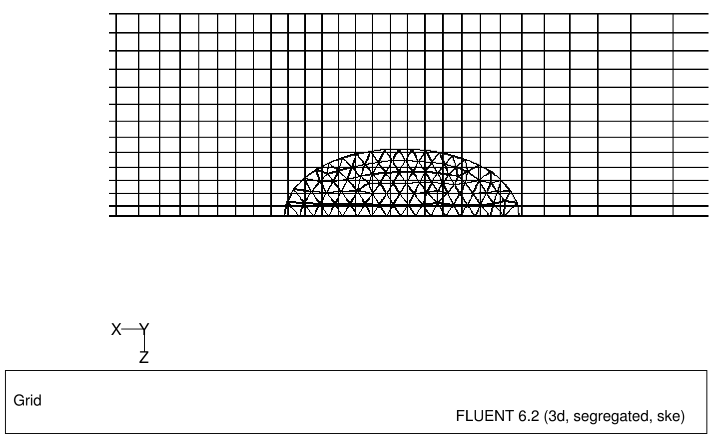
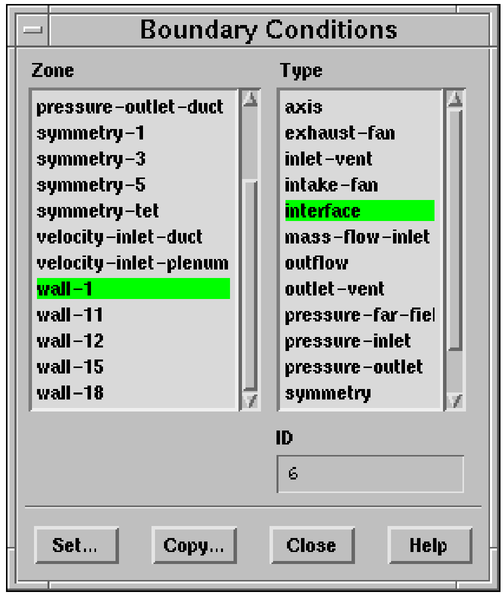


Figure 6.5: Grid for the wall-1 and wall-12 Boundaries

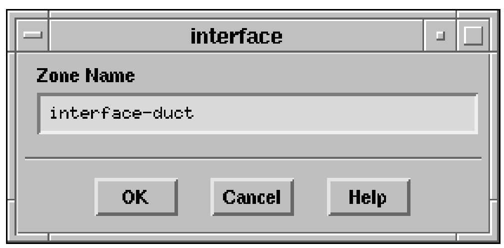
8. Change the Type for wall-1 and wall-12 to interface.

   (a) Select wall-1 in the Zone list and choose interface as the new Type.



   (b) Click Yes when asked OK to change wall-1's type from wall to interface?.

   (c) Change the Zone Name to `interface-duct`.



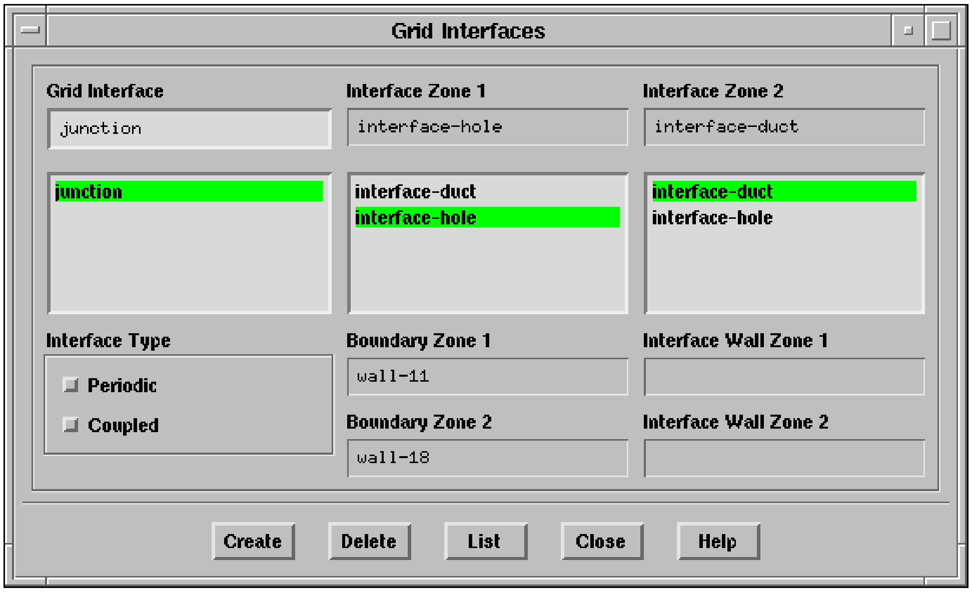
   (d) Repeat this procedure to convert wall-12 to an interface boundary zone named `interface-hole`.

