Tutorial 7. Modeling Flow Through Porous Media

Introduction

Many industrial applications involve the modeling of flow through porous media, such as filters, catalyst beds, and packing. This tutorial illustrates how to set up and solve a problem involving gas flow through porous media.

The industrial problem solved here involves gas flow through a catalytic converter. Catalytic converters are commonly used to purify emissions from gasoline and diesel engines by converting environmentally hazardous exhaust emissions to acceptable substances. Examples of such emissions include carbon monoxide (CO), nitrogen oxides (NO_x), and unburned hydrocarbon fuels. These exhaust gas emissions are forced through a substrate, which is a ceramic structure coated with a metal catalyst such as platinum or palladium.

The nature of the exhaust gas flow is a very important factor in determining the performance of the catalytic converter. Of particular importance is the pressure gradient and velocity distribution through the substrate. Hence CFD analysis is used to design efficient catalytic converters: by modeling the exhaust gas flow, the pressure drop and the uniformity of flow through the substrate can be determined. In this tutorial, FLUENT is used to model the flow of nitrogen gas through a catalytic converter geometry, so that the flow field structure may be analyzed.

This tutorial demonstrates how to do the following:

- Set up a porous zone for the substrate with appropriate resistances.
- Calculate a solution for gas flow through the catalytic converter using the pressure-based solver.
- Plot pressure and velocity distribution on specified planes of the geometry.
- Determine the pressure drop through the substrate and the degree of non-uniformity of flow through cross sections of the geometry using X-Y plots and numerical reports.

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

The catalytic converter modeled here is shown in Figure 7.1. The nitrogen flows in through the inlet with a uniform velocity of 22.6 m/s, passes through a ceramic monolith substrate with square shaped channels, and then exits through the outlet.

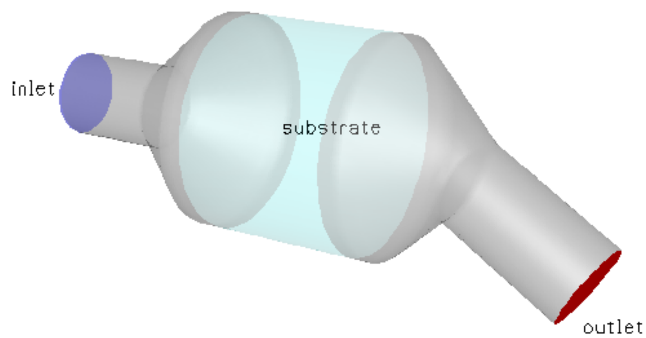


Figure 7.1: Catalytic Converter Geometry for Flow Modeling

While the flow in the inlet and outlet sections is turbulent, the flow through the substrate is laminar and is characterized by inertial and viscous loss coefficients in the flow (X) direction. The substrate is impermeable in other directions, which is modeled using loss coefficients whose values are three orders of magnitude higher than in the X direction.

Setup and Solution

Preparation

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

2. Unzip `porous.zip`.

   `catalytic_converter.msh` *can be found in the* `porous` *folder created after unzipping the file.*

3. Start the 3D (3d) version of FLUENT.

Step 1: Grid

1. Read the mesh file (`catalytic_converter.msh`).

   File → Read → Case...

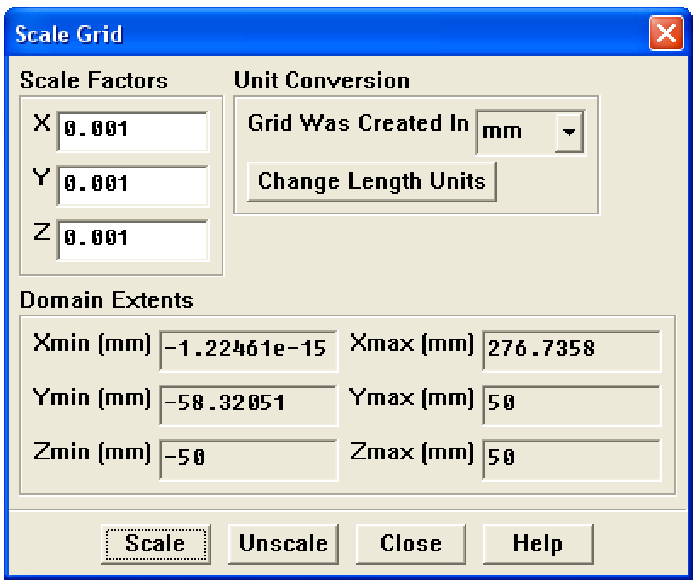
2. Check the grid.

   Grid → Check

   FLUENT *will perform various checks on the mesh and report the progress in the console. Make sure that the minimum volume reported is a positive number.*

3. Scale the grid.

   Grid → Scale...



   (a) Select mm from the Grid Was Created In drop-down list.

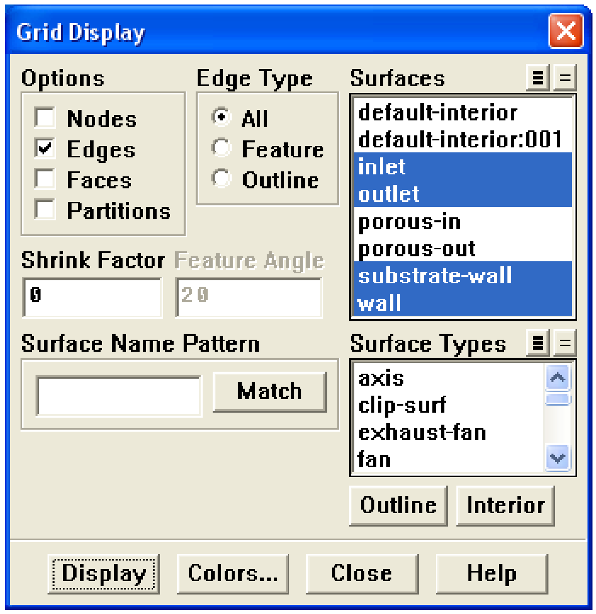
   (b) Click the Change Length Units button.

   All dimensions will now be shown in millimeters.

   (c) Click Scale and close the Scale Grid panel.

4. Display the mesh.

   Display ⟶Grid...



   (a) Make sure that inlet, outlet, substrate-wall, and wall are selected in the Surfaces selection list.

   (b) Click Display.

   (c) Rotate the view and zoom in to get the display shown in Figure 7.2.

   (d) Close the Grid Display panel.

   The hex mesh on the geometry contains a total of 34,580 cells.

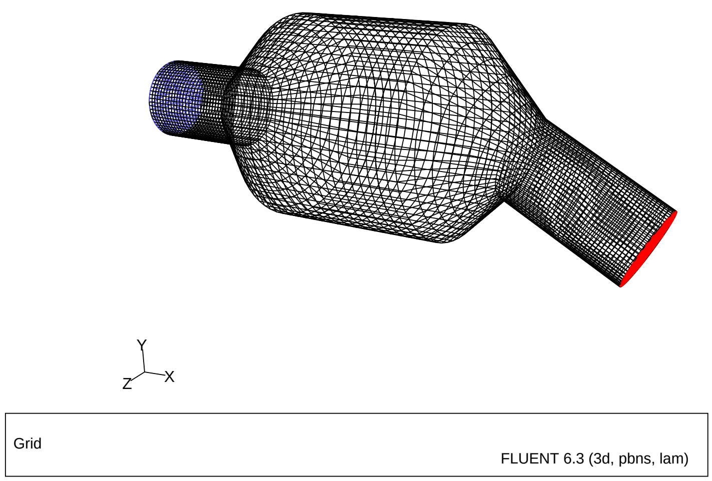
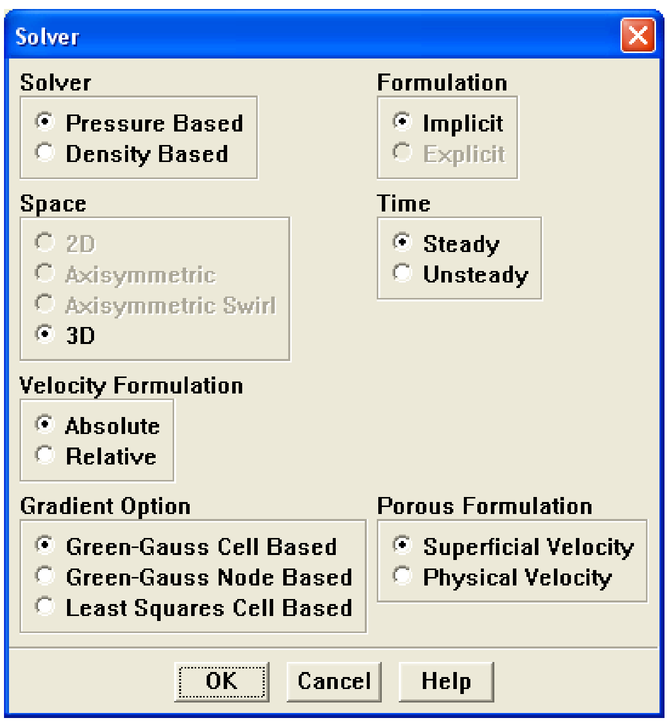


