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
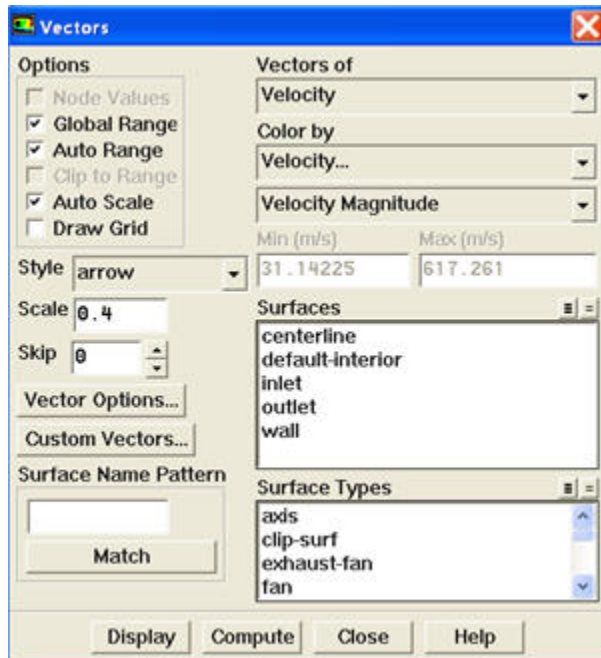
Step 6: Results

> (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.

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

(i)

> ⓘ **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.
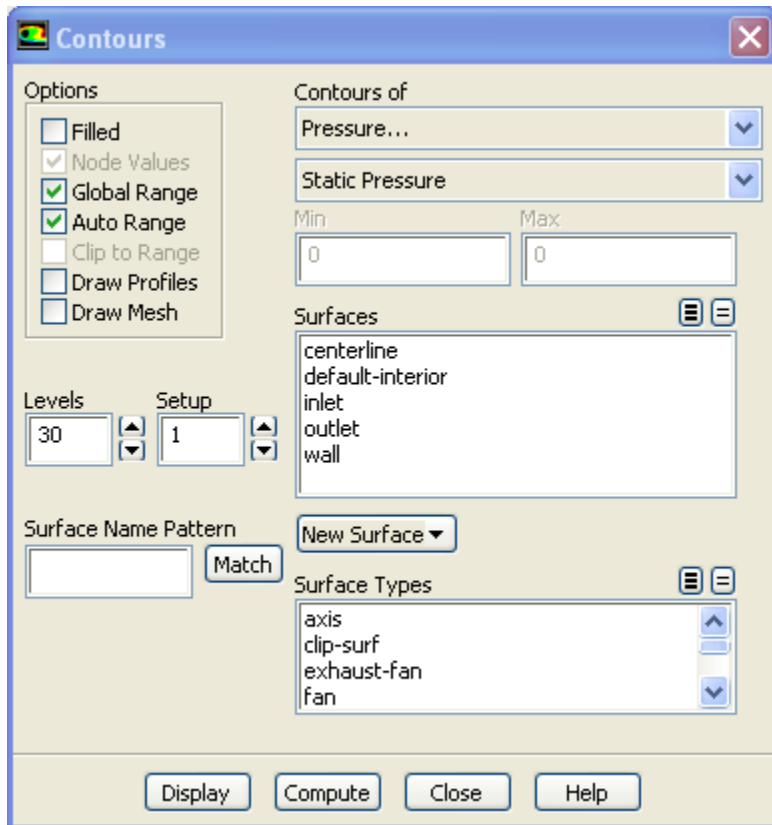>
> Click *Preview*. Click *No* when prompted "*Reset graphics window?*"
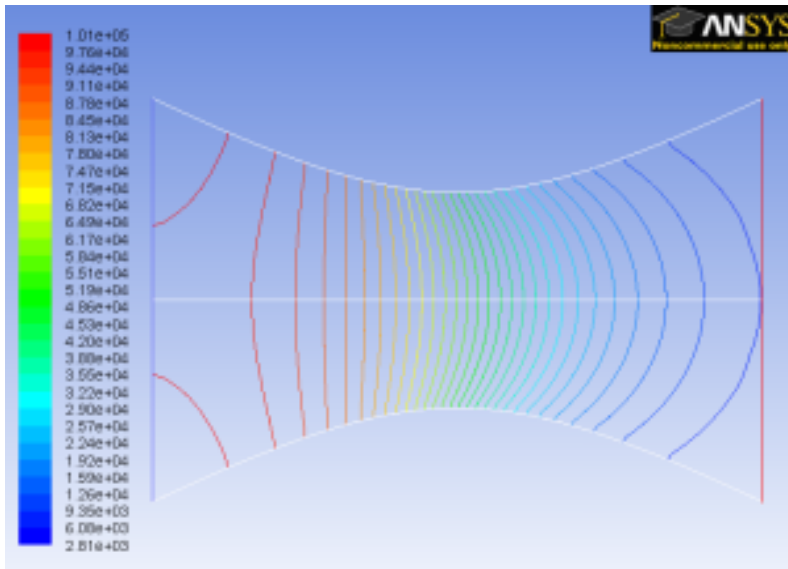
Pressure Contour Plot

