FLUENT - Compressible Flow in a Nozzle- Step 6 *New

FLUENT - Compressible Flow in a Nozzle- Step 6

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

```
Problem Specification

1. Pre-Analysis & Start-up

2. Geometry

3. Mesh

4. Setup (Physics)

5. Solution

6. Results

7. Verification & Validation

Problem 1

Problem 2
```

Step 6: Results

(i) Useful Information

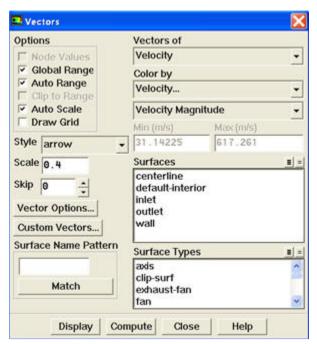
These instructions are for FLUENT 12. Click here for instructions for FLUENT 6.3.26.

Velocity Vector

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

Display > Graphics and Animations > Vectors > Set up...

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



Click *Display*. step6_2.png

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

0

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?"

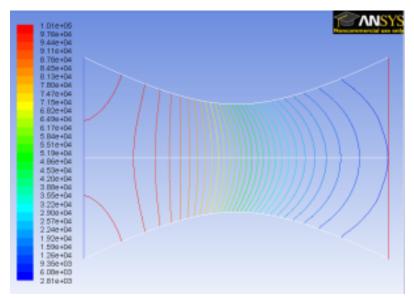
Pressure Contour Plot

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

Display > Graphics and Animations > Contours > Set up

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

Contours		×
Options	Contours of	
Filled	Pressure	~
✓ Node Values ✓ Global Range	Static Pressure	~
Auto Range	Min Max	
Clip to Range	0 0	
Draw Mesh	Surfaces	
Levels Setup 30 I I	centerline default-interior inlet	
Surface Name Pattern	New Surface 🔻	
Match		Ð
	axis	^
	clip-surf exhaust-fan	
	fan	×
Display	Compute Close Help	



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

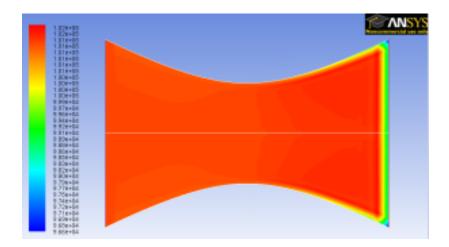
Total Pressure Contour Plot

Let's look at the total pressure in the nozzle

Display > Graphics and Animations > Contours

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

Contours		×
Options	Contours of	
Filled	Pressure	~
✓ Node Values ✓ Global Range	Total Pressure	*
Auto Range	Min (pascal) Max (pascal)	
Clip to Range	96614.34 101779	
Draw Mesh	Surfaces	⊟
Levels Setup	centerline default-interior inlet outlet wall	
Surface Name Pattern	New Surface 🕶	
Match	Surface Types	⊟
Display	Compute Close Help	



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

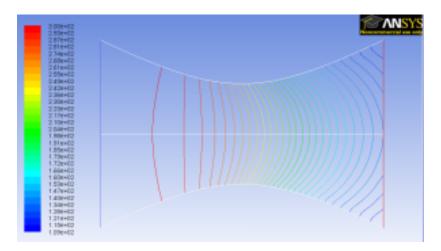
Temperature Contour Plot

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

Display > Graphics and Animations > Contours

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

Contours		×
Options	Contours of	
Filled	Temperature	*
✓ Node Values ✓ Global Range	Static Temperature	~
Auto Range	Min Max	
Clip to Range	0 0	
Draw Mesh	Surfaces	
Levels Setup	centerline default-interior inlet outlet wall	
Surface Name Pattern	New Surface Surface Surface Types	
	axis clip-surf exhaust-fan fan	
Display	Compute Close Help	



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

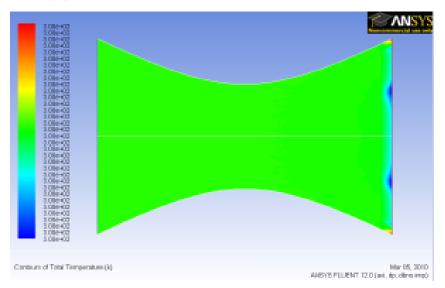
Now let's look at the total pressure in the nozzle

Display > Graphics and Animations > Contours

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

Options	Contours of		
Filled	Temperature		Ĩ
Node Values Global Range	Total Temper	ature	
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	
Match	Surface Type	s	
	axis clip-surf exhaust-fan fan		< >

Click Display.



