

FLUENT - Supersonic Flow Over a Wedge- Step 6 *New

Problem Specification

1. Pre-Analysis & Set-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solutions
- 6. Results**
7. Verification & Validation



Useful Information

[Click Here](#) for the FLUENT 6.3.26 version.

Step 6: Analyze Results

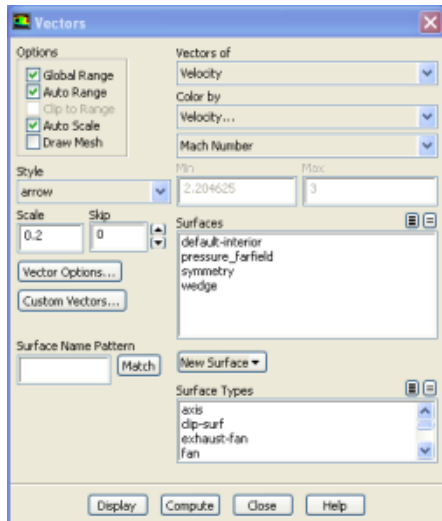
Plot Velocity Vectors

Let's plot the velocity vectors obtained from the FLUENT solution.

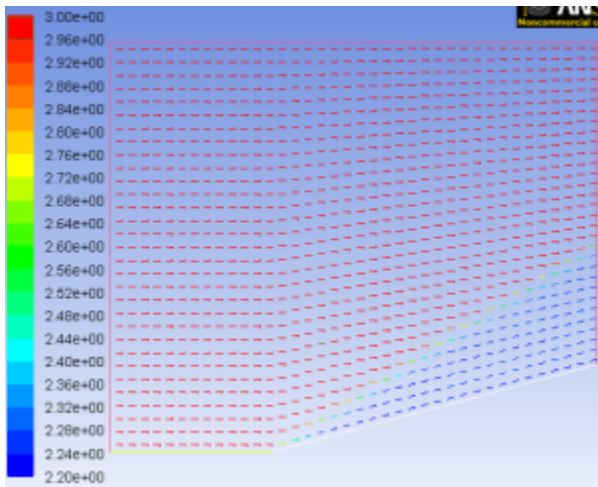
Display > Graphics and Animations or **Results > Graphics and Animations**

Select **Vectors**, click on the **Set up** button. Under **Color by**, select **Mach Number** in place of **Velocity Magnitude** since the former is of greater interest in compressible flow. The colors of the velocity vectors will indicate the Mach number. Use the default settings by clicking **Display**.

This draws an arrow at the center of each cell. The direction of the arrow indicates the velocity direction and the magnitude is proportional to the velocity magnitude (not Mach number, despite the previous setting). The color indicates the corresponding Mach number value. The arrows show up a little more clearly if we reduce their lengths. Change Scale to 0.2. Click **Display**.



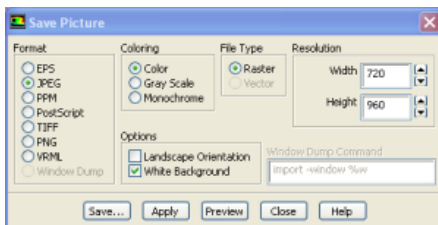
Zoom in a little using the middle mouse button to peer more closely at the velocity vectors.



We can see the flow turning through an oblique shock wave as expected. Behind the shock, the flow is parallel to the wedge and the Mach number is 2.2. Save this figure to a file:

Main Menu > File > Hardcopy

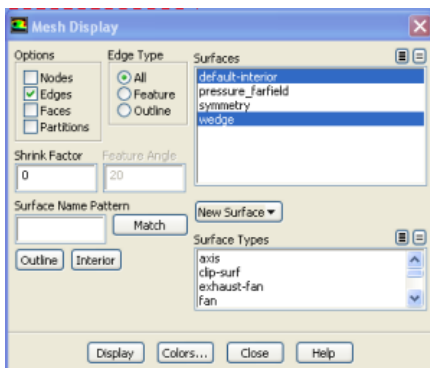
Select **JPEG** and **Color**. Uncheck **Landscape Orientation**. Save the file as wedge_vv.jpg in your working directory. Check this image by opening this file in an image viewer.



Let's investigate how many mesh cells it takes for the flow to turn. To turn on the mesh do the following:

Display > Graphics and Animations or Results > Graphics and Animations

Select **Vectors**, click on the **Set up** button. Under options select the **Display Mesh** checkbox and a new window will pop up. Under Surfaces select **default-interior**, click **Display** and close it.



The mesh will substitute the arrows and to display both the mesh and the arrows go back to the Vector Set up window click **Display** again and now the mesh and arrows will appear on the same picture.

