Tutorial 7. Inviscid and Compressible Flow through a Converging-Diverging Nozzle

Introduction

The purpose of this tutorial is to illustrate the setup and solution of an axisymmetric fluid flow through a nozzle.

The flow through a converging-diverging nozzle is one of the benchmark problems used for modeling the compressible flow through computational fluid dynamics. Occurrence of shock in the flow field displays one of the most prominent effects of compressibility over fluid flow. Accurate shock predication is a challenge to the CFD fraternity. In order to resolve the high pressure gradients we need to use some special numerical schemes along with fine grid. In some cases, local grid adaption can be helpful.

This tutorial demonstrates how to do the following:

- Read an existing mesh file in FLUENT.
- Check the grid for dimensions and quality.
- Change the material properties.
- Perform inviscid calculations.
- Compare the results for different models.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

Figure 7.1 shows longitudinal section of a converging-diverging nozzle, symmetric about the axis. The length of the nozzle (L) is 0.6 m. The inlet radius (r1) is 0.1 m and the outlet radius (r2) is 0.12 m. The ratio of throat area to the inlet area is 0.5625. The pressure difference across the nozzle is 0.12 MPa.

This tutorial has two sections. In the first you will solve the problem using a turbulent model and in the second section you will use the inviscid model.

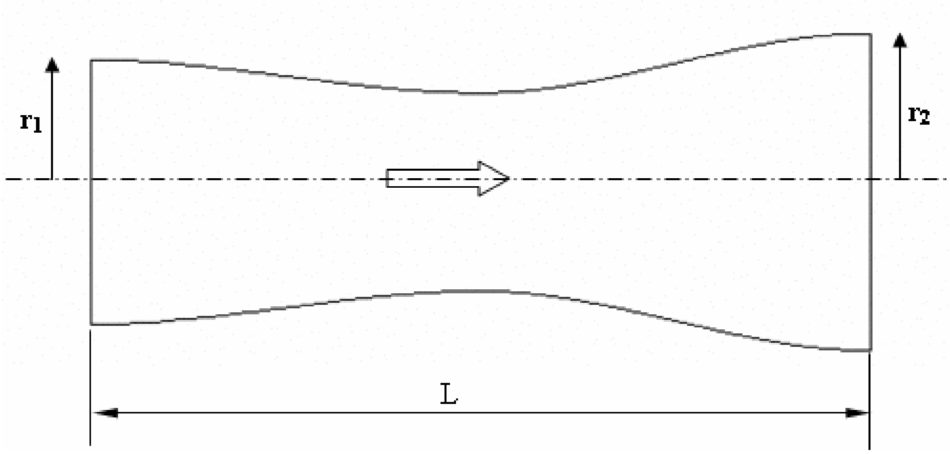


Figure 7.1: Problem Schematic

Preparation

1. Copy the mesh file, `nozzle.msh` to your working folder.

2. Start the 2D double precision (2ddp) solver of FLUENT.

Setup and Solution for Viscous Flow

Step 1: Grid

1. Read the grid file `nozzle.msh`.

   File ⟶ Read ⟶ Case...

   FLUENT *will read the mesh file and report the progress in the console.*

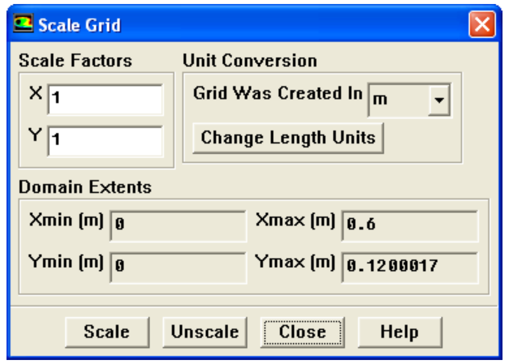
2. Check the grid.

   Grid ⟶ Check

   This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

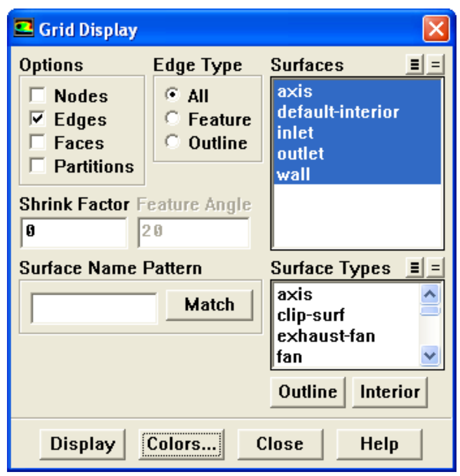
   Grid ⟶ Scale...



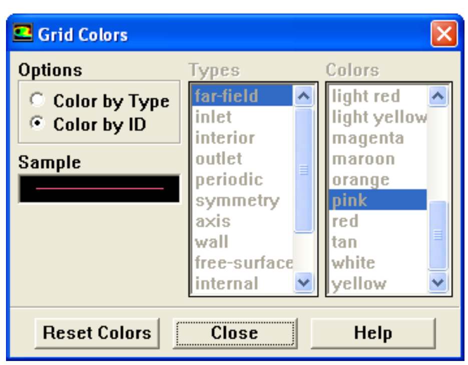
Check the domain extents to see if they correspond to the actual physical dimensions. Otherwise the grid has to be scaled with proper units.

