FLUENT - Compressible Flow in a Nozzle- Step 6

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University



Step 6: Results

Useful Information

These instructions are for FLUENT 6.3.26. Click here for instructions for FLUENT 12.

Velocity Vector

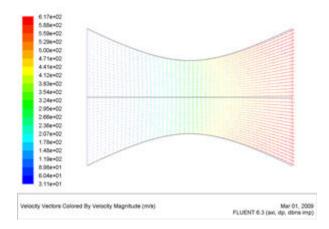
Let's first look at the velocity vector in the nozzle.

Display > Vectors...

Select Velocity under Vectors of and Velocity ... under Color by. Set Scale to 0.4

Vectors			×
Options	Vectors of		
☐ Node Values	Velocity		•
Global Range	Color by		
Clip to Range	Velocity		÷
Auto Scale	Velocity Mag	Velocity Magnitude	
T Draw Grid	Min (m/s)	Max (m/s)	
Style arrow	• 31.14225	617.261	
Scale 0.4	Surfaces		==
Skip 0 1	centerline default-interi inlet outlet wall	or	
Custom Vectors Surface Name Patte		ç	= =
Match	axis clip-surf exhaust-fan fan		-

Click Display.



Higher Resolution Image

We see that the flow is smoothly accelerating from subsonic to supersonic.

To include the lower half of nozzle, do the following:
 Display > Views...
 Select centerline and click Apply

White Background on Graphics Window

To get white background go to: Main Menu > File > Hardcopy Make sure that *Reverse Foreground/Background* is checked and select *Color* in *Coloring* section. Click *Preview*. Click *No* when prompted " *Reset graphics window?*"

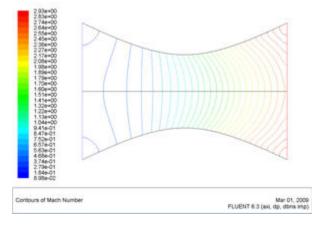
Mach Number Contour

Let's now look at the Mach number

Display > Contours

Select Velocity... under Contours of and select Mach Number. Set Levels to 30.

Options	Contours of	
Filled	Velocity	
Node Values Global Range	Mach Number	
Auto Range	Min	Max
Clip to Range	8	0
Draw Profiles Draw Grid	Surfaces	11
Levels Setup 30 + 1 + Surface Name Patte	centerline default-interior inlet outlet wall	
	Surface Types	I =
Match	axis clip-surf exhaust-fan fan	< >



Higher Resolution Image

For 1D case, mach number is a function of x position. For 1D case, we are supposed to see vertical contour of mach numbers that are parallel to each other.

For 2D case, we are seeing curving contour of mach number. The deviation from vertical indicates the 2D effect.

Do note that 1D approximation is fairly accurate around the centerline of nozzle.

Pressure Contour Plot

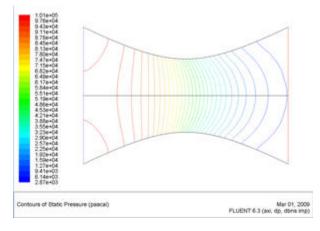
Let's look at how pressure changes in the nozzle.

Display > Contours...

Select Pressure... and Static Pressure under Contours of. Use Levels of 30

Options	Contours of	
Filled	Pressure	
Node Values Global Range	Static Pressure	
Auto Range	Min (pascal)	Max (pascal)
Clip to Range	2874.701	100853.3
Draw Profiles	Surfaces	=
Levels Setup 30 + 1 + Surface Name Patte	centerline default-interio inlet outlet wall	r
	Surface Types	
Match	axis clip-surf exhaust-fan fan	

Click Display.





Notice that the pressure decreases as it flows to the right.

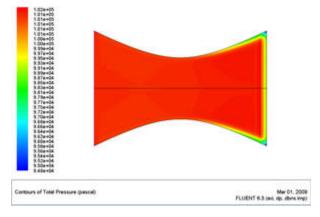
Total Pressure Contour Plot

Let's look at the total pressure in the nozzle

Display > Contours...

Select Pressure... and Total Pressure under Contours of. Select Filled. Use Levels of 100.

Options	Contours of	
 ✓ Filled ✓ Node Values ✓ Global Range 	Pressure	
	Total Pressure	
Auto Range	Min (pascal)	Max (pascal)
Clip to Range	94764.66	101664.6
Draw Profiles Draw Grid	Surfaces	
Levels Setup 100 1 1 1 Surface Name Patte	centerline default-interio inlet outlet wall	or
	Surface Types	e ()
Match	axis clip-surf exhaust-fan fan	





Around the nozzle outlet, we see that there is a pressure loss because of the numerical dissipation.