9. Close the Boundary Conditions panel.

Step 7: Grid Interfaces

In this step, you will create a non-conformal grid interface between the hexahedral and tetrahedral meshes.

Define ⟶Grid Interfaces...



1. Select interface-hole in the Interface Zone 1 list.

   ⚠ When one interface zone is smaller than the other, it is recommended that you choose the smaller zone as Interface Zone 1.

2. Select interface-duct in the Interface Zone 2 list.

3. Enter the name `junction` under Grid Interface.

4. Click Create.

   Note: *In the process of creating the grid interface,* FLUENT *creates two new wall boundary zones:* wall-11 *and* wall-18. *You will* **not** *be able to display these walls.*

   wall-11 *is the non-overlapping region of the* interface-hole *zone that results from the intersection of the* interface-hole *and* interface-duct *boundary zones, and is listed under* Boundary Zone 1 *in the* Grid Interfaces *panel.* wall-11 *is empty, since* interface-hole *is completely contained within the* interface-duct *boundary.*

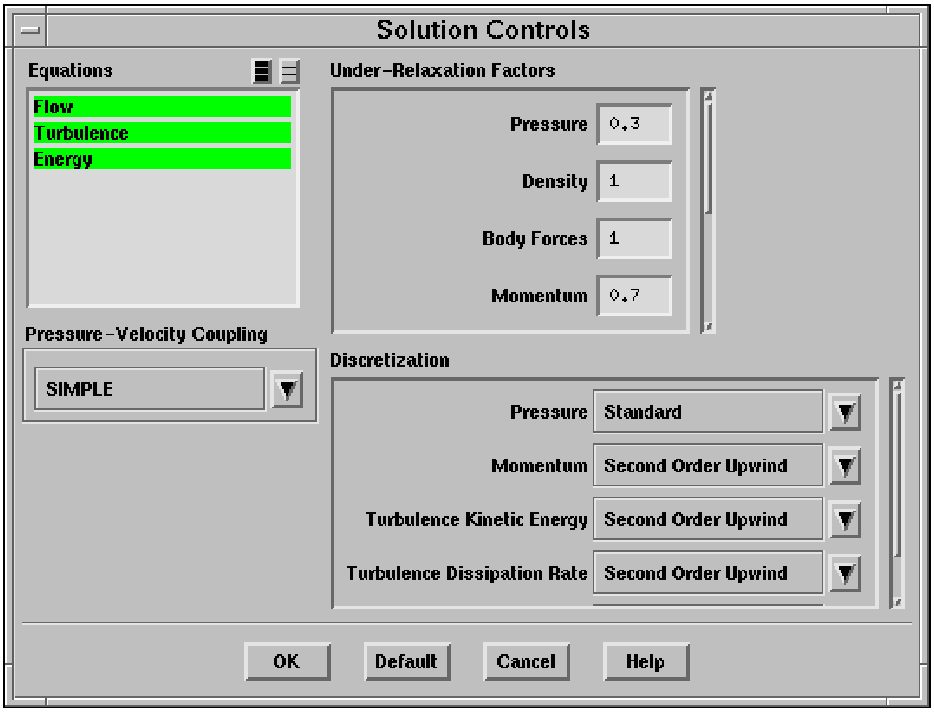
   wall-18 *is the non-overlapping region of the* interface-duct *zone that results from the intersection of the two interface zones, and is listed under* Boundary Zone 2 *in the* Grid Interfaces *panel.*

⚠ In general, you will need to set boundary conditions for wall-18 (since it is not empty). In this case, default settings are used.