4. Display the grid.

   Display ⟶Grid...



   (a) Click the Colors... button to open the Grid Colors panel.



      i. Select Color by ID from the Options group box.

      ii. Close the Grid Colors panel.

(b) Click Display in the Grid Display panel (Figure 7.2).

The grid adjacent to the walls is finer as compared to that in the central region. The purpose for such fine mesh is to capture sharp gradients near the walls correctly.

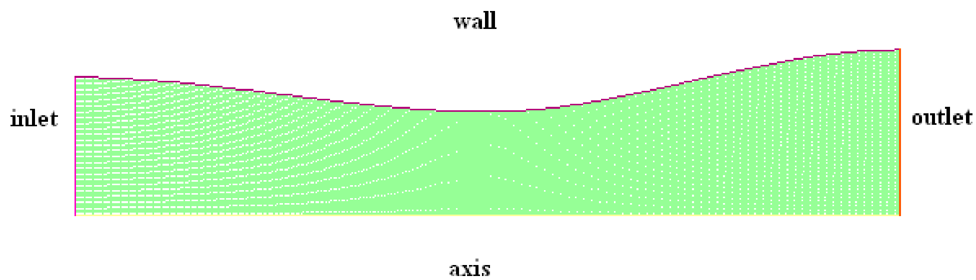


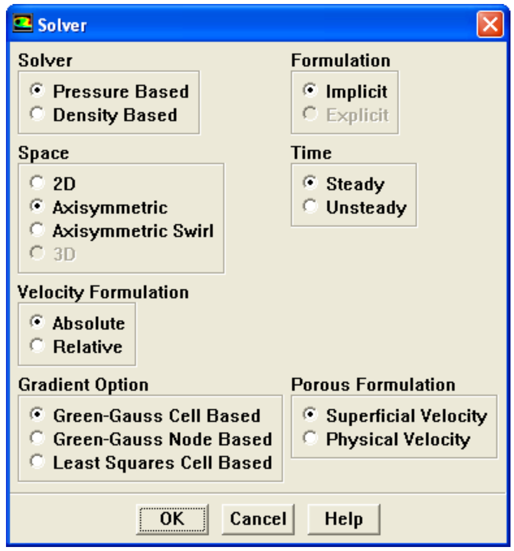
Figure 7.2: Grid Display

(c) Close the Grid Display panel.

Step 2: Models

1. Enable axisymmetric solver.

   Define ⟶ Models ⟶ Solver...

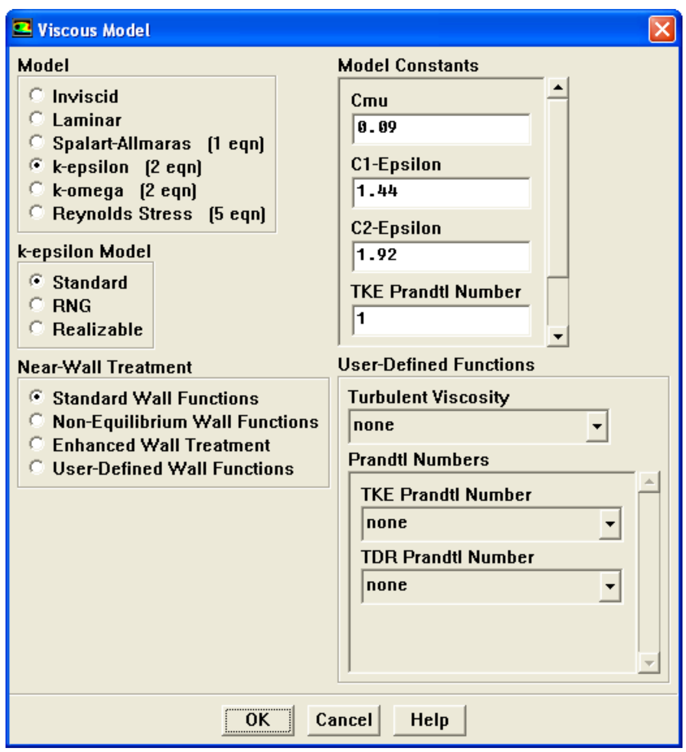


(a) Select Axisymmetric from the Space list.

(b) Click OK to close the Solver panel.

2. Enable standard k-epsilon model.

Define ⟶ Models ⟶ Viscous...