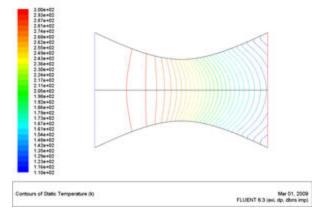
Temperature Contour Plot

Let's investigate or verify the temperature properties in the nozzle.

Display > Contours...

Select Temperature... and Static Temperature under Contours of. Use Levels of 30.

Options	Contours of	
Filled	Temperature Static Temperature	
Node Values Global Range		
Auto Range	Min (k)	Max (k)
Clip to Range	109.9191	299.601
Draw Profiles Draw Grid	Surfaces	= =
Levels Setup 30 + 1 + Surface Name Patte	centerline default-interi inlet outlet wall	or
	Surface Type	s II =
Match	axis clip-surf exhaust-fan fan	~ •





As we can see, the temperature decreases from left to right in the nozzle, indicating a transfer of internal energy to kinetic energy as the fluid speeds up.

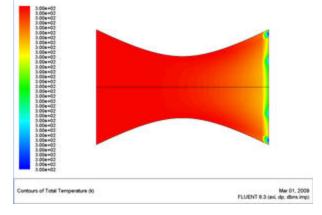
Total Temperature Contour Plot

Let's look at the total pressure in the nozzle

Display > Contours...

Select Temperature... and Total Temperature under Contours of. Select Filled. Use Levels of 100.

Options	Contours of	
Filled	Temperature	
Node Values Global Range	Total Temperature	
Auto Range	Min (k)	Max (k)
Clip to Range	299.9712	300
Draw Profiles Draw Grid	Surfaces	
Levels Setup 100 + 1 + Surface Name Patte	centerline default-interi inlet outlet wall	or
	Surface Type	S 📕 =
Match	axis clip-surf exhaust-fan fan	< >

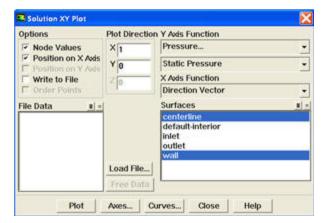


Higher Resolution Image

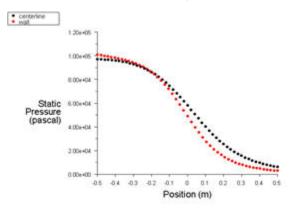
Looking at the scale, we see that the total temperature is uniform 300 K throughout the nozzle. The contour abnormality at the outlet of the nozzle is due to the round off errors.

Pressure Plot

Let's look at the pressure along the centerline and the wall.



Make sure that under Y-Axis Function, you see Pressure... and Static Pressure. Under Surfaces, select centerline and wall. Click Plot.



Higher Resolution Image

It is good to write the data into a file to have greater flexibility on how to present the result in the report. At the same XY Plot windows, select *Write to File*. Then click *Write…* Name the file "p.xy" in the directory that you prefer.

Open "p.xy" file with notepad or other word processing software. At the top, we see:

```
(title "Static Pressure")
(labels "Position" "Static Pressure")
```

First line tells us the properties we are comparing. For our case, we are looking at Static Pressure. Second line tells us about the x and y label.

There is a header at the beginning of each the data sets so that we can differentiate which data sets we are looking at. For our case, we have "centerline" and "wall" data sets.

Following is an example of two data sets (centerline and wall).

```
((xy/key/label "centerline")
-0.5
        97015.3
-0.48
       96949.9
.
.
.
0.5
       6012.92
)
((xy/key/label "wall")
-0.5
      100853
-0.480911
             100496
.
.
.
0.5
       2874.7
)
```

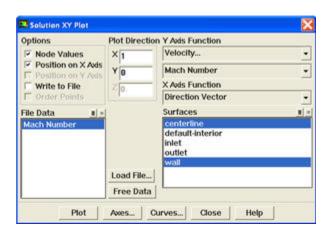
Try copy the appropriate data sets to excel and plot the results.

Mach Number Plot

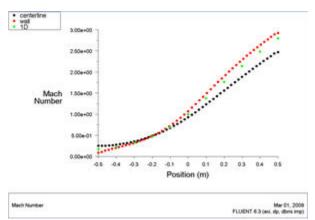
Let's plot the variation of Mach number in the axial direction at the axis and wall. In addition, we will plot the corresponding variation from 1D theory. You can download the file here: mach_1D.xy.*Plot* > XY *Plot*Under the Y *Axis Function*, select *Velocity...* and *Mach Number*.

Also, since we are going to plot this number at both the wall and axis, select centerline and wall under Surfaces.

Then, load the mach_1D.xy by clicking on Load File....



Click Plot.



Higher Resolution Image

How does the FLUENT solution compare with the 1D solution?

Is the comparison better at the wall or at the axis? Can you explain this?

Save this plot as machplot.xy by checking *Write to File* and clicking *Write...*.

Go to Step 7: Verification & Validation

See and rate the complete Learning Module

Go to all FLUENT Learning Modules