Supersonic Flow Over a Wedge - Numerical Results (FPP)

Author: Rajesh Bhaskaran, Cornell University

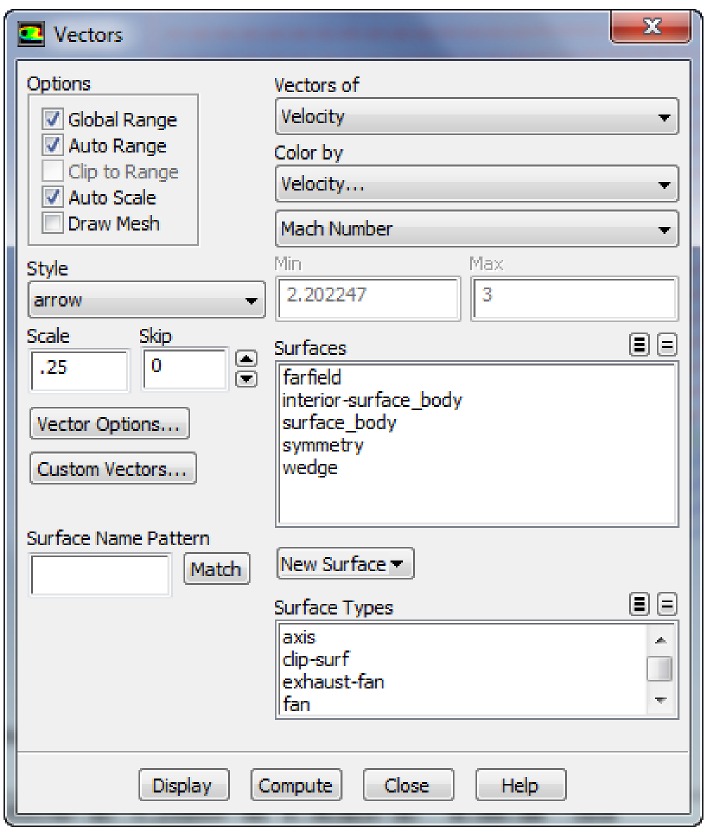
This page goes through the post-processing steps for the Supersonic Flow Over a Wedge in the classic FLUENT post-processor. For instructions on performing the post-processing using the newer CFD Post post-processor included in ANSYS Workbench, see this page.

Numerical Results (in FLUENT Post-Processor)

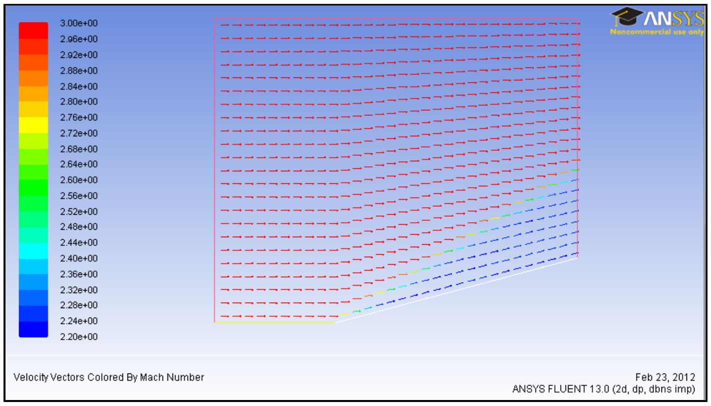
Velocity Vectors

First, let's take a look at the velocity vectors. In the *Outline* window, under ***Results*** select ***Colors and Animations***. In the *Colors and Animations* window, under *Graphics*, select ***Vectors***. Then press ***Set Up...***.

