
Flow Through a Converging-Diverging Nozzle

Abstract—*In this exercise, two-dimensional or axisymmetric flow through a converging-diverging nozzle is modeled. Coarse, medium, and fine mesh types are available. Inviscid or viscous flow fields can be simulated. The material properties of air are used with ideal gas behavior assumed for density calculations. Inlet Mach number, exit pressure, pressure ratio, and inlet temperature can be specified. Contours of pressure, velocity, temperature, Mach number, stream function, turbulent kinetic energy, and dissipation rate can be displayed. A velocity vector plot is also available.*

1 Introduction

The purpose of this exercise is to simulate the operation of a converging-diverging nozzle, which is an important and basic component associated with propulsion and the high speed flow of gases. In this exercise, both inviscid and viscous flows through a converging-diverging nozzle can be investigated. Mach number variation and shock formation may be evaluated.

2 Modeling Details

FlowLab creates the geometry and mesh, and exports the mesh to FLUENT. The boundary conditions and flow properties are set through parameterized case files. The FLUENT solver performs iterations until the convergence limit is met or the number of iterations specified is achieved.

2.1 Geometric Details

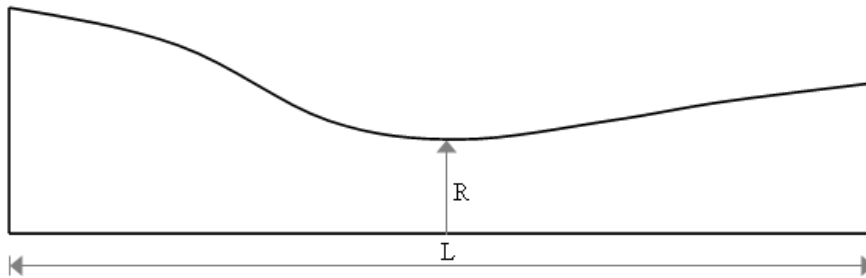


Figure 2.1: Schematic of the Nozzle Geometry Using Nozzle Wall Profile

There are three methods available for creating the nozzle geometry:

1. By importing a wall profile data file of the nozzle into FlowLab. The wall data file is a simple text file with a `.dat` extension. It contains x-, y-, and z-coordinate data representing the nozzle wall profile using three columns as shown in Table 2.1.

x-coordinate	y-coordinate	z-coordinate
0.0000000000e+000	2.5000000000e+000	0
5.1402296865e-002	2.4960978530e+000	0
1.0280185069e-001	2.4921597913e+000	0
1.5419616548e-001	2.4881540183e+000	0
2.0558258051e-001	2.4840481938e+000	0
2.5695681531e-001	2.4797928487e+000	0

Table 2.1: Coordinate Data Representing Nozzle Wall Profile

The radius of the throat can also be specified in the geometry panel when using wall profile data.

2. By importing an area data file. This is similar to the vertex data file approach previously described, except that the second column contains the cross-section area of the nozzle at the corresponding x-coordinate.
3. By using the inbuilt curve option available in the exercise. The inbuilt curve is a parabolic profile as defined by the following equation:

$$y = R + (c \times (x - L_{th}) \times (x - L_{th})) \quad (2-1)$$

where,

- y = y-coordinate
- R = Radius of the throat
- c = User-specified nozzle-wall curvature
- x = x-coordinate
- L_{th} = x-coordinate of throat location

The recommended range of nozzle-wall curvature is 0.0 to 0.15.

2.2 Mesh

Coarse, medium, and fine mesh types are available. Mesh density varies based upon the assigned Refinement Factor. The Refinement Factor values for the mesh densities are given in Table 2.2.

Mesh Density	Refinement Factor
Fine	1
Medium	1.5
Coarse	2.25

Table 2.2: Refinement Factor

First Cell Height is kept at 0.015 m for inviscid flows. In the case of viscous flows, First Cell Height is calculated using the following formula:

$$First\ Cell\ Height = Refinement\ Factor \left[\frac{Y_{plus} \times (Characteristic\ Length^{0.125} \times Viscosity^{0.875})}{(0.199 \times Velocity^{0.875} \times Density^{0.875})} \right] \quad (2-2)$$

Reynolds number based upon inlet height for the two-dimensional setup or inlet diameter for the axis-symmetric setup, is used to determine Yplus. A standard wall function is used to model near wall turbulence effects, with Yplus > 30.