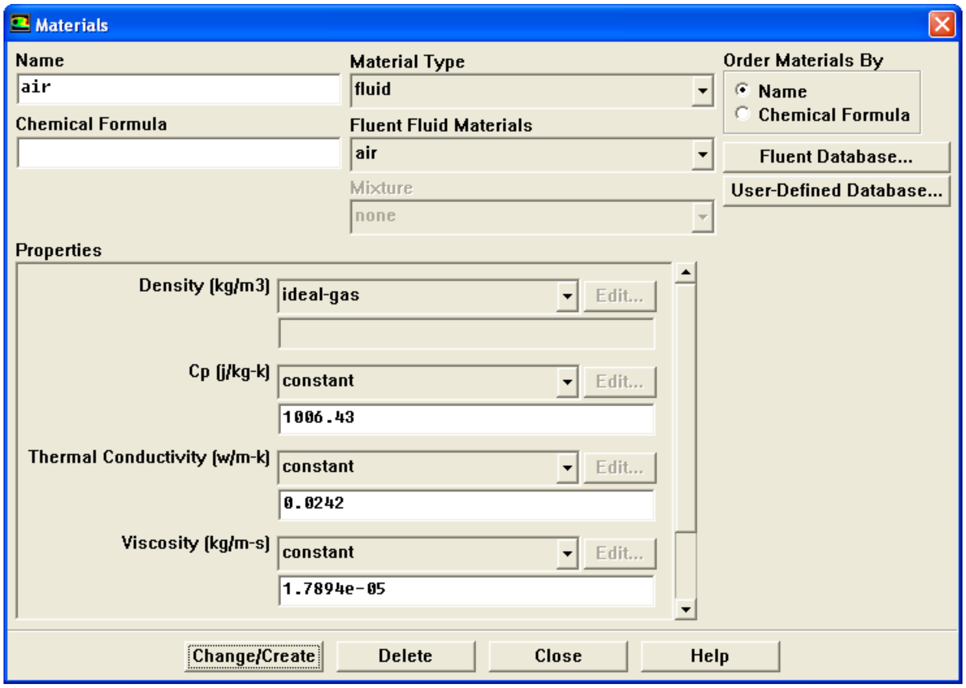
(a) Select k-epsilon (2 eqn) from the Model list.

(b) Click OK to close the Viscous Model panel.

Step 3: Materials

1. Change the properties of air.

   Define ⟶Materials...



   (a) Select ideal-gas from the Density drop-down list.

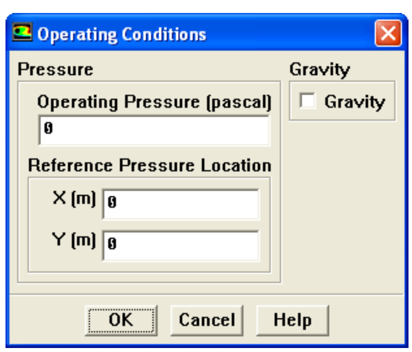
   (b) Click Change/Create and close the Materials panel.

   Energy equation will get enabled as soon as ideal-gas *density formulation is used.*

Step 4: Operating Conditions

Define ⟶Operating Conditions...

1. Enter 0 for Operating Pressure.



For compressible flows, it is recommended to set the operating pressure to zero to minimize the errors due to pressure fluctuations.

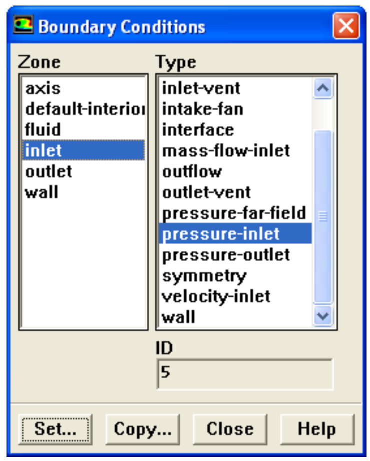
2. Click OK to close the Operating Conditions panel.

Step 5: Boundary Conditions

1. Set boundary conditions for inlet and outlet.

   Define ⟶Boundary Conditions...

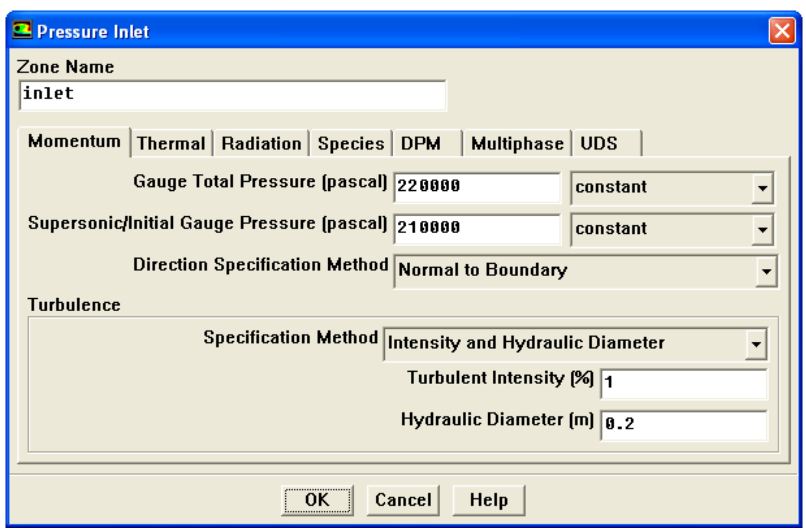
   (a) Set the boundary conditions for inlet.



   i. Select inlet from the Zone selection list.

      The Type *will be reported as* pressure-inlet*.*

   ii. Click the Set... button to open the Pressure Inlet panel.



   iii. Enter 220000 Pascal for Gauge Total Pressure and 210000 Pascal for Supersonic/Initial Gauge Pressure.

iv. Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop-down list.

v. Enter 1% and 0.2 m for Turbulence Intensity and Hydraulic Diameter respectively.

