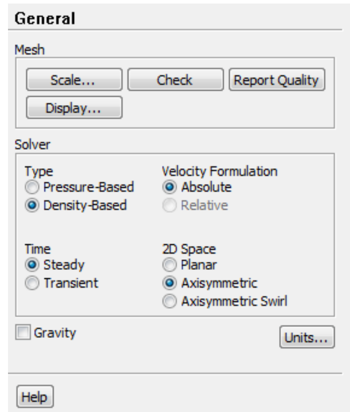
Step-by-Step Guide for Compressible Flow Analysis Using Fluent

I. Preparation

1. Download the file "2D Converging-Diverging Nozzle.msh" from Kalam and save it to your working folder

1. 2. Open workbench and drag "Fluid Flow (Fluent)" toolbox to the "Project Schematic" window
2. Right click on Mesh >Import mesh file>Browse and select the saved mesh file.
3. Click the Setup to start the 2D double precision solver of FLUENT.



II. General

Due the symmetry nature of the nozzle, only one half of the nozzle is simulated.

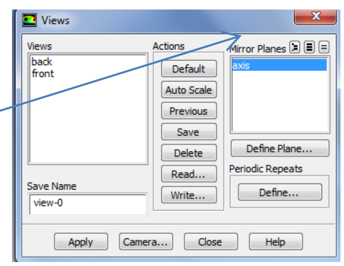
Define > General...

Since the problem is steady high-speed compressible flow of air:

1. Choose Density-based under the solver menu for Type,
2. Choose Axisymmetric under solver menu for 2D space
3. Click the Display panel to see the grid. Make sure all the surfaces are highlighted.

   Display > mesh...
4. To see the full nozzle mesh, click the Display toolbar (See the Figure).

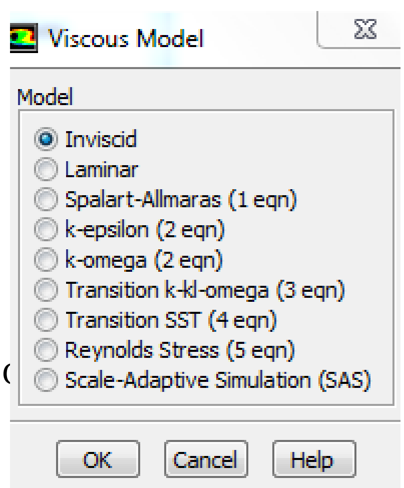
   Display > view
5. Select Axis from the Mirror Planes and Click Apply
6. Close the Views panel.
7. Click the Report Quality panel. Make sure that the orthogonal quality is close to 1.



III. Modeling

1. Since problem to be solved is steady and two-dimensional, change the viscous model settings from laminar to inviscid .

   Click Models > double click Viscous > select Inviscid then click OK



2. Enable the energy equation

   Click Models > …Double click Energy > enable the circle

IV. Defining Materials

1. Modify properties of air

   Click Materials > Create/Edit (for air)

2. Change the density to Ideal gas from the properties menu
3. Click change/create then click close

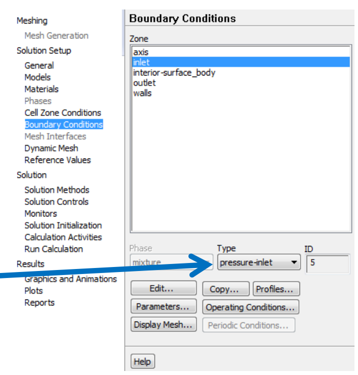
V. Cell Zone Conditions

1. Modify properties of air

   Click Cell Zone Conditions followed by Operating Conditions

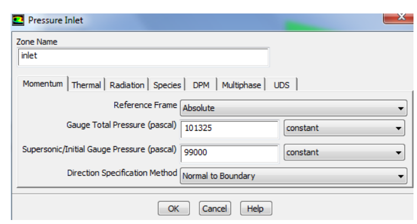
2. Enter 0 for Operating Pressure and leave the remaining default values

VI. Boundary Conditions



Click Boundary Conditions on the left menu

1. Set the boundary condition for pressure inlet.
   a. Select inlet from the Zone selection list.
   b. Select pressure-inlet from the Type selection list
   c. Click the Edit... button to open the Velocity-inlet panel.
   d. Enter 101,325 for Gauge Total Pressure (Pascal).
   e. Enter 99,000 for Supersonic/Initial Gauge pressure (Pascal)
   f. Click OK to close the pressure_inlet panel.