Step 8: Solution

1. Set the solution parameters.

   Solve ⟶ Controls ⟶ Solution...



   (a) Under Discretization, select Second Order Upwind for Momentum and Turbulence Kinetic Energy.

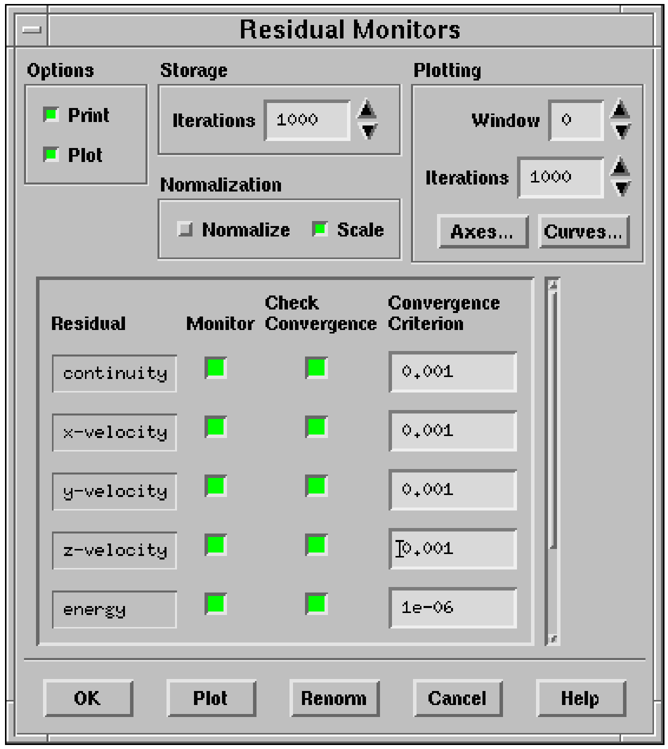
   (b) Scroll down the list and select Second Order Upwind for Turbulence Dissipation Rate and Energy.

2. Enable the plotting of residuals.

   Solve ⟶ Monitors ⟶ Residual...

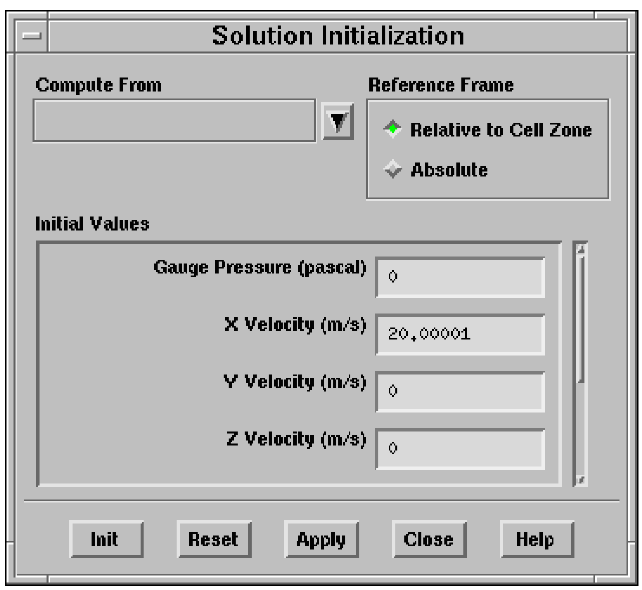
   (a) Under Options, select Plot.

   (b) Click the OK button.



3. Initialize the solution.

Solve ⟶ Initialize ⟶ Initialize...



(a) Select velocity-inlet-duct in the Compute From drop-down list.

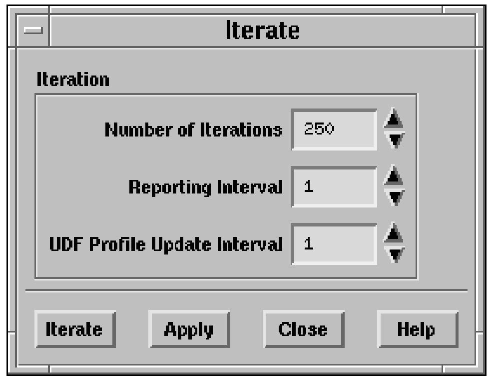
(b) Click Init, and Close the panel.