For higher Reynolds number flow, turbulent intensity is in the range of 1-5%. In this case, set it to 1% as the diameter of inlet is 0.2 m. Set the hydraulic diameter to 0.2 m.

vi. Click the Thermal tab and enter 300 K for Total Temperature.

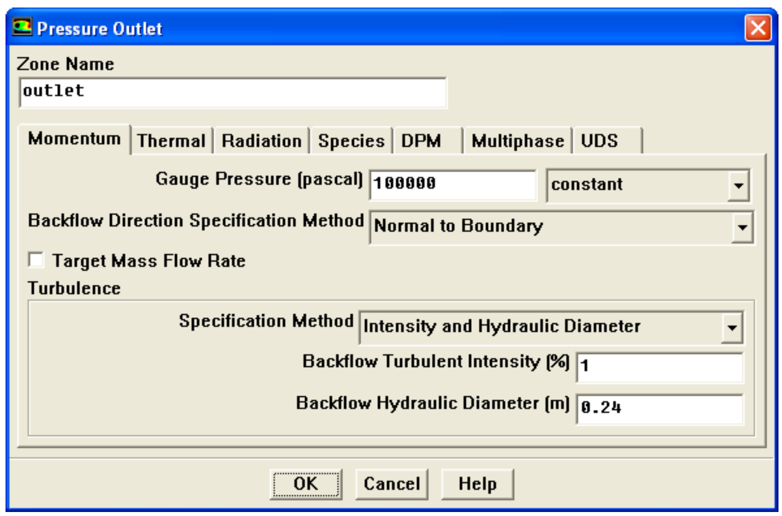
vii. Click OK to close the Pressure Inlet panel.

(b) Set the boundary conditions for outlet.

i. Select outlet from the Zone selection list.

The Type *will be reported as* pressure-outlet.

ii. Click the Set... button to open the Pressure Outlet panel.



iii. Enter 100000 Pascal for Gauge Pressure.

The outlet is assumed to open in the atmosphere. So the outlet pressure is set approximately equal to the atmospheric pressure.

iv. Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop-down list.

v. Enter 1% and 0.24 m for Turbulence Intensity and Hydraulic Diameter respectively.

The turbulence boundary conditions will be used only in case of reverse flow from the outlet. Keep the intensity of flow same as that of inlet and hydraulic diameter is set corresponding to the diameter of the outlet.

vi. Click the Thermal tab and enter 300 K for Backflow Total Temperature.

vii. Click OK to close the Pressure Outlet panel.

(c) Close the Boundary Conditions panel.

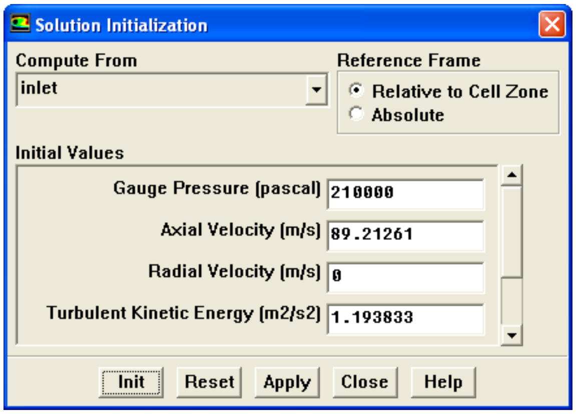
Step 6: Solution

1. Retain the default settings for the solution controls.

   Solve ⟶ Controls ⟶ Solution...

2. Initialize the flow.

   Solve ⟶ Initialize ⟶ Initialize...



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

   It will update values of all the variables based on the boundary conditions at inlet.

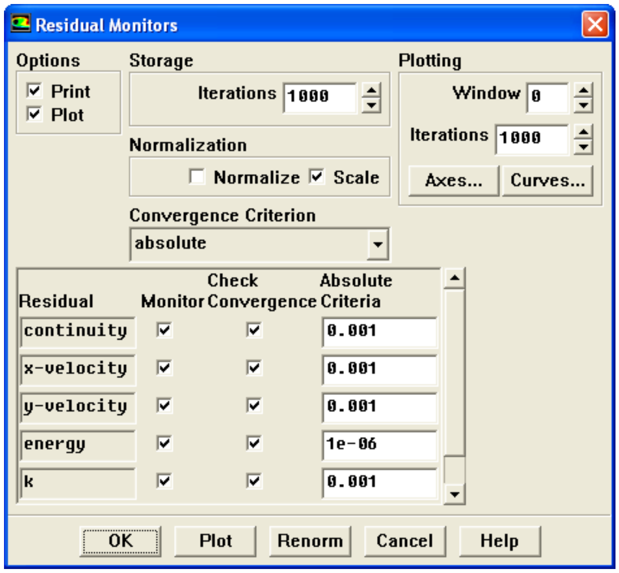
   (b) Click Apply.

   This will save the settings.

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

3. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residuals...