Figure 7.2: Mesh for the Catalytic Converter Geometry

Step 2: Models

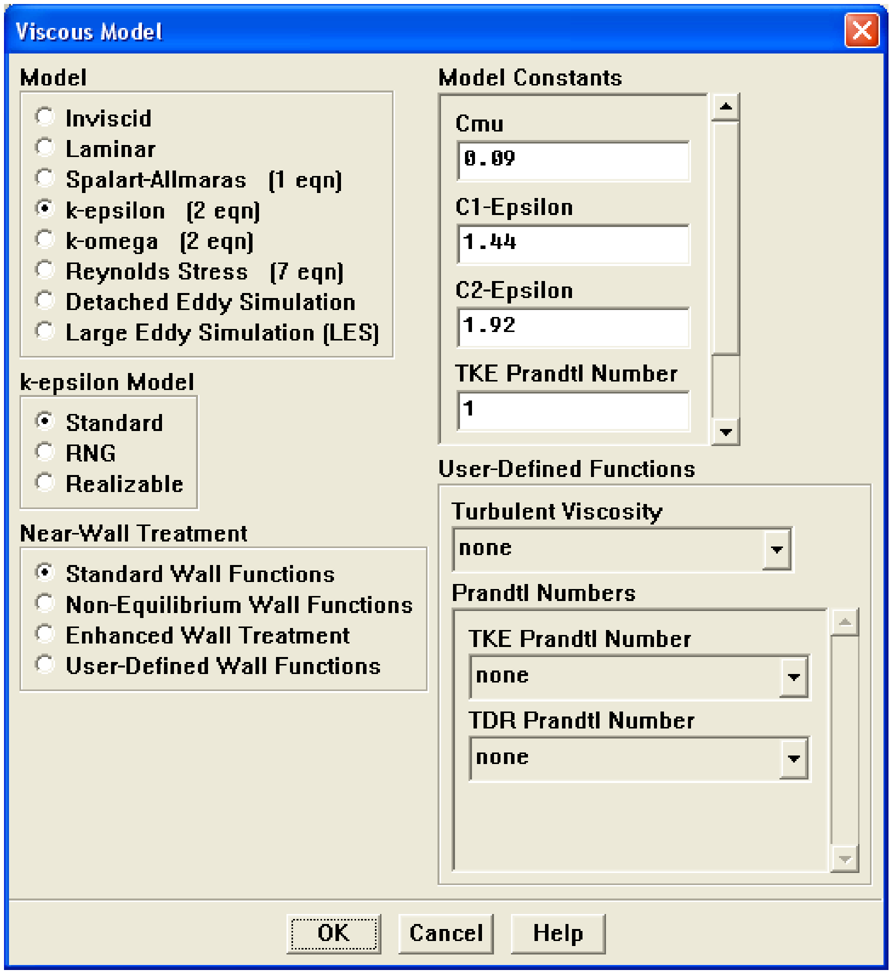
1. Retain the default solver settings.

   Define ⟶ Models ⟶ Solver...



2. Select the standard k-ϵ turbulence model.

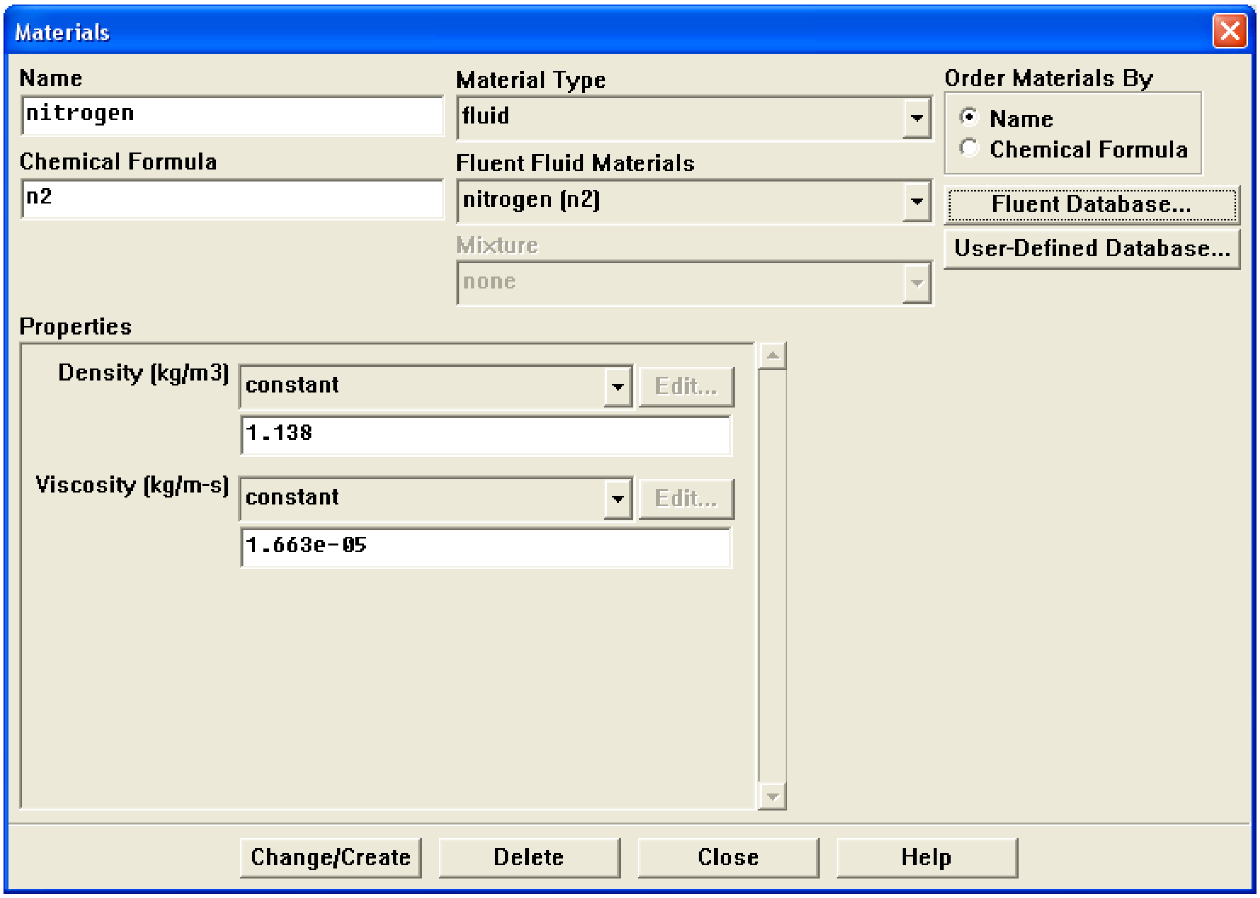
Define ⟶ Models ⟶ Viscous...



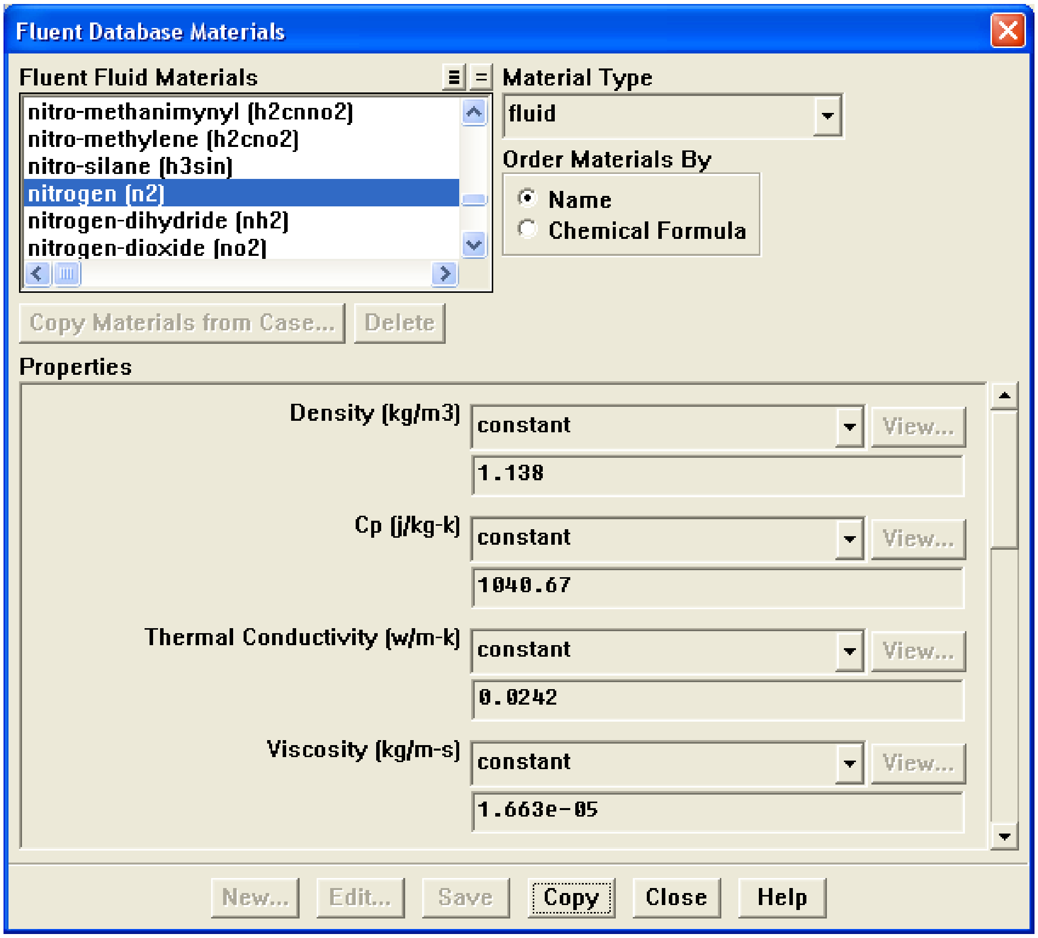
Step 3: Materials

1. Add nitrogen to the list of fluid materials by copying it from the Fluent Database for materials.

   Define →Materials...