4. Save the case file (`filmcool.cas`).

   File → Write → Case...

5. Start the calculation by requesting 250 iterations.

   Solve → Iterate...



   (a) Set the Number of Iterations to 250.

   (b) Click Iterate.

   Note: *During the first few iterations, the console window will report that turbulent viscosity is limited in a couple of cells. This message should go away as the solution converges and the turbulent viscosity approaches more reasonable levels.*

   The solution will converge after about 135 iterations.

6. Save the case and data files (`filmcool.cas` and `filmcool.dat`).

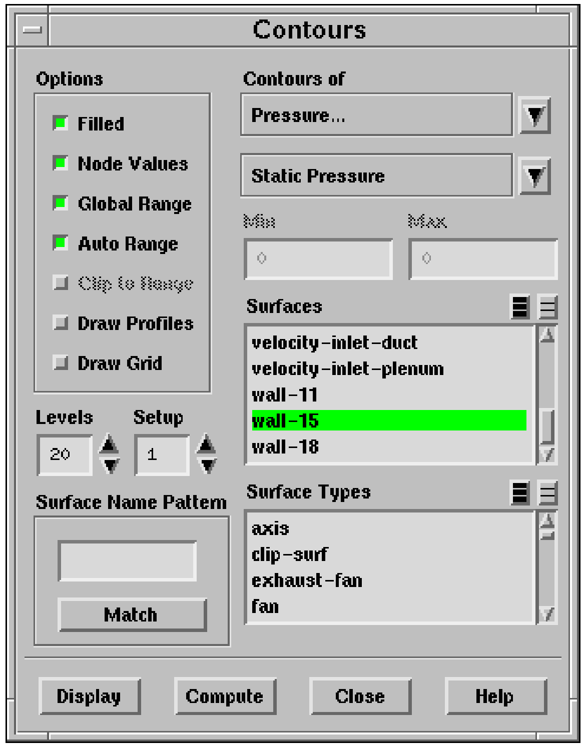
   File → Write → Case & Data...

   Note: *If you choose a file name that already exists in the current directory,* FLUENT *will prompt you for confirmation to overwrite the file.*

Step 9: Postprocessing

1. Display filled contours of static pressure (Figure 6.6).

   Display →Contours...



   (a) Select Filled under Options.

   (b) Select Pressure... and Static Pressure in the Contours of drop-down lists.

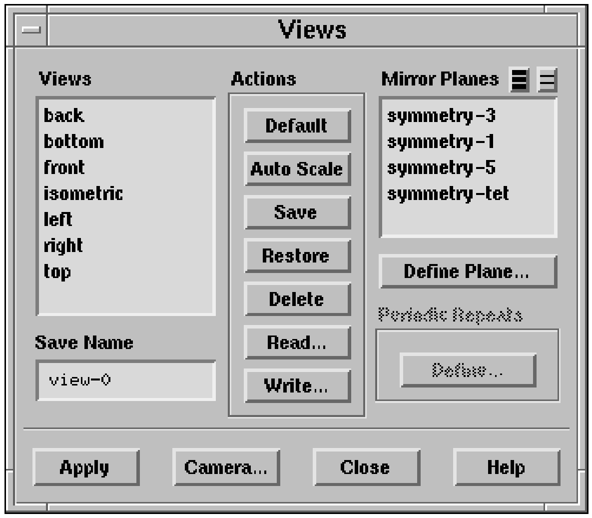
   (c) In the Surfaces list, select interface-duct and interface-hole.

   (d) Scroll down the Surfaces list and select symmetry-1, symmetry-tet, and wall-15.

(e) Reset the view to the default view.

Display ⟶Views...

i. Click Default under Actions.



ii. Close the panel.

(f) In the Contours panel, click Display.

(g) Zoom in on the view to get the display shown in Figure 6.6.