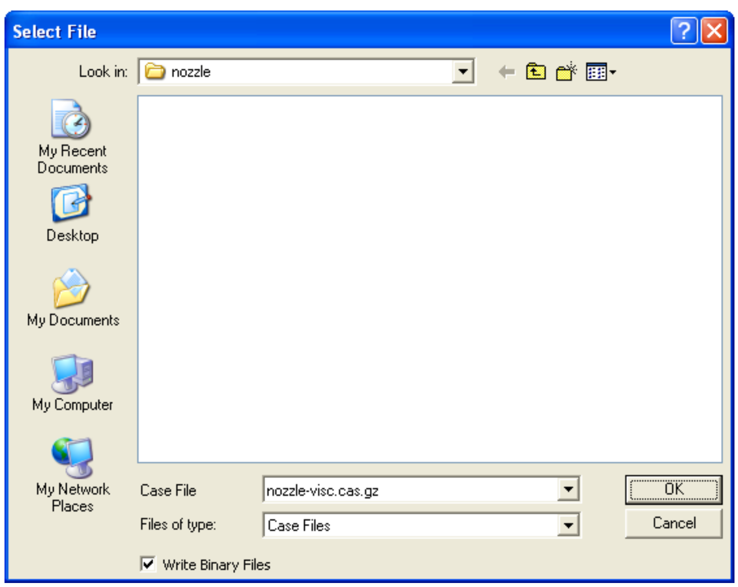
(a) Enable Plot in the Options group box.

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

4. Save the case file (`nozzle-visc.cas.gz`).

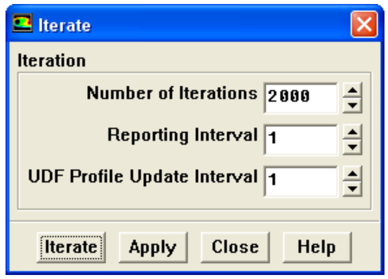
File ⟶ Write ⟶ Case...

Retain the default Write Binary Files *option so that you can write a binary file. The* **.gz** *extension will save compressed files on both,* Windows *and* LINUX/UNIX *platforms.*



5. Start the calculation by requesting 2000 iterations.

Solve ⟶ Iterate...



(a) Set Number of Iterations to 2000.

(b) Click Iterate.

The solution converges in about 1166 iterations with specified convergence criteria. The residuals plot is shown in Figure 7.3.

(c) Close the Iterate panel.

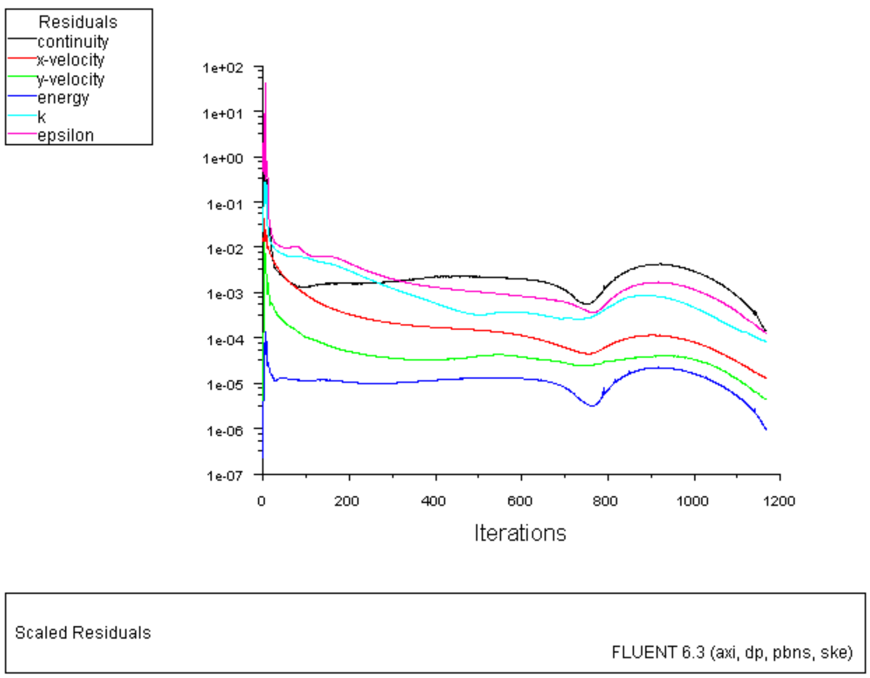


Figure 7.3: Scaled Residuals

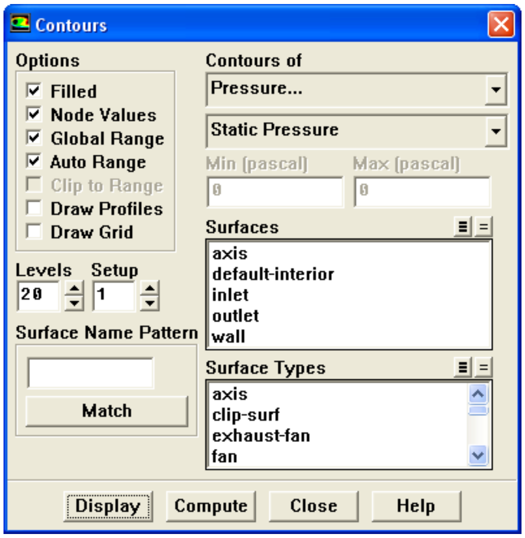
6. Save the data file (`nozzle-visc.dat.gz`).

File ⟶ Write ⟶ Data...

Step 7: Postprocessing

1. Display filled contours of static pressure (Figure 7.4).

   Display ⟶Contours...