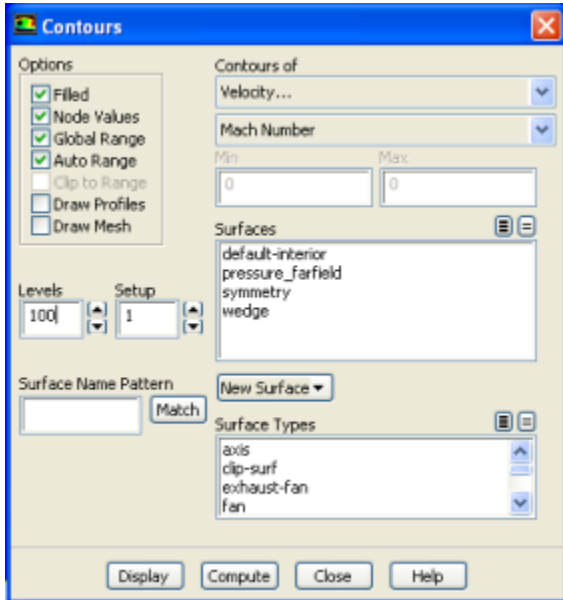
We see that it takes 2-3 mesh cells for the flow to turn. According to inviscid theory, the shock is a discontinuity and the flow should turn instantly. In the FLUENT results, the shock is "smeared" over 2-3 cells. In the discrete equations that FLUENT solves, there are terms that act like viscosity. This introduced viscosity contributes to the smearing. A more thorough explanation would have to go into the details of the numerical solution procedure.

Plot Mach Number Contours

Let's take a look at the Mach number variation in the flowfield.

Display > Graphics and Animations or Results > Graphics and Animations

Select **Contours**, click on the **Set up** button. Under **Contours of**, choose **Velocity..** and **Mach Number**. Select the **Filled** option. Increase the number of contour levels plotted: set **Level** to 100.



Click **Display**.

We see that the Mach number behind the shockwave is uniform and equal to 2.2. Compare this to the corresponding analytical result.

Plot Pressure Coefficient Contours

Pressure Coefficient is a dimensionless parameter defined by the equation

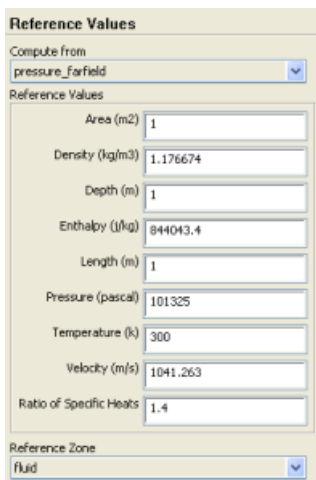
$$C_p = \frac{(p - p_{ref})}{q_{ref}}$$

where p is the static pressure, p_{ref} is the reference pressure, and q_{ref} is the reference dynamic pressure defined by $q_{ref} = \frac{1}{2} \rho_{ref} v_{ref}^2$

The appropriate reference values for this problem are the freestream values. Let's set the reference values necessary to calculate the pressure coefficient.

Report > Reference Values

Select **pressure-farfield** under **Compute From**.



This fills in the values from the farfield boundary into this panel. These reference values of density, velocity and pressure will be used to calculate the pressure coefficient from the pressure.

Display > Graphics and Animations or Results > Graphics and Animations

Select **Contours**, click on the **Set up** button. Select **Pressure...** and Static Pressure from under **Contours Of**. Then select **Pressure Coefficient**.

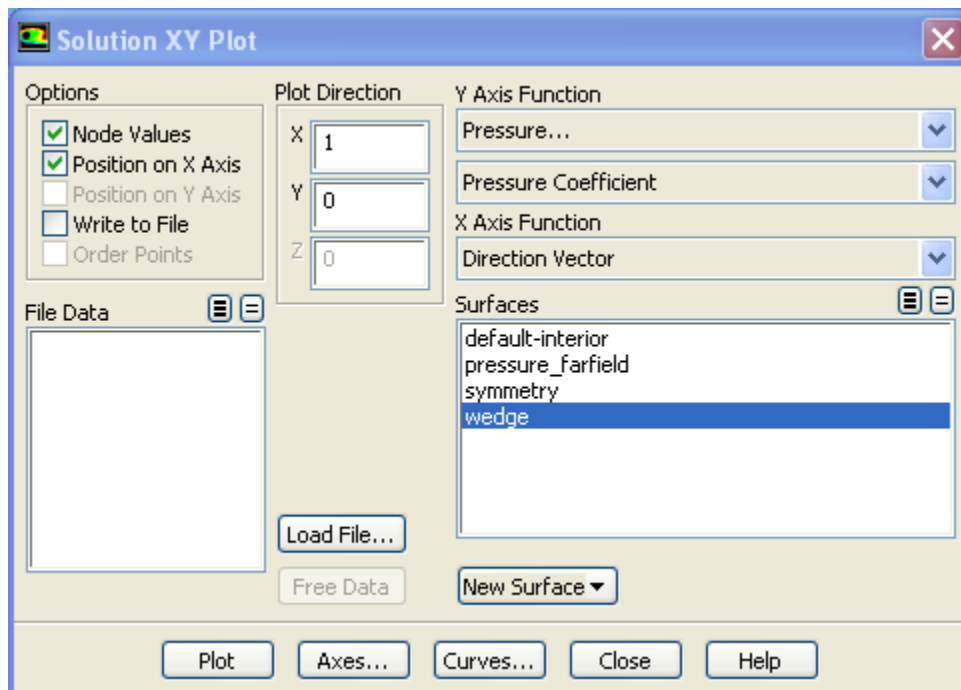
The pressure coefficient after the shockwave is 0.293, very close to the theoretical value of 0.289. The pressure increases after the shockwave as we would expect.