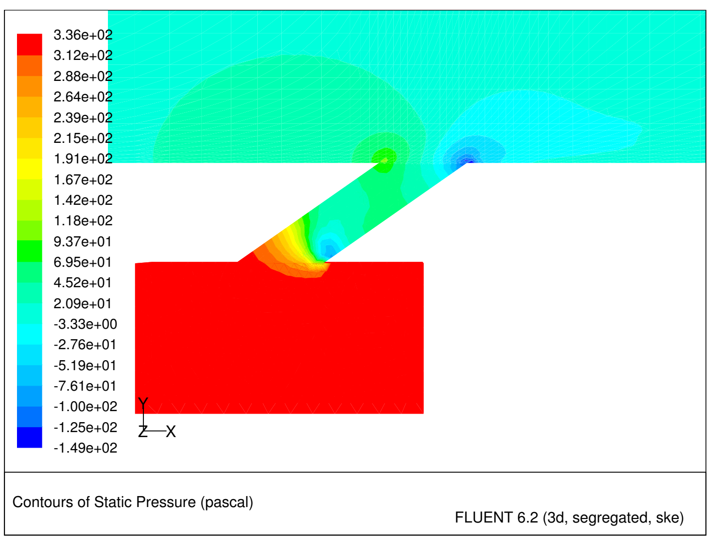
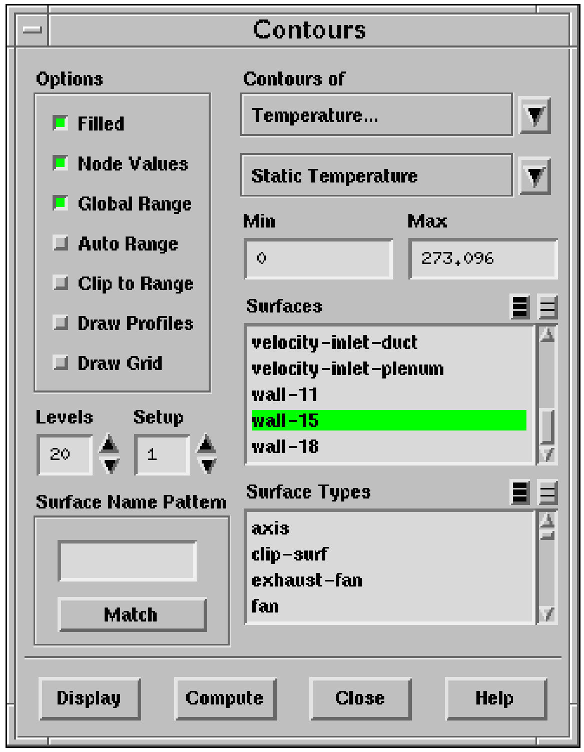


Figure 6.6: Contours of Static Pressure

Note the high/low pressure zones on the upstream/downstream sides of the coolant hole, where the jet first penetrates the primary flow in the duct.

2. Display filled contours of static temperature (Figures 6.7 and 6.8).

   Display ⟶Contours...



   (a) Select Temperature... and Static Temperature in the Contours of drop-down lists.

   (b) Under Options, deselect Auto Range so that you can change the maximum and minimum temperature gradient values to be plotted.

   (c) Keep the default Min value of `0`.

   (d) Enter a new Max value of `273.096`.

   (e) Under Options, deselect Clip to Range.

   (f) Click Display.

   (g) Zoom in on the view to get the display shown in Figure 6.8.

   Figures 6.7 and 6.8 clearly show how the coolant flow insulates the bottom of the duct from the higher-temperature primary flow.