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
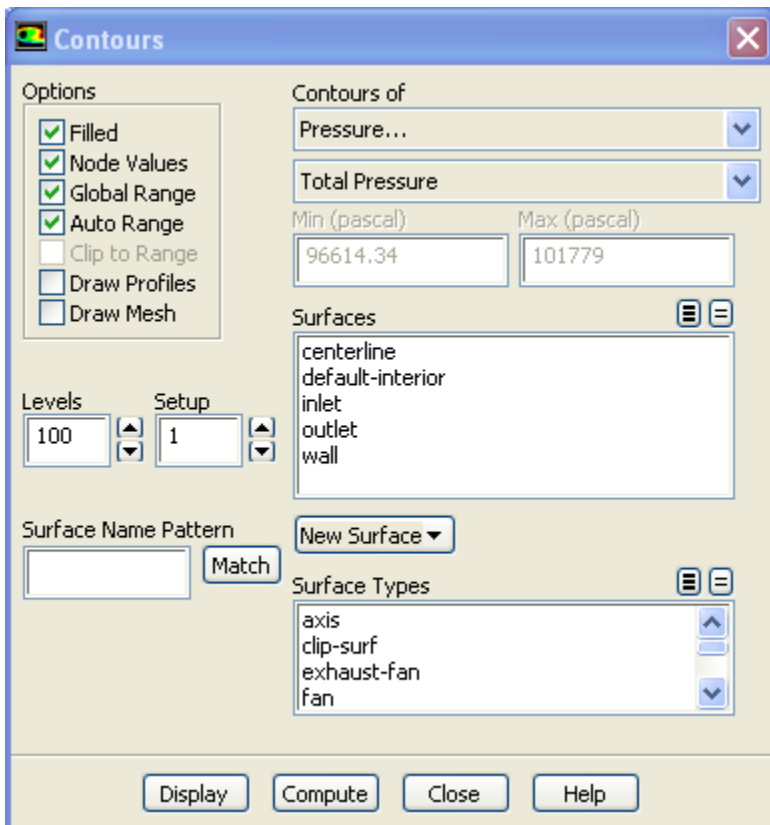

Click **Display**.