   (a) Select Pressure... and Static Pressure from the Contours of drop-down lists.

   (b) Enable Filled in the Options group box.

   (c) Click Display.

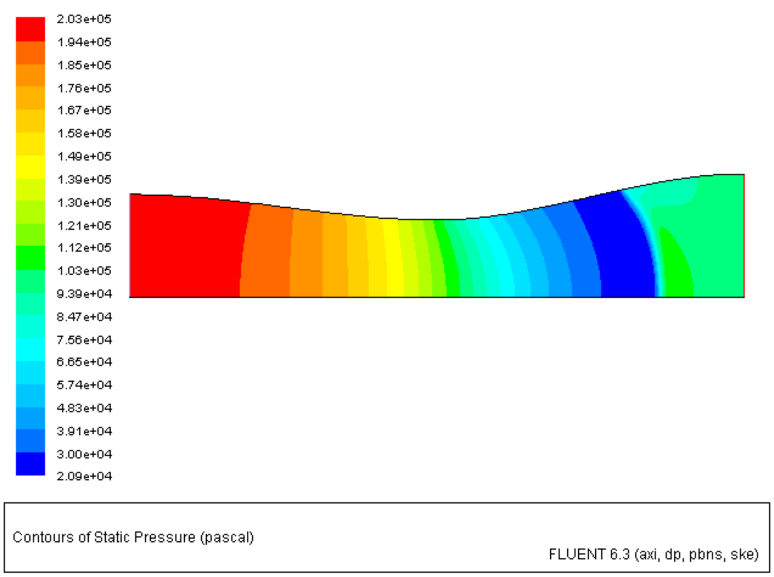
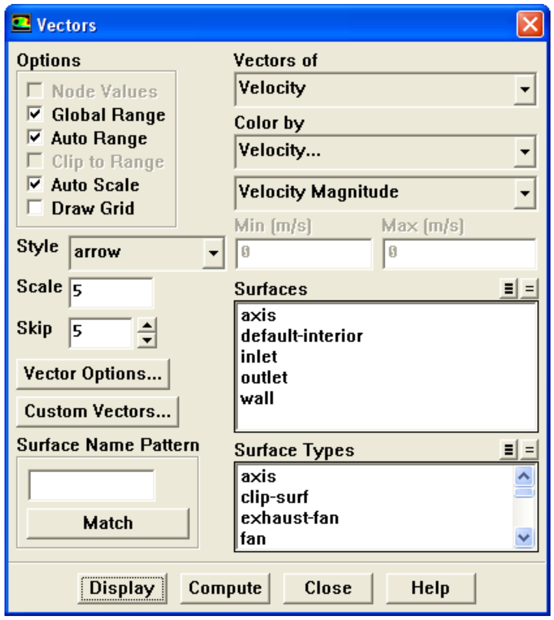


Figure 7.4: Contours of Static Pressure

   (d) Close the Contours panel.

2. Display velocity vectors (Figure 7.5).

   Display ⟶Vectors...



   (a) Select Velocity from the Vectors of drop-down list.

   (b) Select Velocity... and Velocity Magnitude from the Color by drop-down lists

   (c) Enter 5 for Scale and Skip.

   (d) Click Display.

   A sharp velocity drop can be observed at the shock.

   (e) Close the Vectors panel.

3. Zoom the view to get the display as shown in Figure 7.6.

   FLUENT *will report a message about a reversed flow in the console.*

   Observe the reversed flow at the top end of the outlet. The reason for this is the shock after which the pressure gradient becomes adverse. This causes flow separation and a vortex is formed. The pressure outlet intersects the vortex and results in a reversed flow. You can extend the domain to avoid reverse flow.