(a) Click the Fluent Database... button to open the Fluent Database Materials panel.



i. Select nitrogen (n2) from the list of Fluent Fluid Materials.

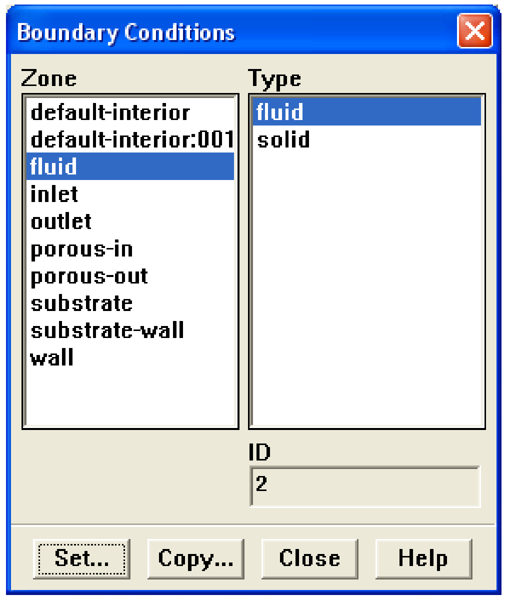
ii. Click Copy to copy the information for nitrogen to your list of fluid materials.

iii. Close the Fluent Database Materials panel.

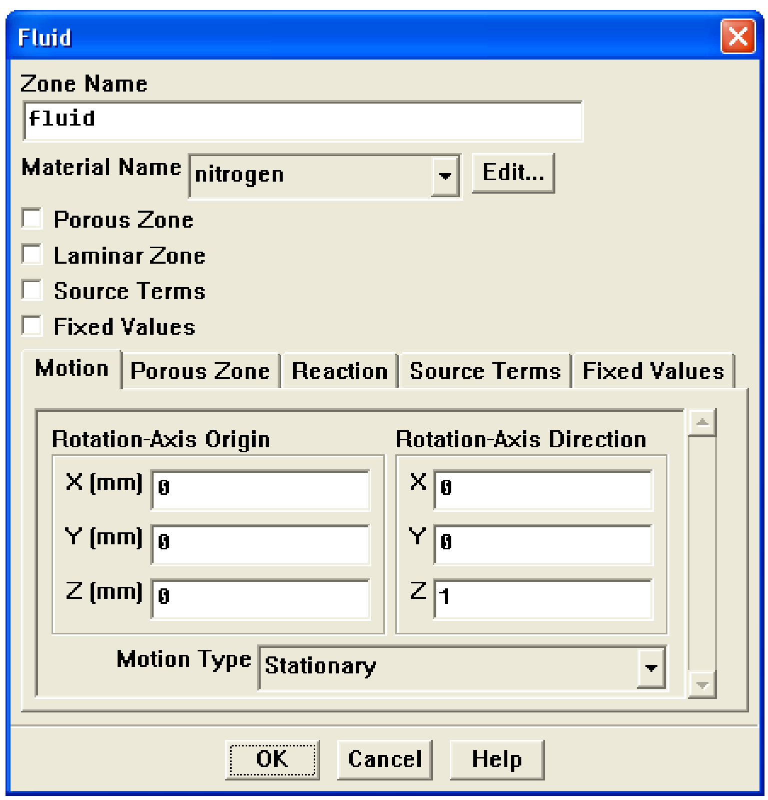
(b) Close the Materials panel.

Step 4: Boundary Conditions

Define ⟶Boundary Conditions...



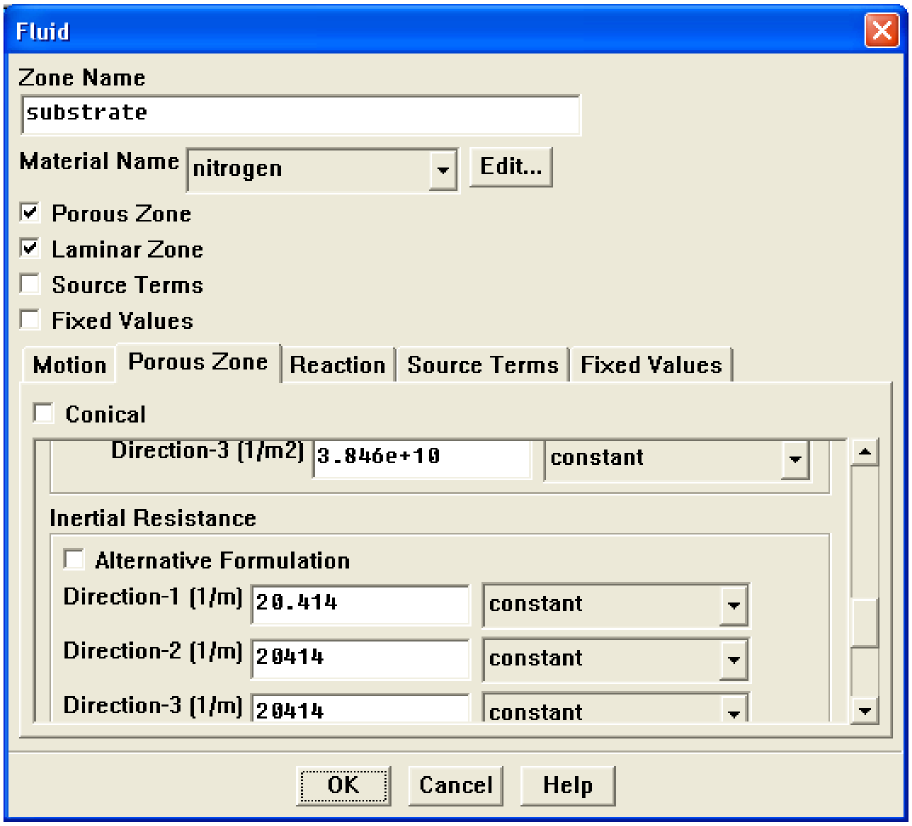
1. Set the boundary conditions for the fluid (fluid).



(a) Select nitrogen from the Material Name drop-down list.

(b) Click OK to close the Fluid panel.

2. Set the boundary conditions for the substrate (substrate).



(a) Select nitrogen from the Material Name drop-down list.

(b) Enable the Porous Zone option to activate the porous zone model.

(c) Enable the Laminar Zone option to solve the flow in the porous zone without turbulence.

(d) Click the Porous Zone tab.

i. Make sure that the principal direction vectors are set as shown in Table 7.1.

Use the scroll bar to access the fields that are not initially visible in the panel.

Axis	Direction-1 Vector	Direction-2 Vector
X	1	0
Y	0	1
Z	0	0

Table 7.1: Values for the Principle Direction Vectors

ii. Enter the values in Table 7.2 for the Viscous Resistance and Inertial Resistance.

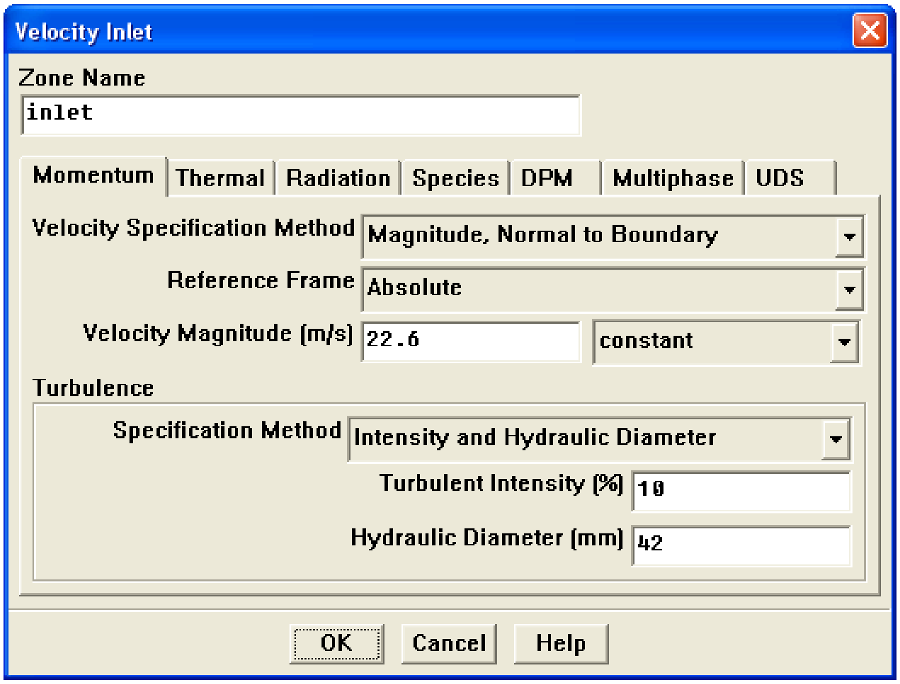
Scroll down to access the fields that are not initially visible in the panel.

Direction	Viscous Resistance (1/m2)	Inertial Resistance (1/m)
Direction-1	3.846e+07	20.414
Direction-2	3.846e+10	20414
Direction-3	3.846e+10	20414

Table 7.2: Values for the Viscous and Inertial Resistance

(e) Click OK to close the Fluid panel.

3. Set the velocity and turbulence boundary conditions at the inlet (inlet).



(a) Enter 22.6 m/s for the Velocity Magnitude.