Notice that the pressure decreases as it flows to the right.
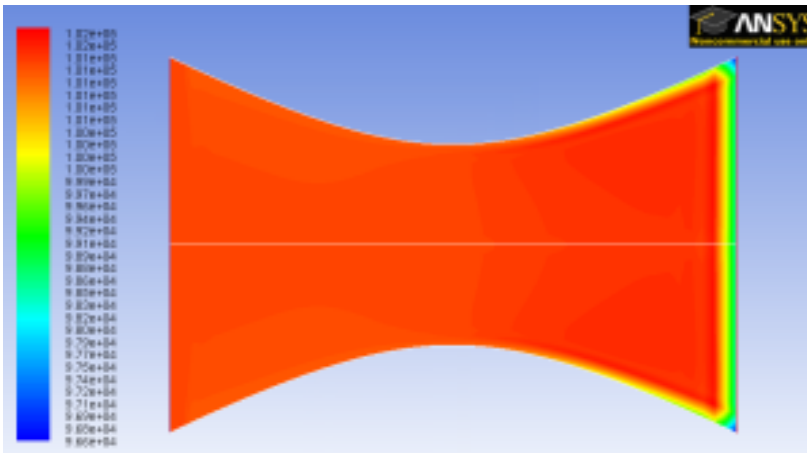
Total Pressure Contour Plot

Let's look at the total pressure in the nozzle

Display > Graphics and Animations > Contours

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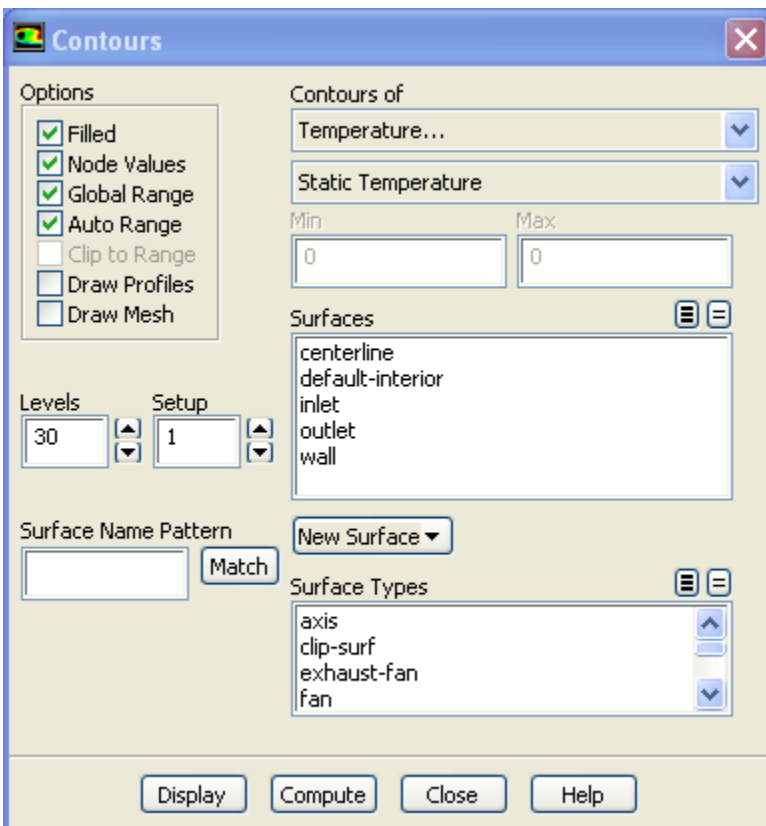

Click *Display*.