The number of intervals along each edge is determined using geometric progression and the following equation:

$$Intervals = INT \left[\frac{\text{Log} \left\{ \frac{Edge_Length \times (Growth_ratio - 1)}{First\ Cell\ Height} + 1.0 \right\}}{\text{Log}(Growth_ratio)} \right] \quad (2-3)$$

The edges are meshed using the First Cell Height and the calculated number of intervals. The entire domain is meshed using a map scheme. The resulting mesh is shown in Figure 2.2.

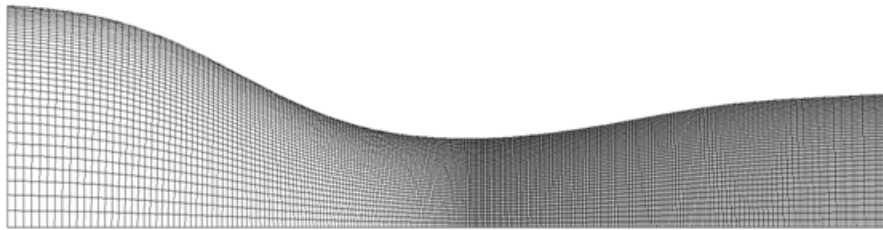


Figure 2.2: Mesh Generated by FlowLab Showing Fine Mesh Density

2.3 Physical Models for FLUENT

Either inviscid flow or viscous flow can be modeled. For a viscous flow simulation, the standard k-ε turbulence model is applied.

2.4 Material Properties

The fluid in the simulation is air, and the following material properties are used:

- Density (ideal gas law)
- Thermal conductivity (only for viscous flow)
- Viscosity (only for viscous flow)
- Specific heat
- Molecular weight

2.5 Boundary Conditions

The following boundary conditions can be specified:

- Mach number at the inlet
- Static exit pressure
- Pressure ratio—the ratio of static pressure at the exit to static pressure at the inlet
- Static temperature at the inlet

The following boundary conditions are assigned in FLUENT:

Boundary	Assigned As
Nozzle	Wall
Inlet	Pressure Inlet
Exit	Pressure Outlet

Table 2.3: Boundary Conditions Assigned in FLUENT

2.6 Solution

The mesh is exported to FLUENT along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file. Instructions for the solver are provided through a **journal file**. When the solution is converged or the specified number of iterations is met, FLUENT exports data to a **neutral** file and to **.xy** plot files. GAMBIT reads the **neutral** file for postprocessing.

3 Scope and Limitations

This exercise is restricted to high-speed flows. It is not possible to solve a laminar flow field. Pressure ratio should be maintained between 0.1 and 1 to ensure accurate results. The recommended range of inlet Mach number is 0.1 to 3. The inlet temperature should be maintained between 270 K to 400 K to ensure good results. If you use input values outside the upper or lower limits suggested in the problem overview, it may result in poor accuracy or there may be difficulty in obtaining convergence. When the inlet is supersonic, a shock will be produced in the converging section. As the solution proceeds, the shock may move towards the inlet and contact the inlet boundary. Once this occurs, the supersonic boundary condition at the inlet will not prevail. Plotted results (*XY* plots) may be underpredicted or overpredicted at the location of the shock. These inaccuracies are due to numerical error, and can generally be resolved by using sufficiently fine mesh. However, resolution is limited in this exercise to that produced by fine mesh setting, and numerical error is present at this level of grid refinement.

4 Exercise Results

4.1 Reports

Average Mach number is reported at the outlet.

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Mach number along the centerline
- Mach number along the wall
- Static pressure along the centerline
- Static pressure along the wall
- Total pressure along the centerline
- Total pressure along the wall
- Total pressure multiplied by throat area along the axis
- Cross-sectional area divided by throat area along the axis
- Nozzle wall profile

Figure 4.1 presents a plot of Mach number along the centerline.