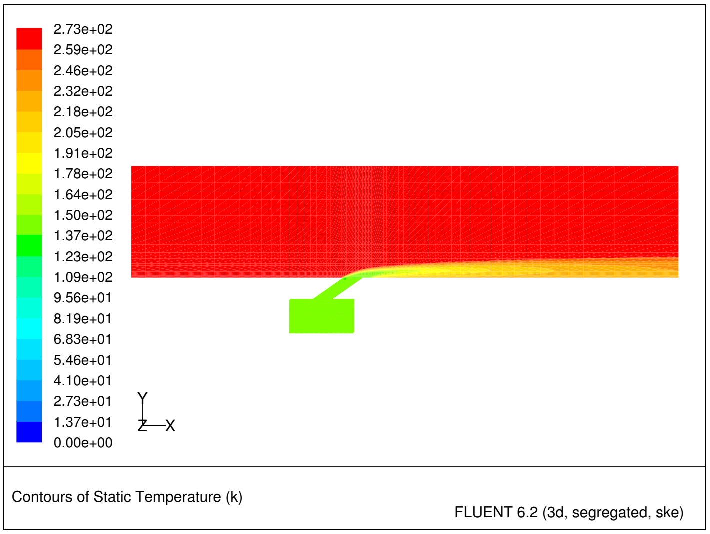


Figure 6.7: Contours of Static Temperature

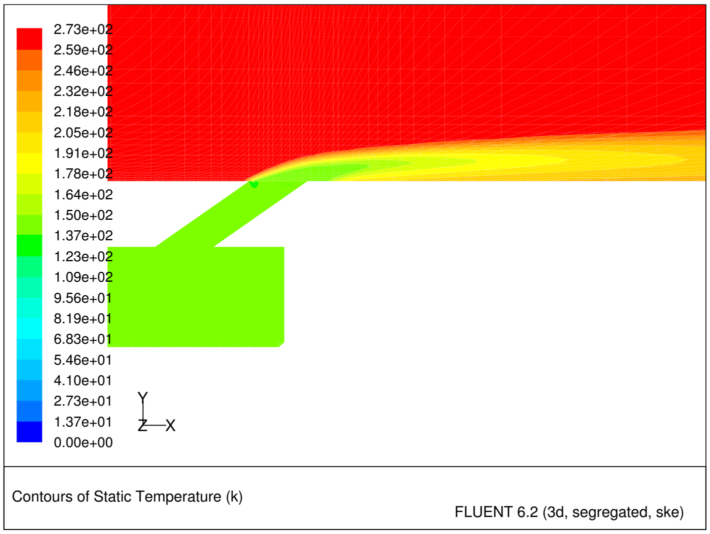
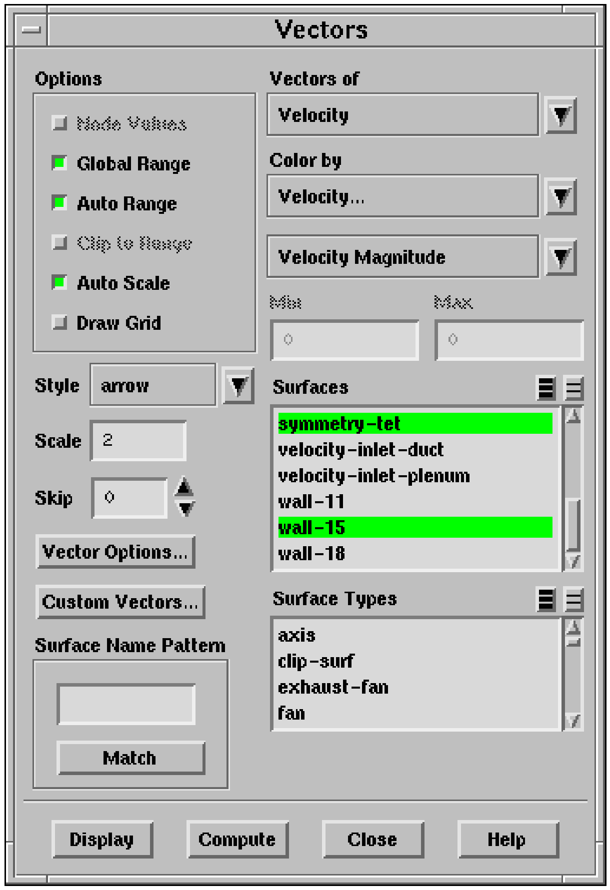


Figure 6.8: Contours of Static Temperature (Zoomed-In View)

3. Display the velocity vectors (Figure 6.9).

   Display ⟶Vectors...



   (a) Select Velocity... and Velocity Magnitude in the Color by drop-down lists.