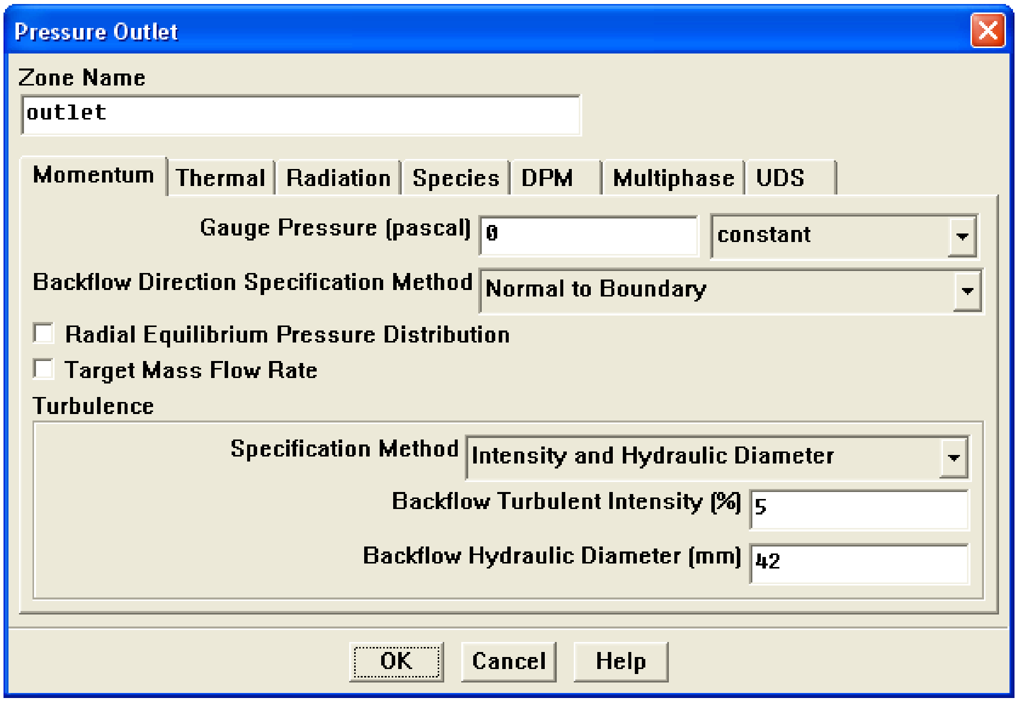
(b) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.

(c) Retain the default value of 10% for the Turbulent Intensity.

(d) Enter 42 mm for the Hydraulic Diameter.

(e) Click OK to close the Velocity Inlet panel.

4. Set the boundary conditions at the outlet (outlet).



(a) Retain the default setting of 0 for Gauge Pressure.

(b) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.

(c) Enter 5% for the Backflow Turbulent Intensity.

(d) Enter 42 mm for the Backflow Hydraulic Diameter.

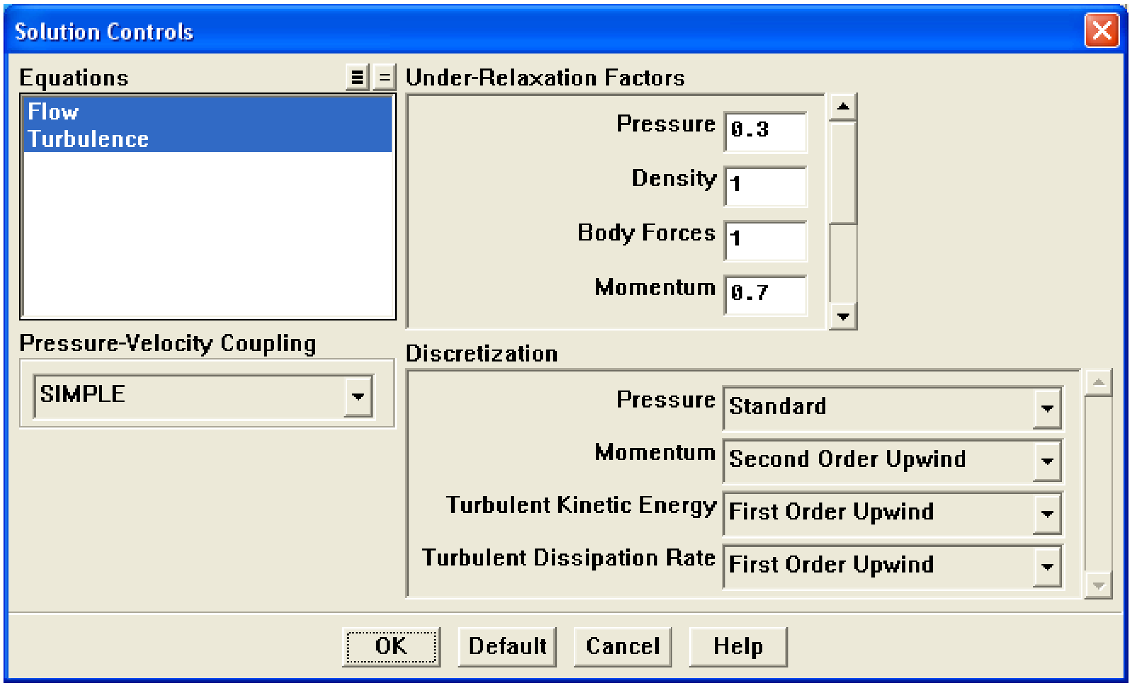
(e) Click OK to close the Pressure Outlet panel.

5. Retain the default boundary conditions for the walls (substrate-wall and wall) and close the Boundary Conditions panel.

Step 5: Solution

1. Set the solution parameters.

   Solve ⟶ Controls ⟶ Solution...



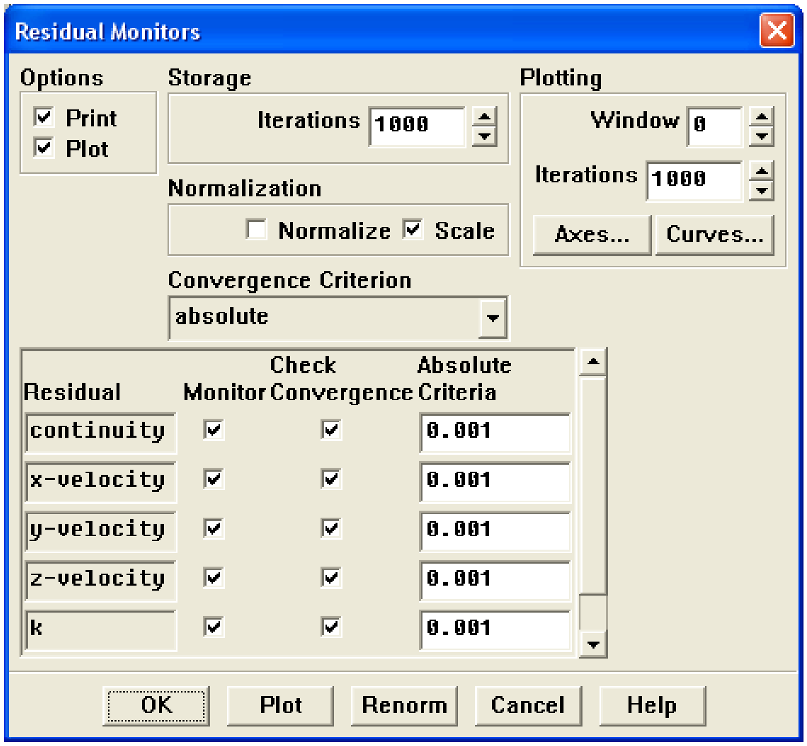
   (a) Retain the default settings for Under-Relaxation Factors.

   (b) Select Second Order Upwind from the Momentum drop-down list in the Discretization group box.

   (c) Click OK to close the Solution Controls panel.

2. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...

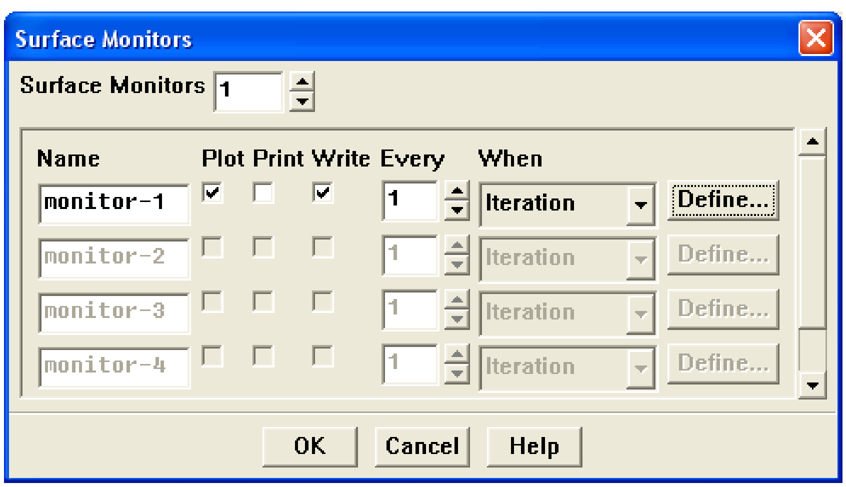


   (a) Enable Plot in the Options group box.

   (b) Click OK to close the Residual Monitors panel.