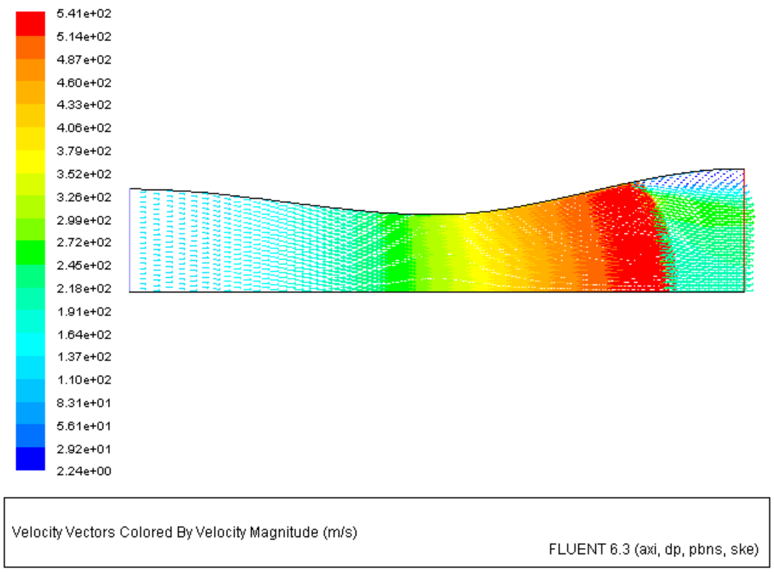


Figure 7.5: Velocity Vectors

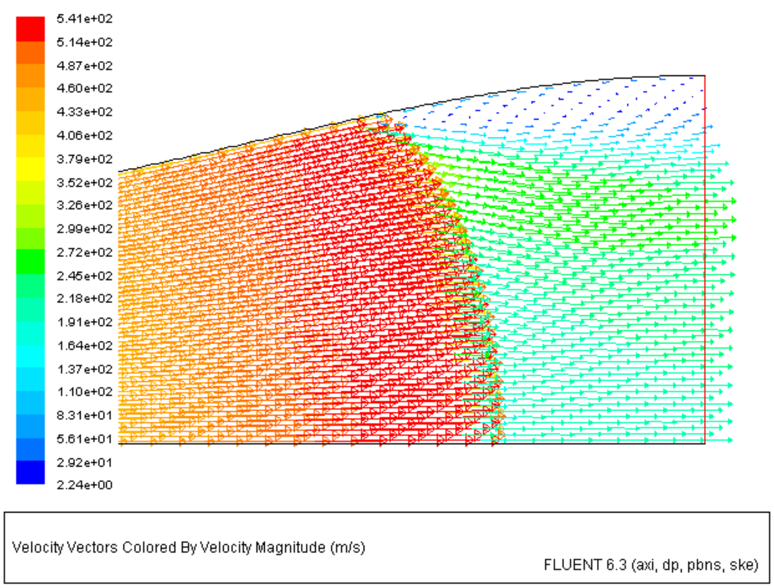


Figure 7.6: Magnified View of Velocity Vectors

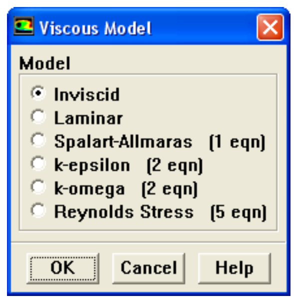
Setup and Solution for Inviscid Flow

Inviscid flow analysis neglects the effect of viscosity on the flow and are appropriate for high Reynolds number applications where inertial forces dominate the viscous forces. In this simulation the velocity is high and we can assume the flow to be inviscid. A quick estimate of shock and flow characteristics can be obtained and compared with the viscous flow simulation. The same case file can be used, only change will be to use Inviscid *as Viscous model.*

Step 8: Models

1. Enable inviscid viscous model.

   Define ⟶ Models ⟶ Viscous...



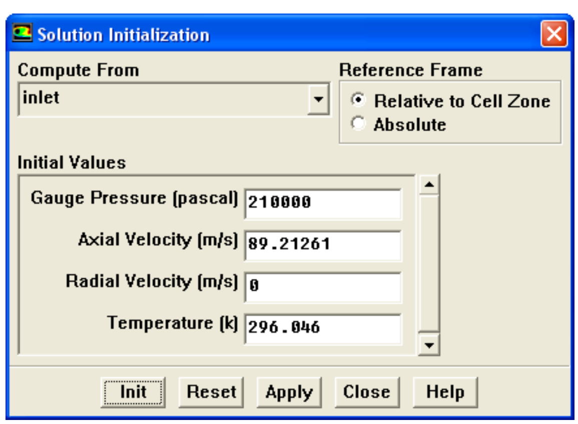
   (a) Select Inviscid from the Model list.

   (b) Click OK to close the Viscous Model panel.

Step 9: Solution

1. Initialize the flow.

   Solve ⟶ Initialize ⟶ Initialize...



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

It will update values of all the variables based on the boundary conditions at inlet.

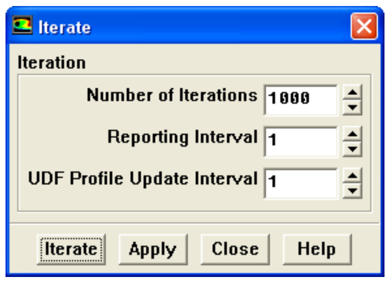
(b) Click Init and close the Solution Initialization panel.

2. Save the case file (`nozzle-invisc.cas.gz`).

File ⟶ Write ⟶ Case...

3. Start the calculation by requesting 1000 iterations.

Solve ⟶ Iterate...



(a) Set Number of Iterations to 1000.

(b) Click Iterate.

The solution converges in about 450 iterations, with the default convergence criteria. The residuals plot is shown in Figure 7.7.

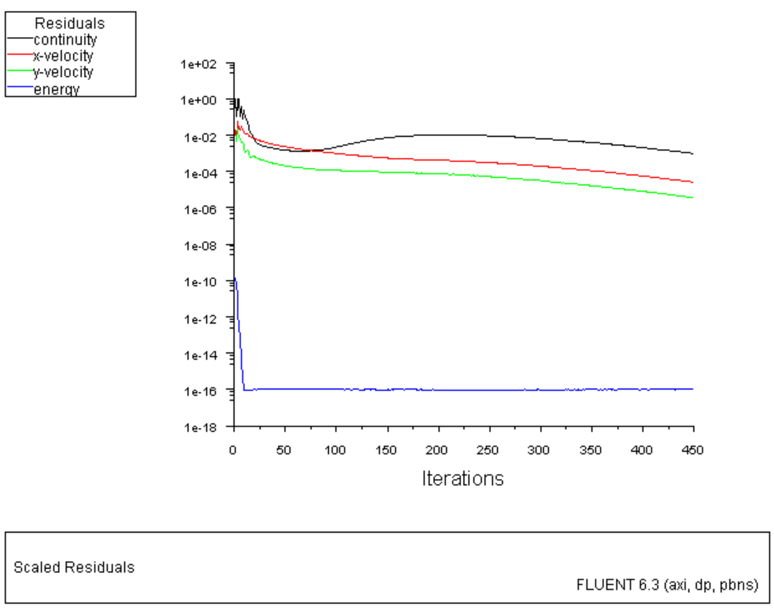


Figure 7.7: Scaled Residuals

(c) Close the Iterate panel.