   (b) Change the Scale to 2.

   This will enlarge the vectors that are displayed, making it easier to view the flow patterns.

   (c) In the Surfaces list, select interface-duct and interface-hole.

   (d) Scroll down the Surfaces list and select symmetry-1, symmetry-tet, and wall-15.

   (e) In the Vectors panel, click Display.

   (f) Zoom in on the view to get the display shown in Figure 6.9.

The flow pattern in the vicinity of the coolant hole shows the level of penetration of the coolant jet into the main flow. Notice that the velocity field varies smoothly across the non-conformal interface.

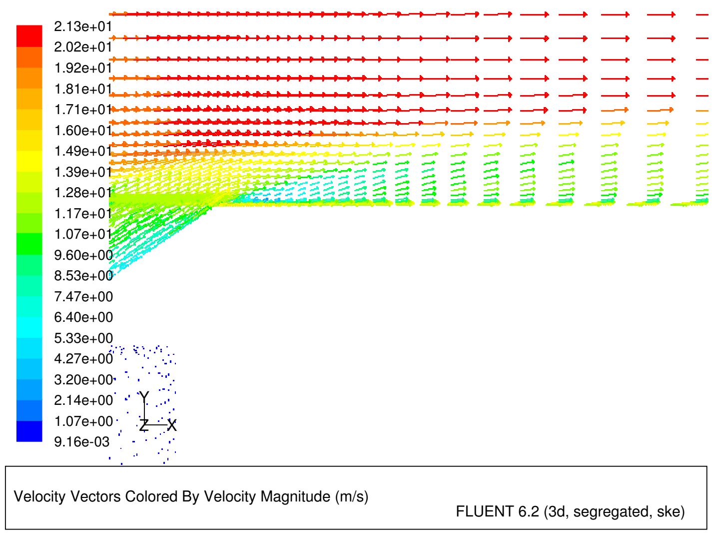
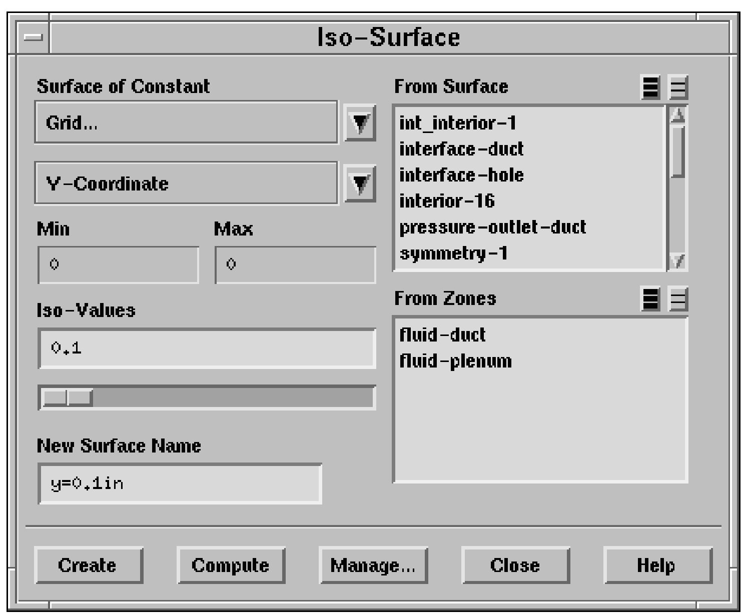


Figure 6.9: Velocity Vectors

4. Plot the temperature profile along a horizontal cross-section of the duct, 0.1 inches above the bottom.

   (a) Create an isosurface on the duct surface at $y = 0.1$ in.

   Surface ⟶Iso-Surface...



   i. Select Grid... and Y-Coordinate in the Surface of Constant drop-down lists.

   ii. Enter `y=0.1in` under New Surface Name.

iii. Enter `0.1` for Iso-Values.

iv. Click Create.