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

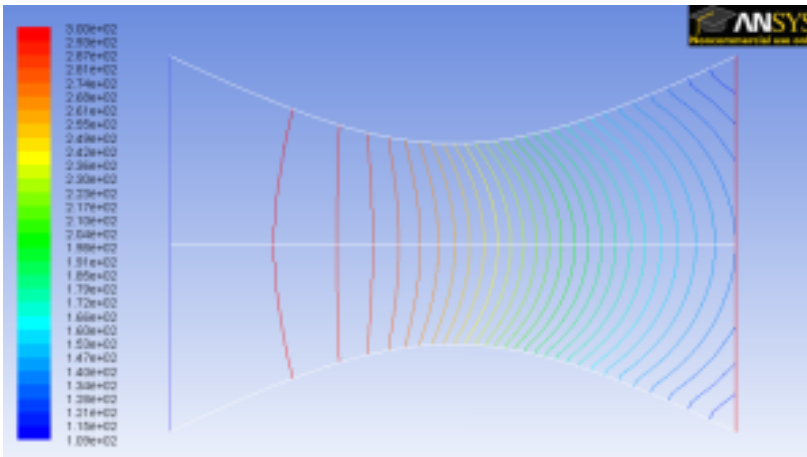**Temperature Contour Plot**

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.
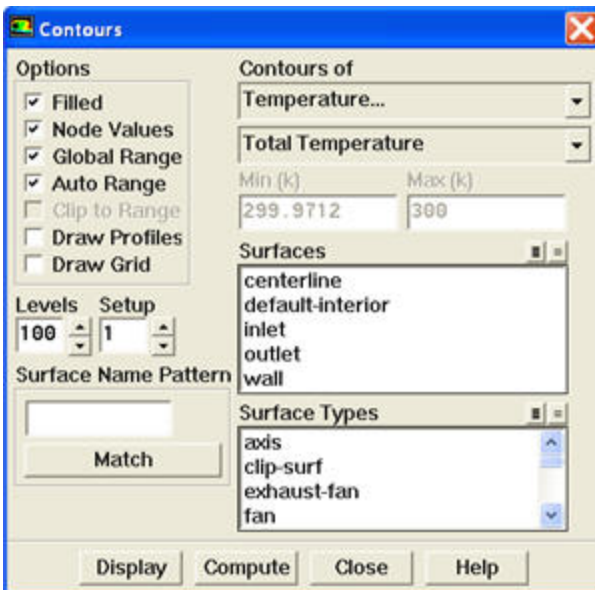

Click *Display*.

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.
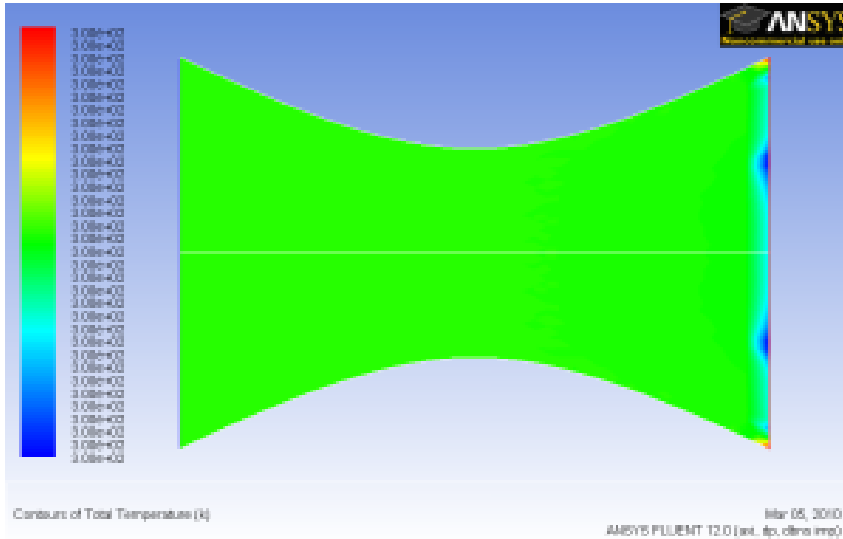
Total Temperature Contour Plot

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.

Click *Display*.