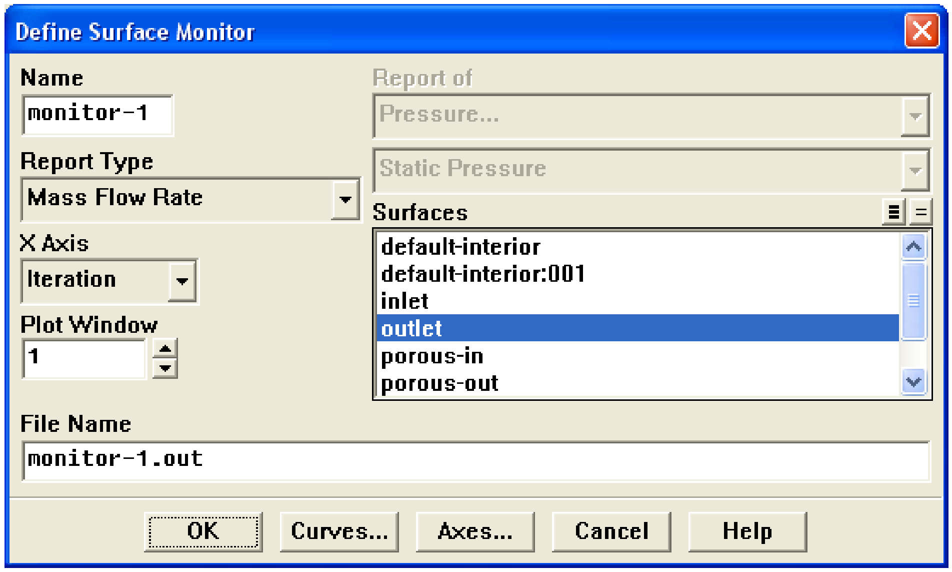
3. Enable the plotting of the mass flow rate at the outlet.

   Solve ⟶ Monitors ⟶ Surface...



   (a) Set the Surface Monitors to 1.

(b) Enable the Plot and Write options for monitor-1, and click the Define... button to open the Define Surface Monitor panel.



i. Select Mass Flow Rate from the Report Type drop-down list.

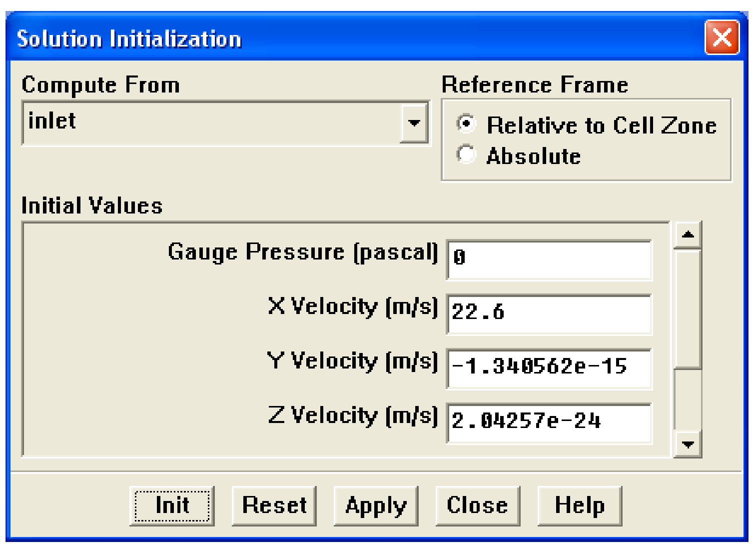
ii. Select outlet from the Surfaces selection list.

iii. Click OK to close the Define Surface Monitors panel.

(c) Click OK to close the Surface Monitors panel.

4. Initialize the solution from the inlet.

Solve ⟶ Initialize ⟶ Initialize...



(a) Select inlet from the Compute From drop-down list.

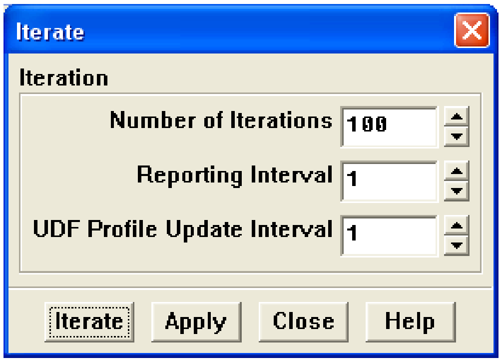
(b) Click Init and close the Solution Initialization panel.

5. Save the case file (`catalytic_converter.cas`).

   File ⟶ Write ⟶ Case...

6. Run the calculation by requesting 100 iterations.

   Solve ⟶ Iterate...



(a) Enter 100 for the Number of Iterations.

(b) Click Iterate.

The FLUENT *calculation will converge in approximately 70 iterations. By this point the mass flow rate monitor has flattened out, as seen in Figure 7.3.*

(c) Close the Iterate panel.

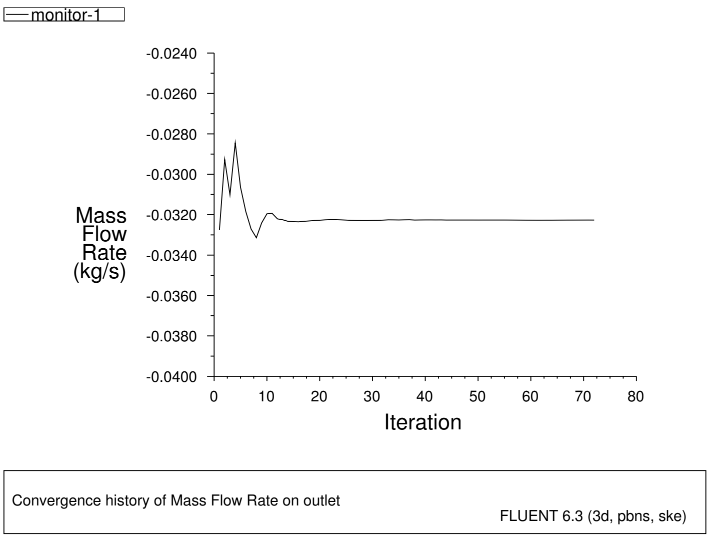


Figure 7.3: Surface Monitor Plot of Mass Flow Rate with Number of Iterations

7. Save the case and data files (`catalytic_converter.cas` and `catalytic_converter.dat`).

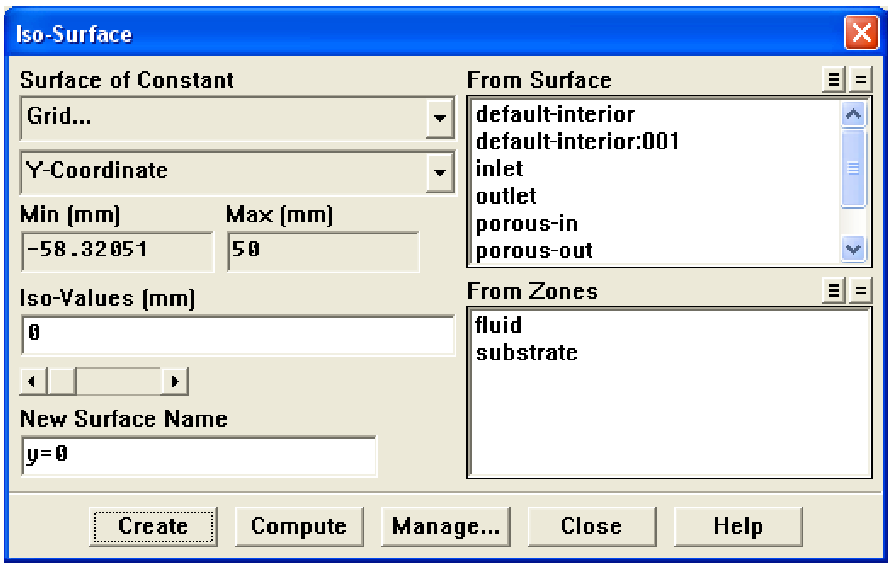
   File ⟶ Write ⟶ Case & Data...

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

Step 6: Postprocessing

1. Create a surface passing through the centerline for postprocessing purposes.

   Surface ⟶ Iso-Surface...



   (a) Select Grid... and Y-Coordinate from the Surface of Constant drop-down lists.

   (b) Click Compute to calculate the Min and Max values.

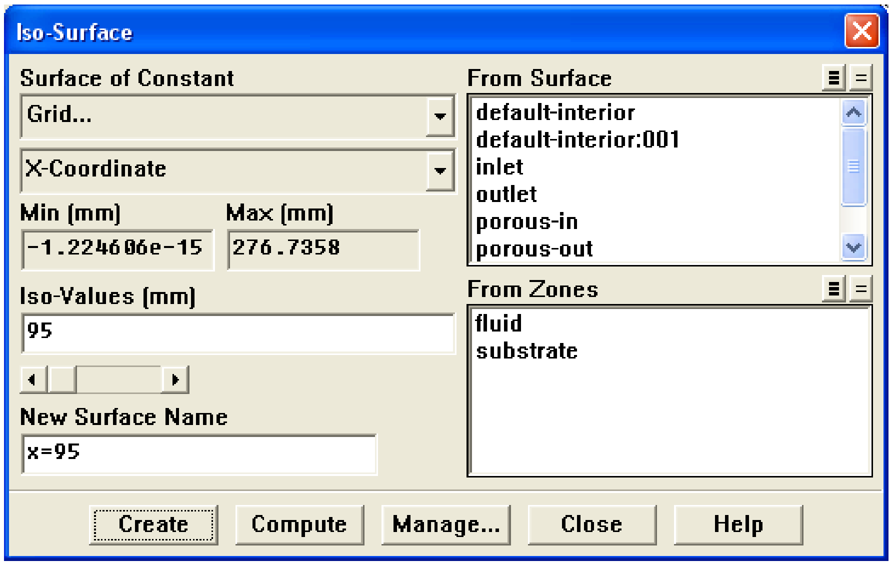
   (c) Retain the default value of 0 for the Iso-Values.