Pressure Coefficient Along Wedge

Let's plot the pressure coefficient along the wedge. Go to:

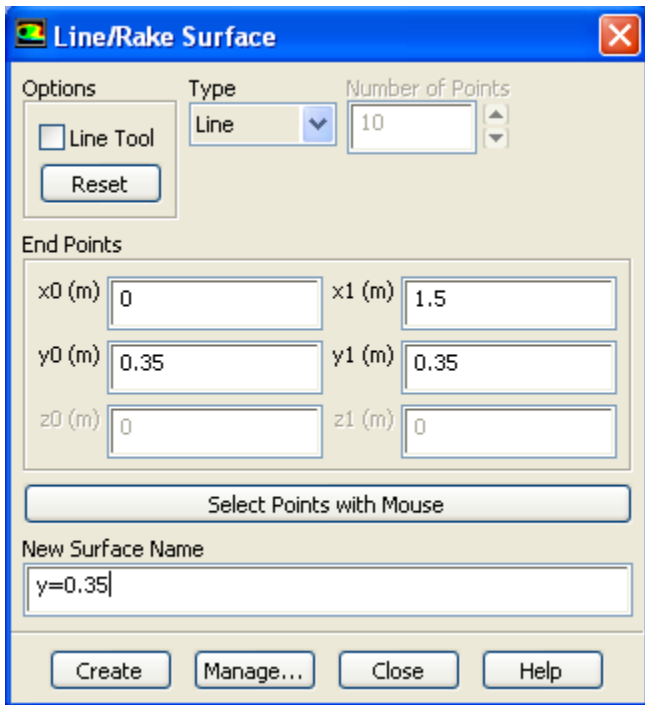
Results (left-hand side of the screen) > Plots

Select **XY Plot** and then click on the **Set up** button. In the pop-up window specify the **Y Axis Function** to **pressure** and **pressure coefficient**. Select the **wedge** from the available surfaces and click **Plot**. Your specifications should look like the picture below.



In order to get the values to compare with the analytical solution, it is better to write this data to a file rather than try to read it off the plot. You can save the data used in this plot to a file by checking the **Write to File** checkbox under **Options**. Then click the **Write...** button, give the file a name and click **Function**.

To measure the Shock Angle we can add a new line to the plot corresponding to the pressure coefficient at, say, $y=0.35\text{m}$. Click on **New Surface** and select the **Line/Rake** option. Set the first point to (0,0.35) and the second one to (1.5,0.35) name the line $y=0.35$ and click on **Create**.



The new line will be added to the available surfaces. From these select **Symmetry, Wedge and y=0.35**. You will notice that there is no **Plot** button, this happened because the **Write to file** checkbox under options is still checked. Uncheck it and then click on **Plot**. You will now be able to see the plot. You should be able to calculate the x-location of the shock at y=0.35 from this plot and from that back out the shock angle approximately.

Total Pressure

Next, we will plot the Total Pressure. On the Solution XY Plot pop-up window, select **Total Pressure** under **Y Axis Function** and click **Plot**.

Drag Coefficient

Finally, we will find out what the drag coefficient is. FLUENT will integrate the pressure over the wedge to find the net force on the wedge. We'll ask it to take the component in the x-direction to get the drag. This needs to be non-dimensionalized using the appropriate reference values, so we have to set these.

Close the pop-up window if it's still open and go to

Problem Setup > Reference Values

The reference values of density and velocity were already specified above. Make sure that the entry for Area is 1. FLUENT uses this as the length scale in calculating the drag coefficient (it doesn't make a distinction between 2D and 3D ... a bit sloppy IMHO). Then go to

Results > Reports > Forces > Set up... > Print

The drag coefficient should be the value listed in the last column of the values in the output window. (The analytical procedure to calculate the drag coefficient is given in Anderson, example 4.9).

Go to Step 7. [Verification](#)

[See and rate the complete Learning Module](#)

Go to [all FLUENT Learning Modules](#)