Contours of Total Temperature (k)

Mar 05, 2010
ANSYS FLUENT 12.0 (axi, dp, dens-imp)

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.
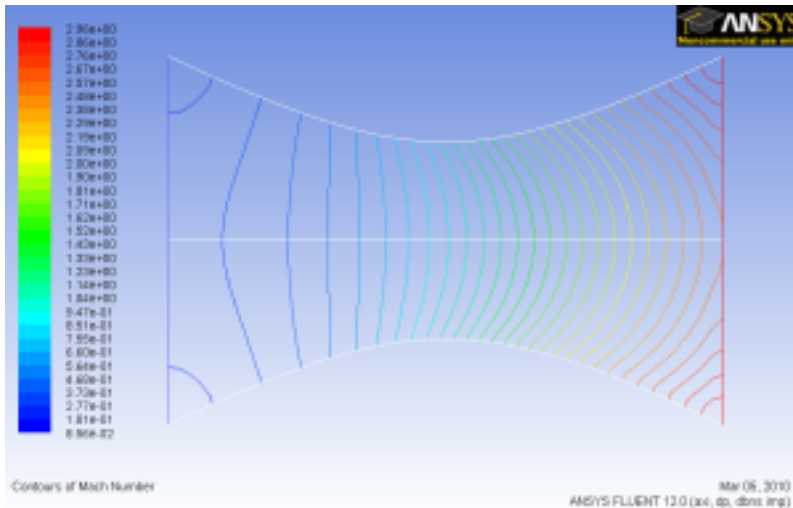
Mach Number Contour

Let's now look at the Mach number

Display > Contours

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

Click *Display*.

Contours of Mach Number

Mar 05, 2010
ANSYS FLUENT 12.0 (axi, dp, dbns imp)
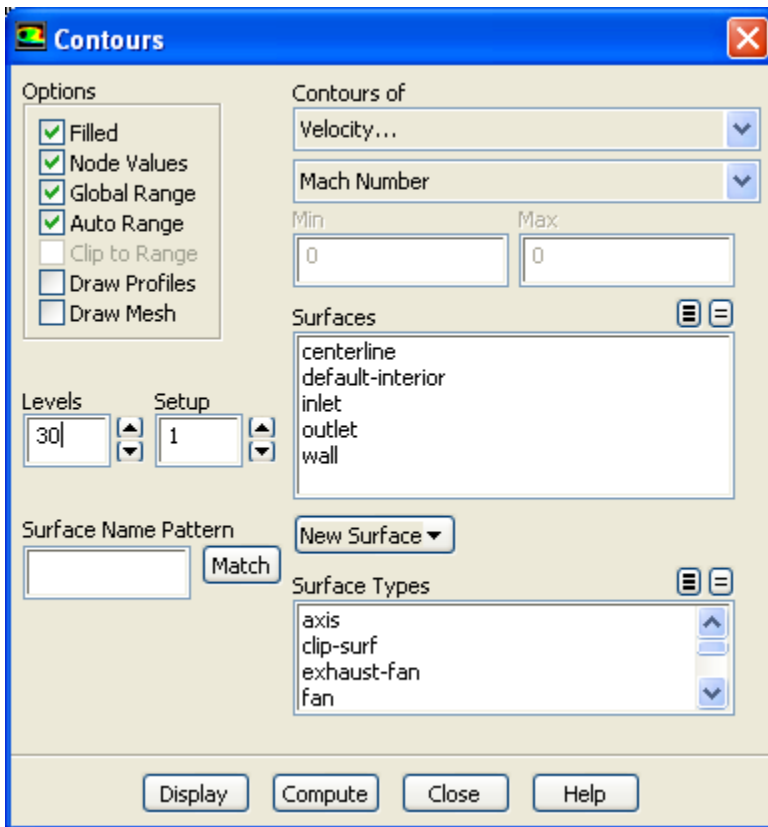
For 1D case, mach number is a function of x position. For 1D case, we are supposed to see vertical contour of mach numbers that are parallel to each other.