2. Set the boundary conditions for pressure outlet (outlet).
   a. Select outlet from the Zone selection list.
   b. Select pressure-outlet from the Type selection list.
   c. Click the Edit... button to open the Pressure Outlet panel.
   d. Enter 3700 for Gauge Total Pressure (Pascal).
   e. Click OK to close the pressure_outlet panel.
3. Retain the default boundary conditions for the Wall (wall).

4. Retain the default boundary conditions for the Axis (Axis).

VII. Reference Values

Click Reference Values on the left menu

Select inlet under Compute from

VIII. Solution

1. Set the solution methods.

   Click Solution Methods on the left menu

   a. Retain the default solution methods

2. Set the convergence criteria.

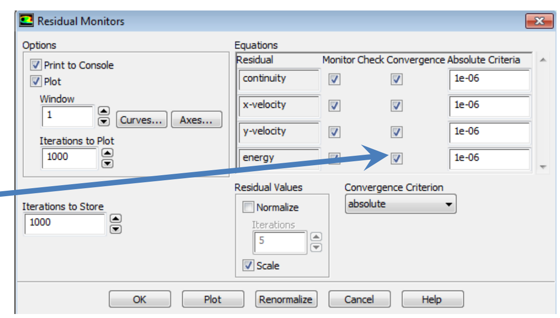
   Click Monitors on the left menu

   a. Double Click Residuals - Print, Plot from the "Residuals, statistics and force monitors" drop-down list.

   b. Enable Plot in the Options group box.

   c. Enter 1e-06 for Absolute Criteria for all the equations.

   Higher order discretization schemes and tighter convergence criteria are desirable for accurate resolution of boundary thermal boundary layer.



3. Initialize the flow.

   Click Initialization on the left menu

   a. Select Standard Initialization from the Initialization Methods list

   b. Select inlet from the "Compute From" drop-down list.

   It will update values of all the variables based on the boundary conditions at the inlet. Make sure that all values are correct.

   c. Click Initialize.

4. Save the case file (filename.cas.gz).

   File > Write > Case...

   Retain the default Write Binary Files option so that you can write a binary file.

5. Start the calculation by requesting 1000 iterations.

Solve > Run Calculation... Set Number of Iterations to 1000.

Click Calculate. (you may have to click Run twice)

IX. Post-processing

1. Display pressure and velocity contours and vectors.

   Display > Graphics and Animations...

   a. Double click Contours from the Graphics and Animation drop-down lists
   b. Select Pressure... and Static Pressure from the "Contours of" drop-down lists.
   c. Enable Filled from the Options list.
   d. Click Display.
   e. If error occur …you need to highlight all zones under surfaces

   Zoom in to get the complete temperature distribution of temperature along the length of plate

   Under step b, change static pressure to total pressure and repeat the steps

   Similarly, for the velocity contour:

   f. Select Velocity... and Mack Number from the "Contours of" drop-down lists.
   g. Enable Filled from the Options list.
   h. Click Display.

   Zoom in to get the complete temperature distribution of temperature along the length of plate

   Similarly, for the temperature contour:

   i. Select Temperature... and Static Temperature from the "Contours of" drop-down lists.
   j. Enable Filled from the Options list.
   k. Click Display.

   Zoom in to get the complete temperature distribution of temperature along the length of plate

   DO THE SAME FOR TOTAL TEMPERATURE

2. Display the velocity vectors.
   a. Double click Vectors from the Graphics and Animation drop-down lists
   b. Select Velocity... and Velocity Magnitude from the "Vector of " drop-down lists.
   c. Enable Filled from the Options list.
   d. Click Display.

   Zoom in to get the complete temperature distribution of temperature along the length of plate

3. Plot the velocity, Mack number, and pressure profiles

   Display > Plots...

   a. Double click XY Plots from the Plots drop-down lists
   b. Select (highlight) *axis* and *wall* from the list of surfaces on the right side under Results, Plots-XY Plot-Set Up.
   c. Select Velocity from the Y Axis Function, and select Mach number, from the row below it.
   d. Click Plot
   e. You can save the data in excel format by clicking write to File under options and then click write

   Similarly,

   f. Select Pressure from the Y Axis Function, and select Static Pressure from the raw below it.
   g. Click Plot

   Similarly,

   h. Select Temperature from the Y Axis Function, and select Static Temperature from the raw below it.
   i. Click Plot

Reference

FLUENT 6.1 Tutorial Guide, February 2003, file:///C:/Users/FKM%2311/Downloads/documents.mx_fluent-tutorials-55844f4e977b8.pdf