   (d) Enter y=0 for the New Surface Name.

   (e) Click Create.

2. Create cross-sectional surfaces at locations on either side of the substrate, as well as at its center.

   Surface ⟶Iso-Surface...



   (a) Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.

   (b) Click Compute to calculate the Min and Max values.

   (c) Enter 95 for Iso-Values.

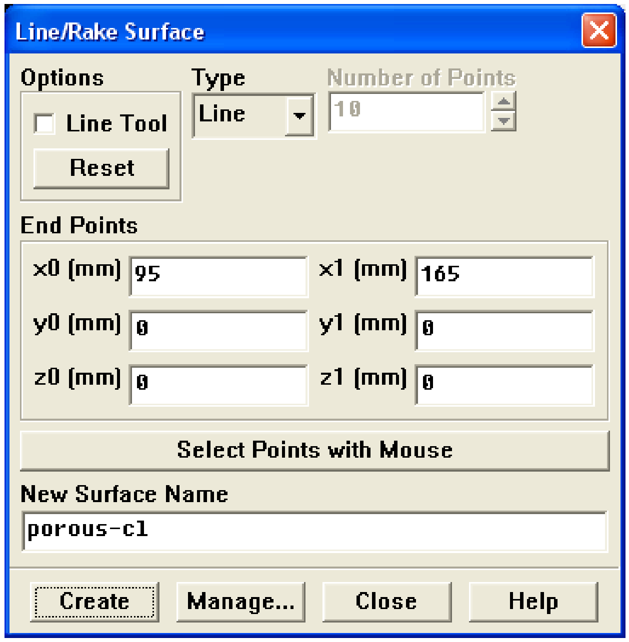
   (d) Enter x=95 for the New Surface Name.

   (e) Click Create.

   (f) In a similar manner, create surfaces named x=130 and x=165 with Iso-Values of 130 and 165, respectively. Close the Iso-Surface panel after all the surfaces have been created.

3. Create a line surface for the centerline of the porous media.

   Surface ⟶ Line/Rake...



   (a) Enter the coordinates of the line under End Points, using the starting coordinate of (95, 0, 0) and an ending coordinate of (165, 0, 0), as shown.

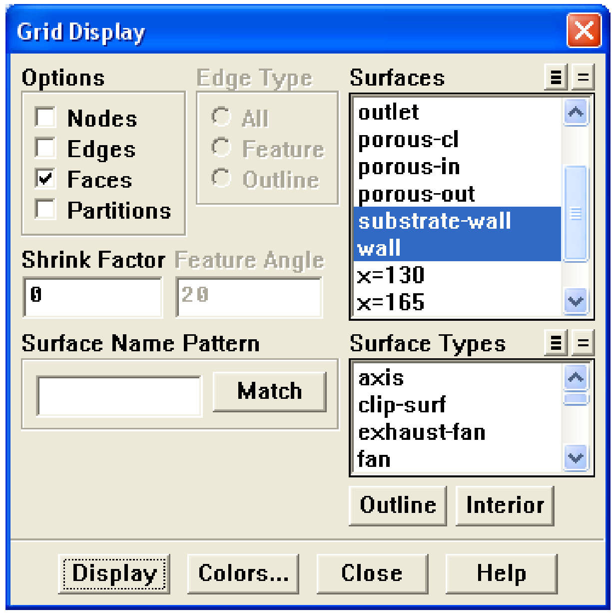
   (b) Enter `porous-cl` for the New Surface Name.

   (c) Click Create to create the surface.

   (d) Close the Line/Rake Surface panel.

4. Display the two wall zones (substrate-wall and wall).

   Display ⟶Grid...



(a) Disable the Edges option.

(b) Enable the Faces option.

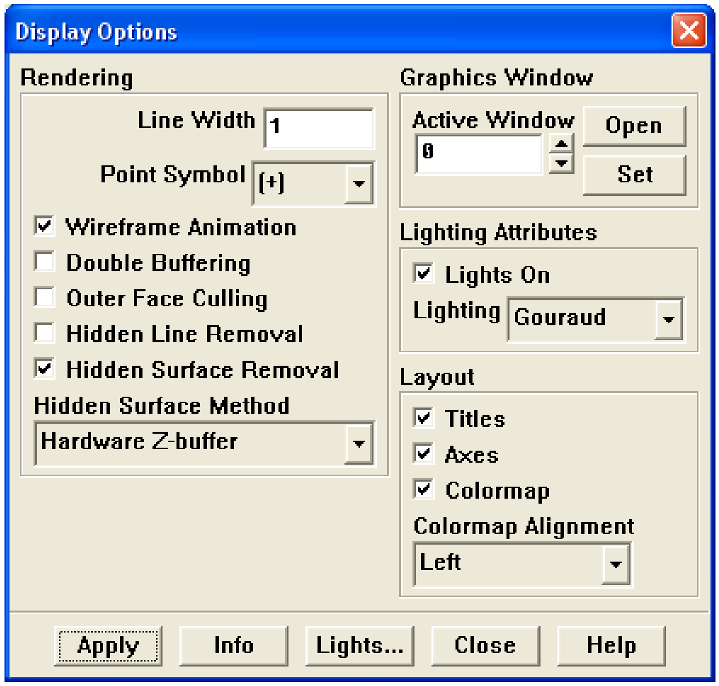
(c) Deselect inlet and outlet in the list under Surfaces, and make sure that only substrate-wall and wall are selected.

(d) Click Display and close the Grid Display panel.

(e) Rotate the view and zoom so that the display is similar to Figure 7.2.

5. Set the lighting for the display.

   Display ⟶Options...



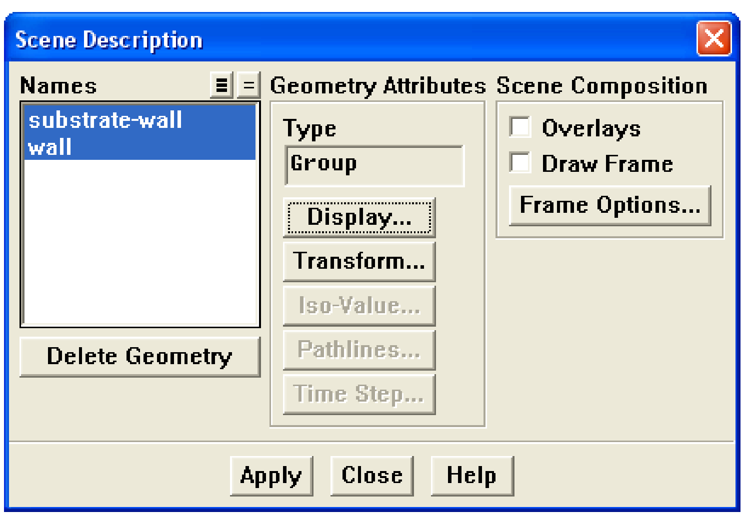
   (a) Enable the Lights On option in the Lighting Attributes group box.

   (b) Retain the default selection of Gourand in the Lighting drop-down list.

   (c) Click Apply and close the Display Options panel.

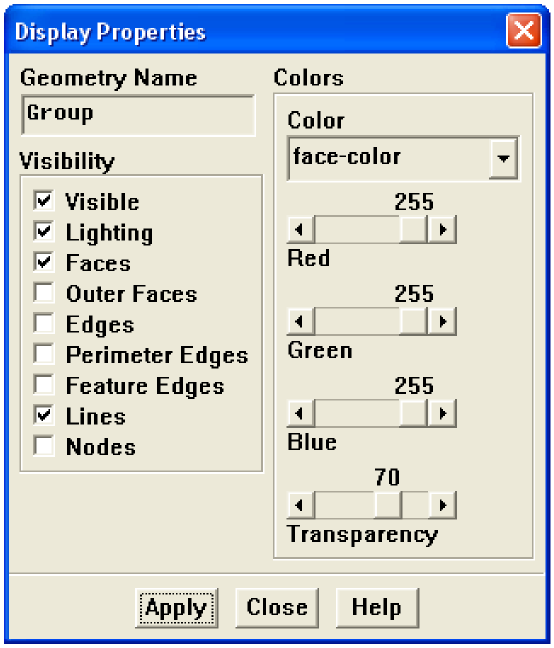
6. Set the transparency parameter for the wall zones (substrate-wall and wall).

   Display ⟶Scene...



   (a) Select substrate-wall and wall in the Names selection list.

(b) Click the Display... button under Geometry Attributes to open the Display Properties panel.