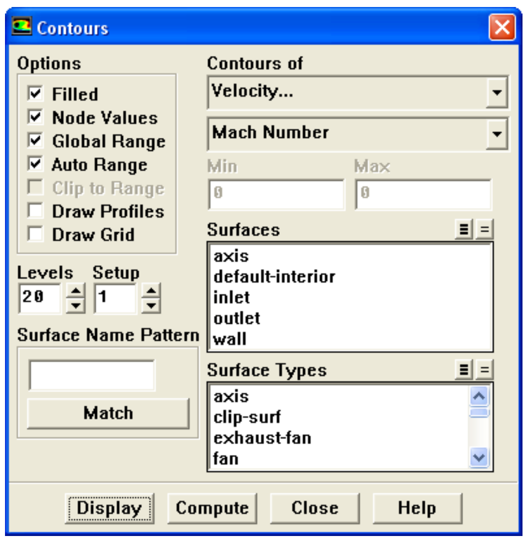
4. Save the data file (`nozzle-invisc.dat.gz`).

   File ⟶ Write ⟶ Data...

Step 10: Postprocessing

1. Display filled contours of Mach number (Figure 7.8).

   Display ⟶ Contours...



   (a) Select Velocity... and Mach Number from the Contours of drop-down list.

   (b) Click Display.

   (c) Close the Contours panel.

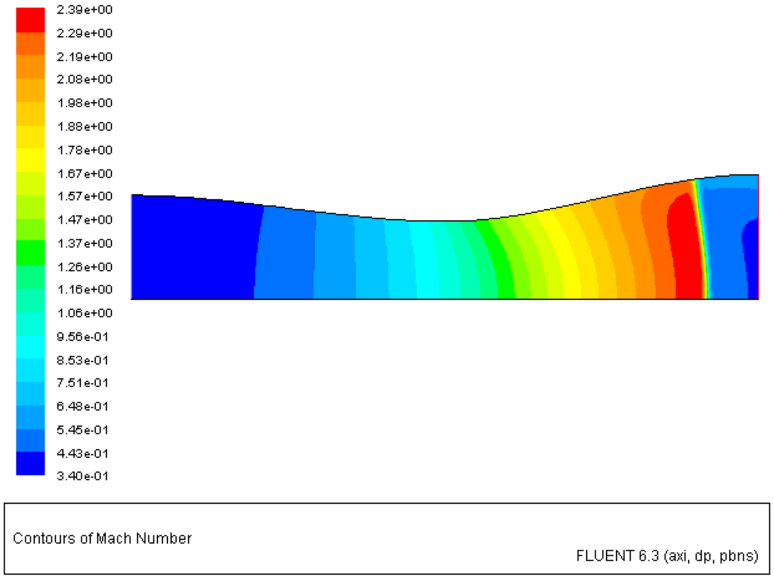


Figure 7.8: Contours of Mach Number (Inviscid Flow)

Summary

- The inviscid flow solution shows a straight shock instead of a curved shock. In viscous flow, velocity is less near the wall (i.e., in the boundary layer). So, the shock takes place near the wall before it takes place in the main flow. Hence the shock location near the wall is upstream to the shock location near the axis. This gives a curved shock.
- The FLUENT console does not show any reversed flow message. Flow separation does not occur in inviscid flows. Hence no vortex is formed near the outlet.
- The viscosity accounts to loss in momentum. As this loss is not considered in inviscid flow, a higher maximum Mach number is obtained.
- The inviscid flow solution is much faster and it predicts the shock with certain amount of over-prediction. Inviscid flow gives a quick estimate of shock location and flow characteristics.

References

J.D. Anderson, *Modern Compressible Flow*, McGraw Hill Inc., New York, 1984.

Exercises/Discussions

1. Change the values of pressure at the inlet and/or outlet to see how the shock changes position. Reach the pressure difference for which shock is observed at the throat. Can you get an idea about the viscous loss from the Mach number?

2. Check if the mesh is within the prescribed limits for wall treatment?

3. The type of shock depends on the pressure ratio, run the simulations with a range of pressure ratios to see different type of shocks like Straight, Curved, and Lambda.

4. Change the solver to 2D and compare the new result with the current result.

Links for Further Reading

- http://www.engapplets.vt.edu/fluids/CDnozzle/cdinfo.html
- http://www.grc.nasa.gov/WWW/K-12/airplane/mflchk.html
- http://www.engsoft.co.kr/download_e/steam_flow_e.htm#3.%20Choked%20Flow