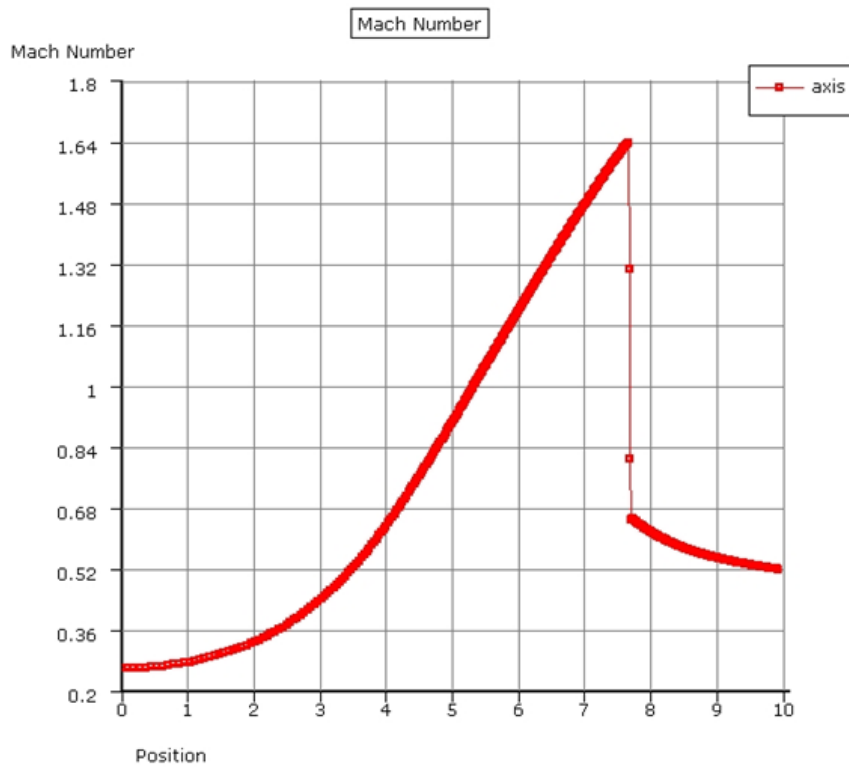


Figure 4.1: Plot of Mach number Along the Centerline

4.3 Contour Plots

Contour plots of pressure, total pressure, temperature, velocity magnitude, Mach number, stream function, x-velocity, and y-velocity are available for both viscous and inviscid flows. Turbulent kinetic energy and dissipation rate are available for viscous flows. Contours of static pressure are presented in Figure 4.2.

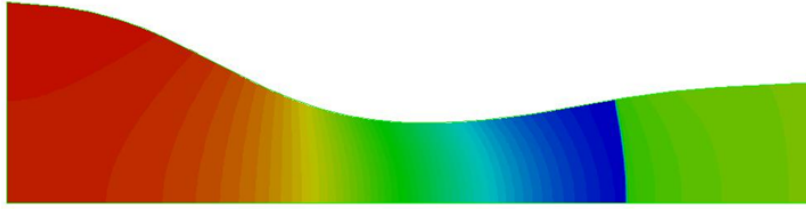


Figure 4.2: Contours of Static Pressure for the Default Case

5 Verification of Results

Exercise results can be compared with the quasi-1D solution for a converging-diverging nozzle.

$$\frac{T}{T_0} = \left[1 + \left(\frac{\gamma - 1}{2} \right) M^2 \right]^{-1} \quad (5-1)$$

$$\frac{p}{p_0} = \left[1 + \left(\frac{\gamma - 1}{2} \right) M^2 \right]^{\frac{\gamma}{\gamma - 1}} \quad (5-2)$$

$$\frac{A}{A_*} = \frac{A}{A_T} = \frac{1}{M} \left[\left(\frac{2}{\gamma + 1} \right) \left(1 + \left[\frac{\gamma - 1}{2} \right] M^2 \right) \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \quad (5-3)$$

6 Sample Problems

1. Solve the flow through different nozzle geometries and observe the flow characteristics.
2. Run the simulation for a range of pressure ratios and determine the effect on flow downstream of the nozzle.
3. Find the effect of back pressure on the flow field.

7 Reference

- [1] *Saad, M. A.*, “Compressible Fluid Flow”, 2nd Edition, Prentice Hall, NJ, 1994
- [2] *J. Anderson*, “Modern Compressible Flow: With Historical Perspective”, 2nd Edition, Mc Graw Hill, New York, 1990