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.
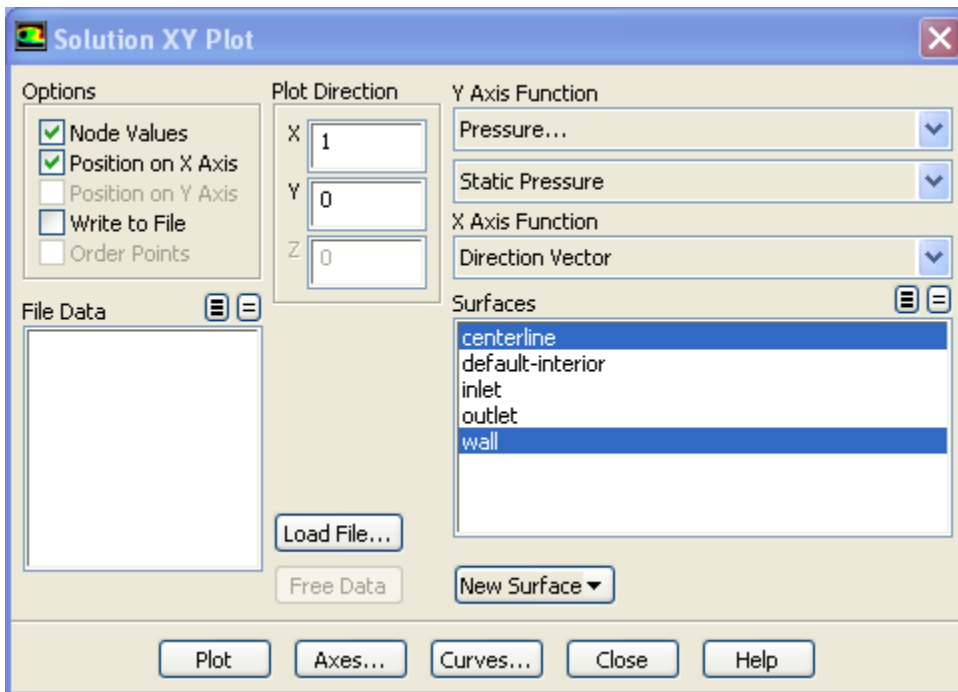
*Under Results > Plots > XY Plot > Set up*

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.
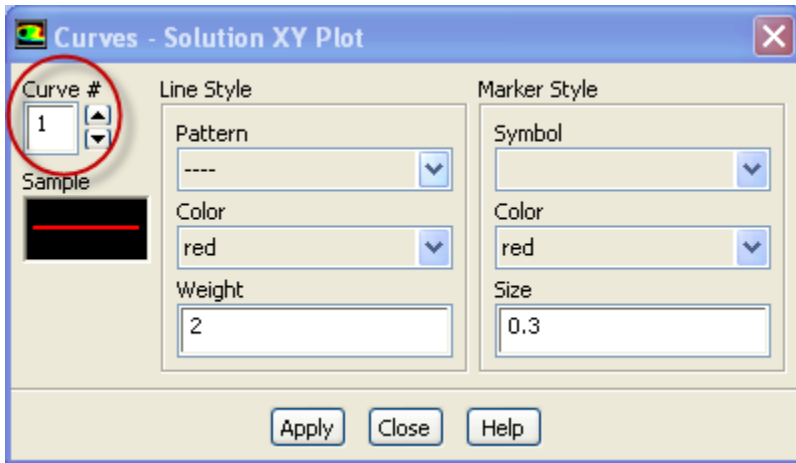
On the XY Plot window click on the **Curves** button. Change the **Pattern** to a line, Select a blank as a **Symbol** and change the **Weight** to any number greater than 2. To do this for multiple curves use the arrow buttons to navigate through the different curves displayed on the plot.
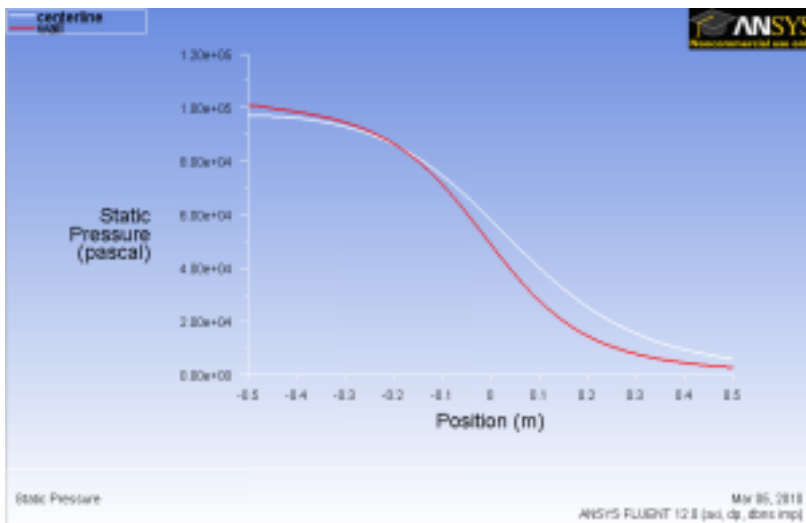
It is good to write the data into a file to have greater flexibility on how to present the result in the report. At the same XY Plot windows, select **Write to File**. Then click **Write...** Name the file "p.xy" in the directory that you prefer.