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.

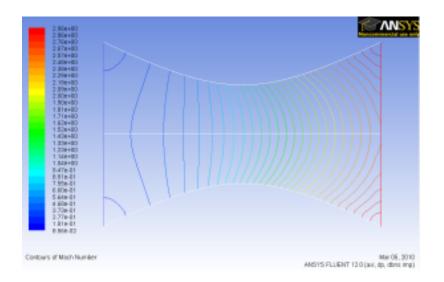
Mach Number Contour

Let's now look at the Mach number

Display > Contours

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

Contours		
Options ✓ Filled ✓ Node Values ✓ Global Range ✓ Auto Range Clip to Range Draw Profiles Draw Mesh	Contours of Velocity Mach Number Min Max 0 Surfaces	• •
Levels Setup	 centerline default-interior inlet outlet wall 	
Surface Name Pattern	ch Surface ▼ Axis clip-surf exhaust-fan fan	
Display	Compute Close Help	



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 Plot

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

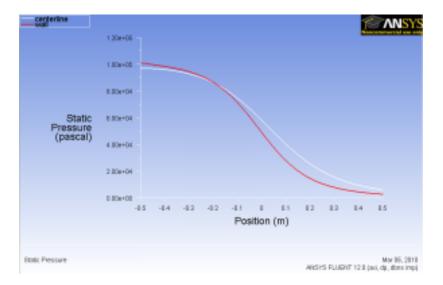
Solution XY Plot			×
Options Vode Values Position on X Axis Position on Y Axis Vrite to File Order Points File Data	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Pressure Static Pressure X Axis Function Direction Vector Surfaces Centerline default-interior inlet outlet wall New Surface ▼	
Plot	Axes	Curves Close Help	



Make sure that under *Y-Axis Function*, you see *Pressure...*and *Static Pressure*. Under *Surfaces*, select *centerline* and *wall*. Click *Plot*. By default FLUENT displays the plots using graphics but we will modify them to display lines which can be more helpful when analyzing the plot.

Curves - Solution	on XY Plot	×
Curve # 1 • Pattern Sample Color red Weight 2	ו ייער איז	Marker Style Symbol Color red Size 0.3
	Apply Close	Help

On the XY Plot window click on the *Curves* button. Change the *Pattern* to a line, Select a blank as a *Symbol* and change the *Weight* to any number greater than 2. To do this for multiple curves use the arrow buttons to navigate through the different curves displayed on the plot.



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")

The first line tells us the properties we are comparing. In this case we are looking at Static Pressure. The second line tells us about the labels for the x and y axes, respectively.

There is a header at the beginning of each the data sets so that we can differentiate which data sets we are looking at. In this 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

```
)
((xy/key/label "wall")
-0.5 100853
-0.480911 100496
0.5 2874.7
)
```

Try copying the appropriate data sets to excel and plotting 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. This file is a text file. You can look at the format of this file by reading it into Wordpad.

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.

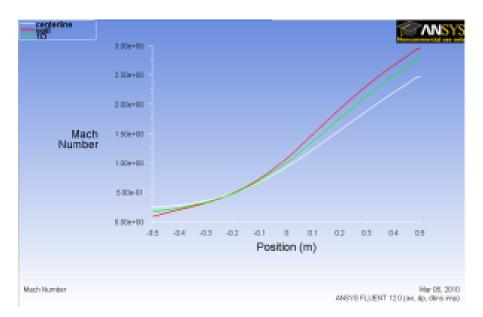
Then, load the mach_1D.xyby clicking on Load File....

Options	Plot Directio	n Y Axis Function
Voide Values Position on X Ads Position on Y Ads Write to File	X 1	Velocity
	Ye	Mach Number 👻
	2 0	X Axis Function
		Direction Vector +
File Data 🔢 -	1	Surfaces 🔳 =
Mach Number	Load File	centerline default-interior inlet outlet wall

Click Plot.



The green line in the plot below SHOULD be in symbols. This will be corrected soon.



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