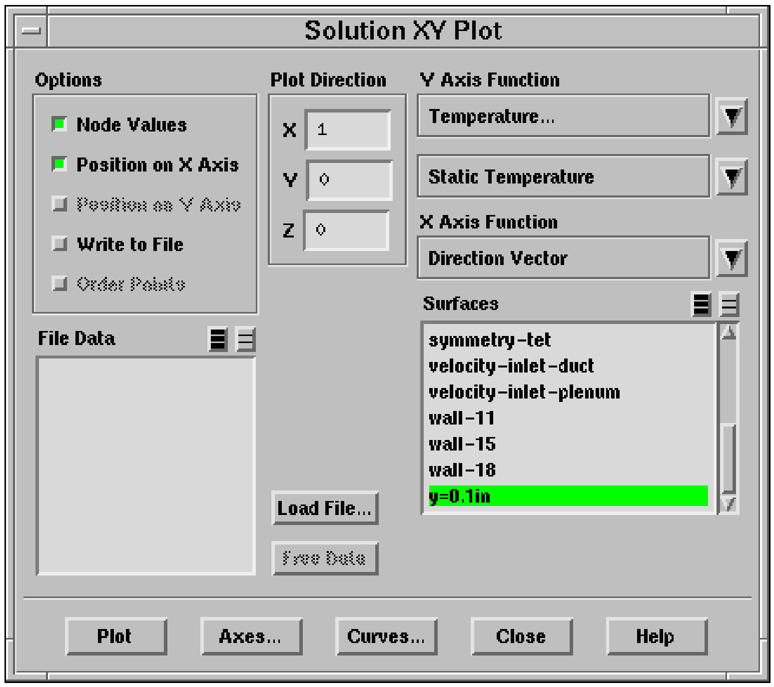
(b) Create an XY plot of static temperature on the isosurface.

Plot ⟶XY Plot...

i. Keep the default Plot Direction.

ii. Select Temperature... and Static Temperature in the Y-Axis Function drop-down lists.

iii. Scroll down the Surfaces list and select y=0.1in.



iv. Click Plot.

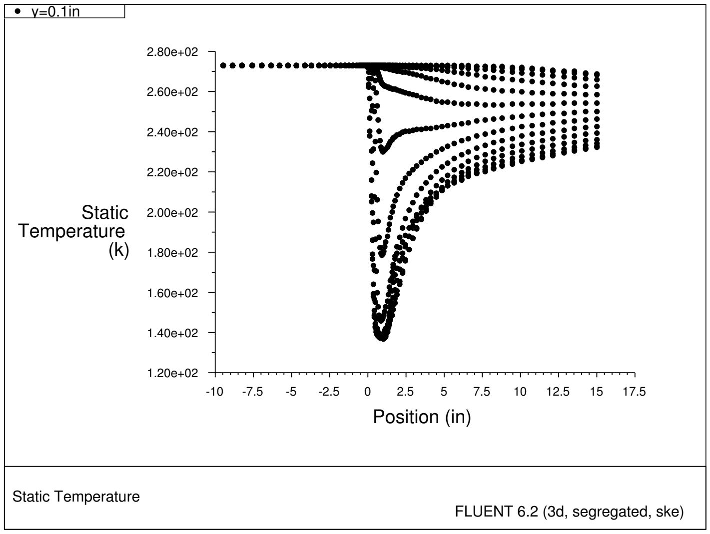


Figure 6.10: Static Temperature at y=0.1 in

In this plot you can see how the temperature of the fluid changes as the cool air from the injection hole mixes with the primary flow. As expected, the temperature is coolest just downstream of the hole. Note that you could also make a similar plot on the lower wall itself, to examine the wall surface temperature.

Summary

This tutorial demonstrates how FLUENT's non-conformal grid interface capability can be used to handle hybrid meshes for complex geometries, such as the film cooling hole configuration examined here. One of the principal advantages of this approach is that it allows you to merge existing component meshes together to create a larger, more complex mesh system, without requiring that the different components have the same node locations on their shared boundaries. Thus, you can perform parametric studies by merging the desired meshes, creating the non-conformal interface(s), and solving the model. For example, in the present case, you can

- Use a different hole/plenum mesh
- Reposition the existing hole/plenum mesh
- Add additional hole/plenum meshes to create aligned or staggered multiple hole arrays