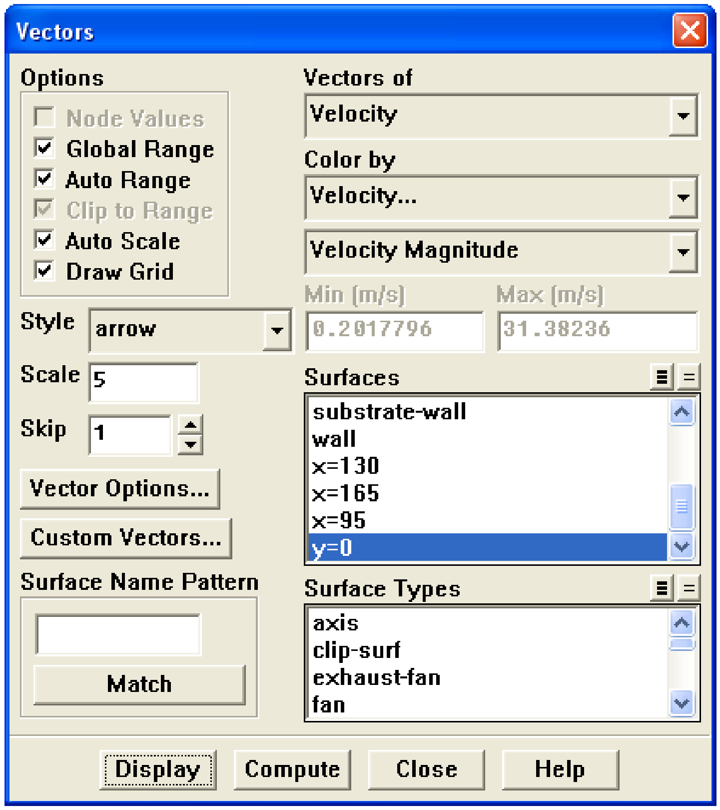
i. Set the Transparency slider to 70.

ii. Click Apply and close the Display Properties panel.

(c) Click Apply and then close the Scene Description panel.

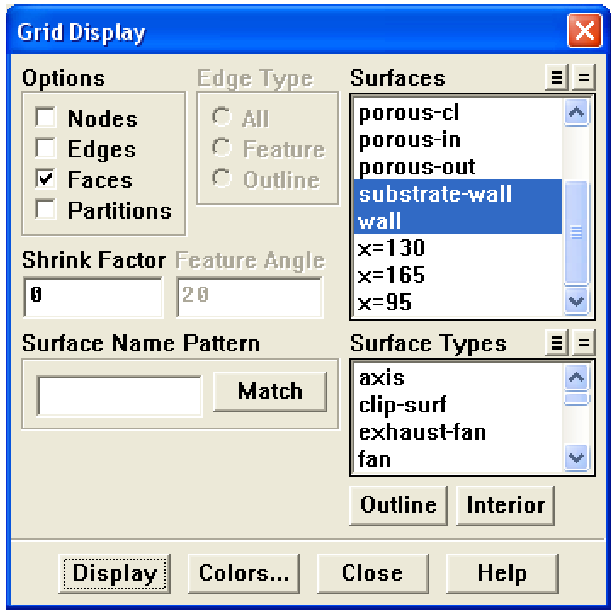
7. Display velocity vectors on the y=0 surface.

   Display ⟶Vectors...



(a) Enable the Draw Grid option.

The Grid Display *panel will open.*



i. Make sure that substrate-wall and wall are selected in the list under Surfaces.

ii. Click Display and close the Display Grid panel.

(b) Enter 5 for the Scale.

(c) Set Skip to 1.

(d) Select y=0 from the Surfaces selection list.

(e) Click Display and close the Vectors panel.

The flow pattern shows that the flow enters the catalytic converter as a jet, with recirculation on either side of the jet. As it passes through the porous substrate, it decelerates and straightens out, and exhibits a more uniform velocity distribution. This allows the metal catalyst present in the substrate to be more effective.

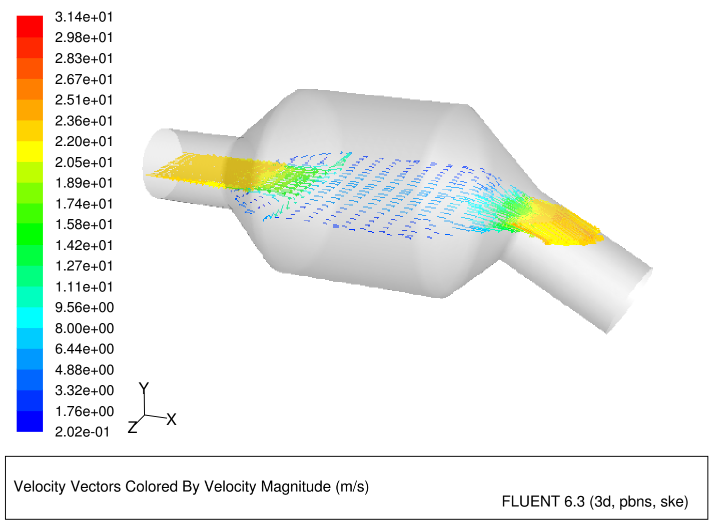
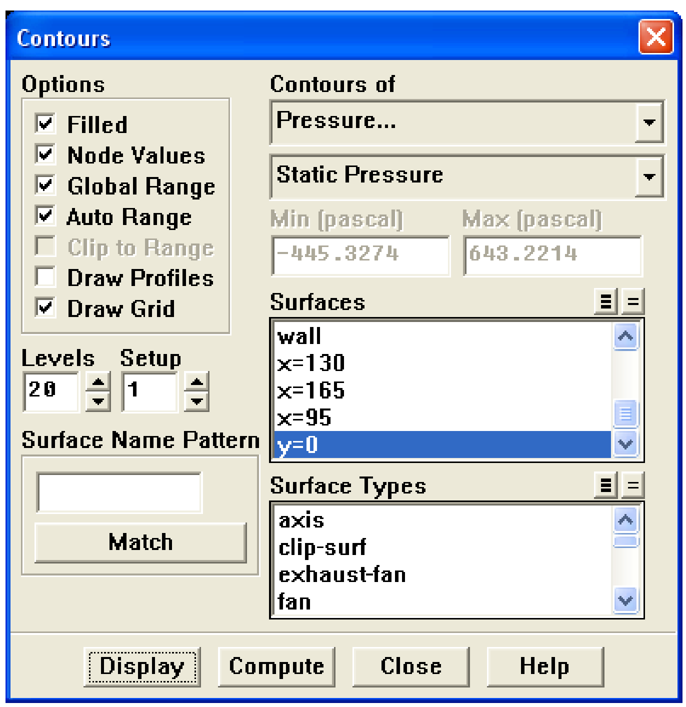


Figure 7.4: Velocity Vectors on the y=0 Plane

8. Display filled contours of static pressure on the y=0 plane.

   Display ⟶Contours...



   (a) Enable the Filled option.

(b) Enable the Draw Grid option to open the Display Grid panel.

i. Make sure that substrate-wall and wall are selected in the list under Surfaces.

ii. Click Display and close the Display Grid panel.

(c) Make sure that Pressure... and Static Pressure are selected from the Contours of drop-down lists.

(d) Select y=0 from the Surfaces selection list.

(e) Click Display and close the Contours panel.

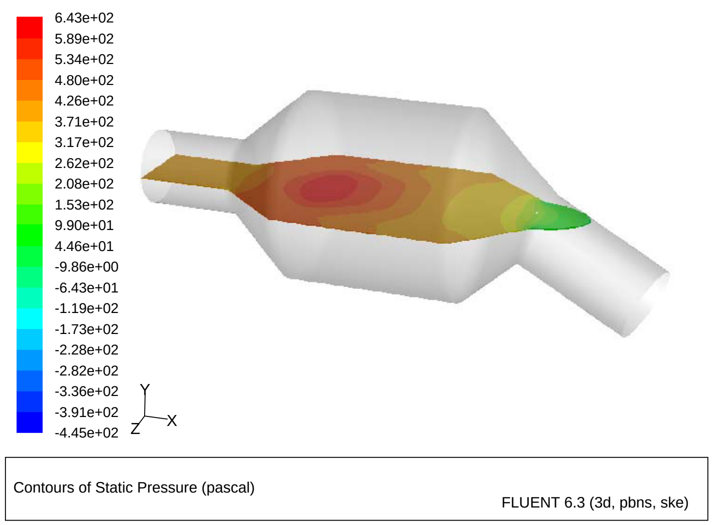
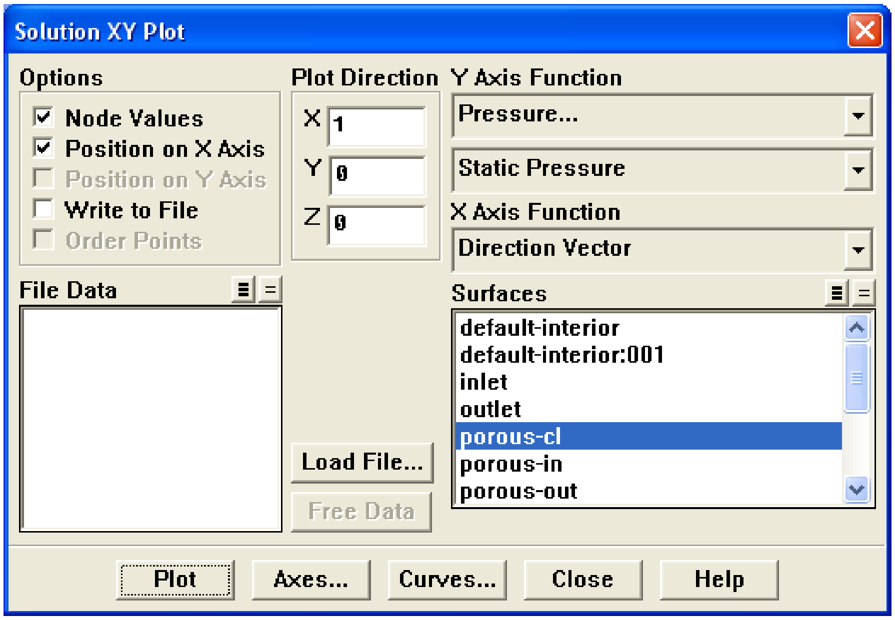


Figure 7.5: Contours of the Static Pressure on the y=0 plane

The pressure changes rapidly in the middle section, where the fluid velocity changes as it passes through the porous substrate. The pressure drop can be high, due to the inertial and viscous resistance of the porous media. Determining this pressure drop is a goal of CFD analysis. In the next step, you will learn how to plot the pressure drop along the centerline of the substrate.