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

.

.

.

0.5   6012.92

)

((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***.
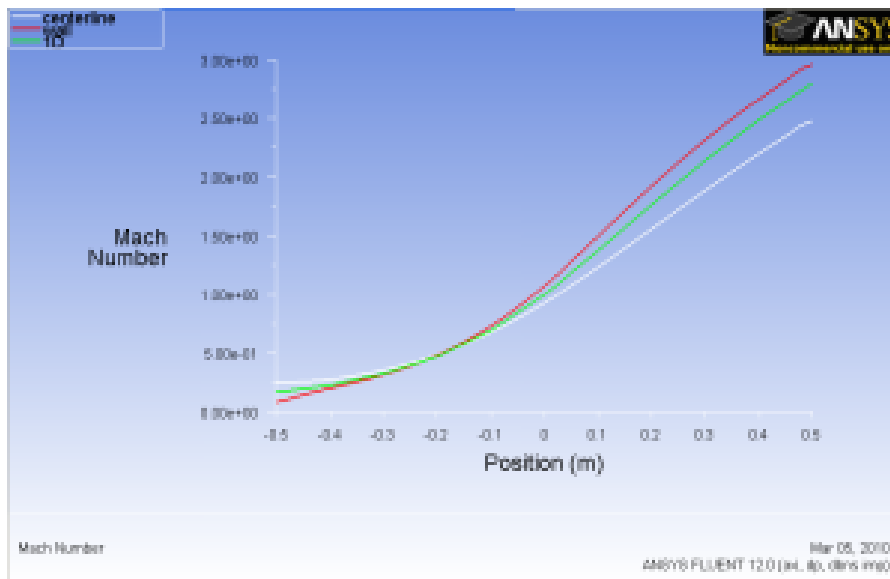
Then, load the mach_1D.xyby clicking on ***Load File....***
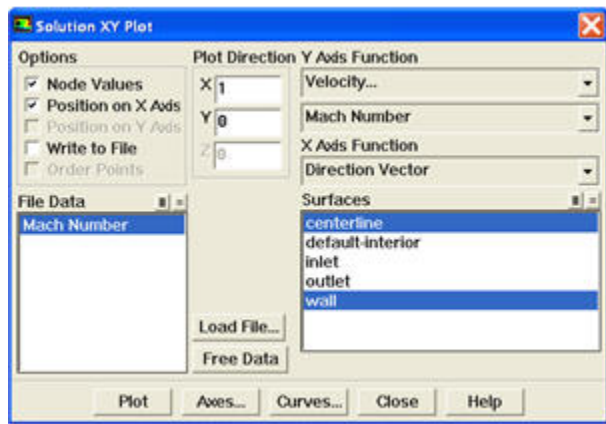
Click ***Plot***.

> ⚠ **Correction**
>
> The green line in the plot below SHOULD be in symbols. This will be corrected soon.

Mach Number

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