9. Plot the static pressure across the line surface porous-cl.

   Plot ⟶XY Plot...



   (a) Make sure that the Pressure... and Static Pressure are selected from the Y Axis Function drop-down lists.

   (b) Select porous-cl from the Surfaces selection list.

   (c) Click Plot and close the Solution XY Plot panel.

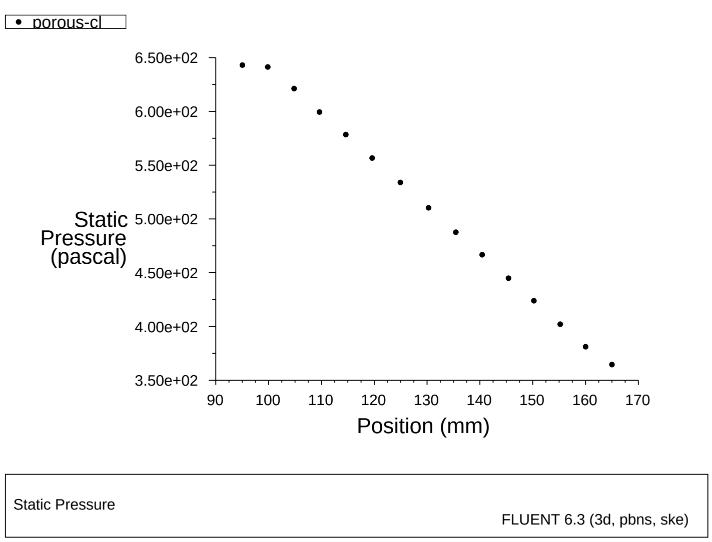
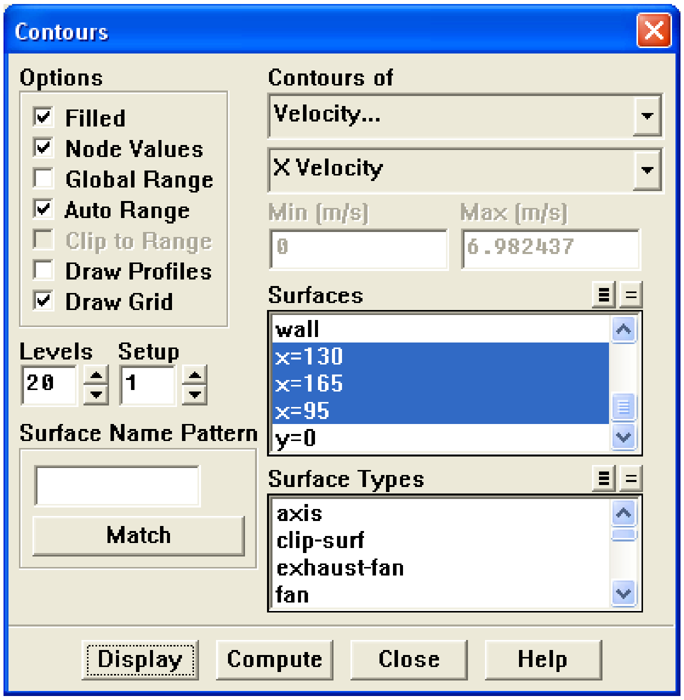


Figure 7.6: Plot of the Static Pressure on the porous-cl Line Surface

In Figure 7.6, the pressure drop across the porous substrate can be seen to be roughly 300 Pa.

10. Display filled contours of the velocity in the X direction on the x=95, x=130 and x=165 surfaces.

    Display ⟶Contours...



(a) Disable the Global Range option.

(b) Select Velocity... and X Velocity from the Contours of drop-down lists.

(c) Select x=130, x=165, and x=95 from the Surfaces selection list, and deselect y=0.

(d) Click Display and close the Contours panel.

The velocity profile becomes more uniform as the fluid passes through the porous media. The velocity is very high at the center (the area in red) just before the nitrogen enters the substrate and then decreases as it passes through and exits the substrate. The area in green, which corresponds to a moderate velocity, increases in extent.

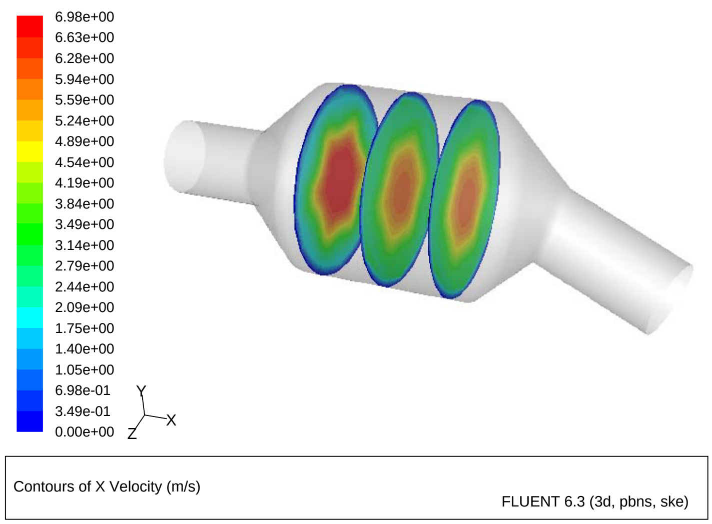
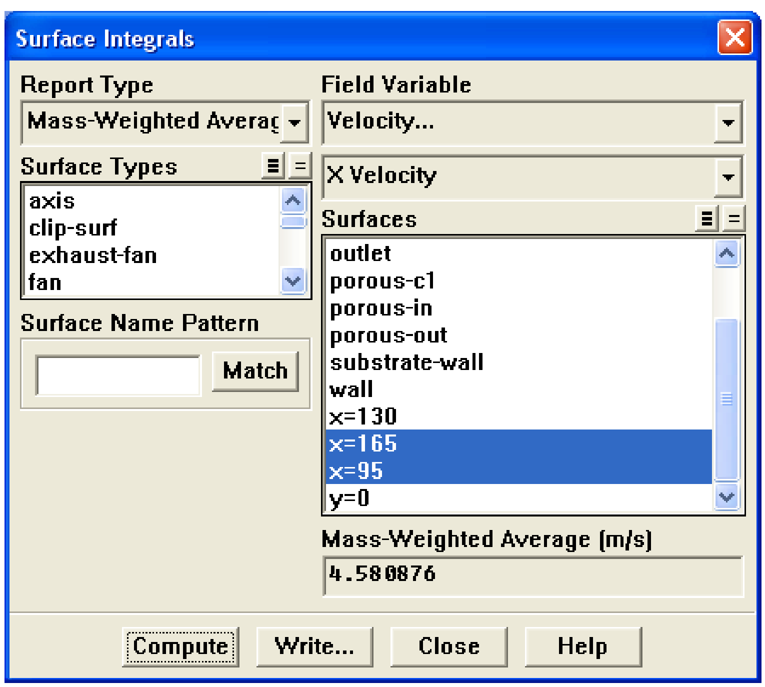


Figure 7.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces

11. Use numerical reports to determine the average, minimum, and maximum of the velocity distribution before and after the porous substrate.

    Report ⟶Surface Integrals...



    (a) Select Mass-Weighted Average from the Report Type drop-down list.

(b) Select Velocity and X Velocity from the Field Variable drop-down lists.

(c) Select x=165 and x=95 from the Surfaces selection list.

(d) Click Compute.

(e) Select Facet Minimum from the Report Type drop-down list and click Compute again.

(f) Select Facet Maximum from the Report Type drop-down list and click Compute again.

(g) Close the Surface Integrals panel.

The numerical report of average, maximum and minimum velocity can be seen in the main FLUENT *console, as shown in the following example:*

```
 Mass-Weighted Average
                     X Velocity                (m/s)
-------------------------------- --------------------
                          x=165            3.9932611
                           x=95            5.1743288
                ---------------- --------------------
                            Net            4.5808764

        Minimum of Facet Values
                     X Velocity                (m/s)
-------------------------------- --------------------
                          x=165            2.4476607
                           x=95            0.4121241
                ---------------- --------------------
                            Net            0.4121241

        Maximum of Facet Values
                     X Velocity                (m/s)
-------------------------------- --------------------
                          x=165            6.1421185
                           x=95            7.6576195
                ---------------- --------------------
                            Net            7.6576195
```

The spread between the average, maximum, and minimum values for X velocity gives the degree to which the velocity distribution is non-uniform. You can also use these numbers to calculate the velocity ratio (i.e., the maximum velocity divided by the mean velocity) and the space velocity (i.e., the product of the mean velocity and the substrate length).

Custom field functions and UDFs can be also used to calculate more complex measures of non-uniformity, such as the standard deviation and the gamma uniformity index.

Summary

In this tutorial, you learned how to set up and solve a problem involving gas flow through porous media in FLUENT. You also learned how to perform appropriate postprocessing to investigate the flow field, determine the pressure drop across the porous media and non-uniformity of the velocity distribution as the fluid goes through the porous media.

See Section 7.19 of the User's Guide for additional details about modeling flow through porous media (including heat transfer and reaction modeling).

Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in Tutorial 1.