In the *Vectors* window that opens, change the ***Scale*** of the arrows to ***0.25***, and change the ***Color by*** parameter to ***Velocity... Mach Number***



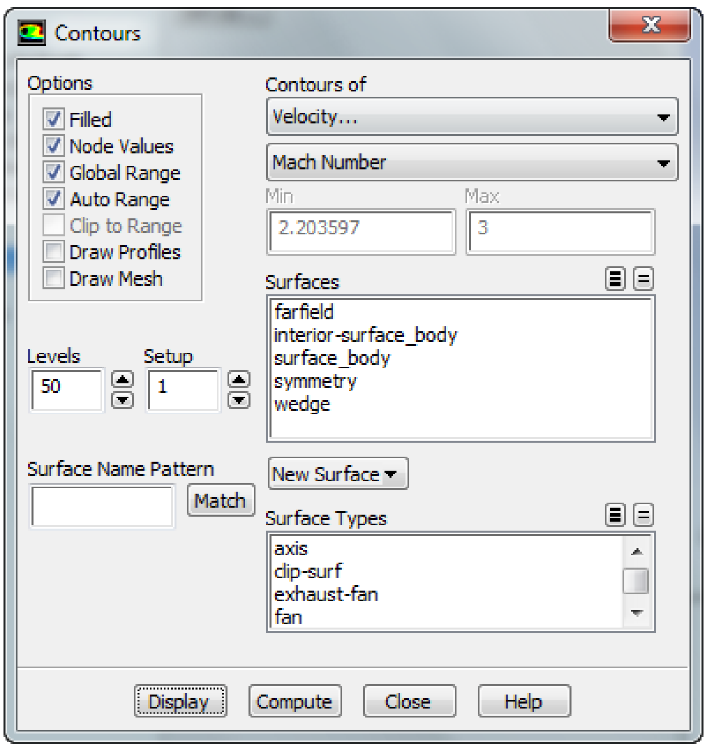
Hit ***Display*** to view the vectors in the graphics window



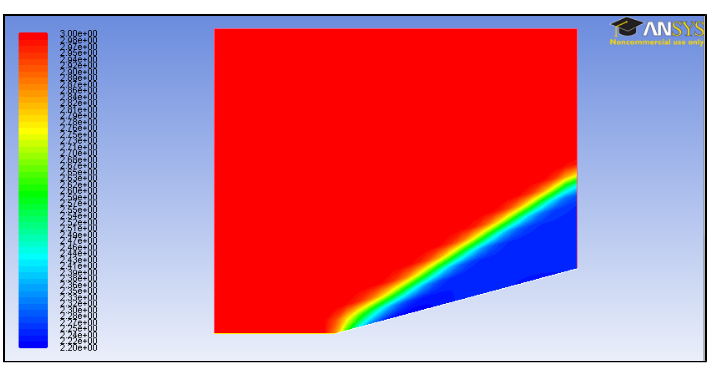
Once you have plotted the velocity vectors, you may close the *Vectors* menu.

Mach Number Contours

Next, we will plot the contours of mach number. In the *Graphics and Animations* window under _Graphics, select ***Contours***, and press ***Set Up...***. Change the ***Contours Of*** parameter to ***Velocity... Mach Number***. Next, check the box next to ***Filled***.



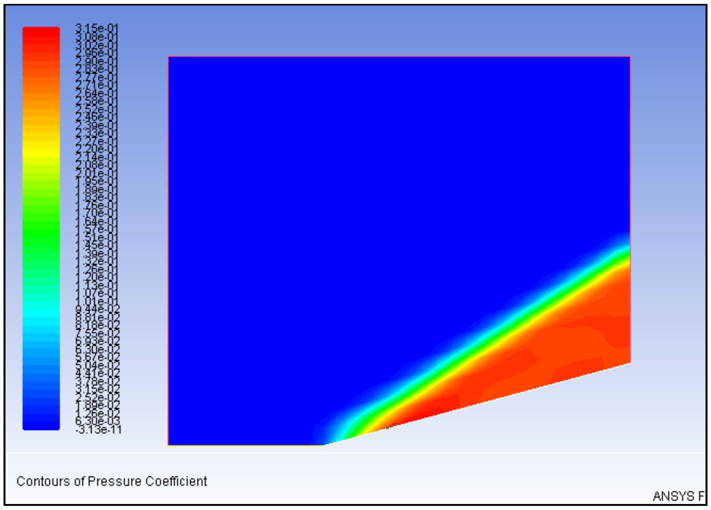
Change the ***Levels*** to ***50*** and press ***Display***



Next, click on print information icon , and click anywhere behind the shock. This will display a range of values. For instance, I received (2.2354529, 2.2513809). These range of values match well with the analytic solution we received in the pre-anaysis (Mach no. = 2.254, 0.8% difference)

Pressure Coefficient/ Static Pressure Contours

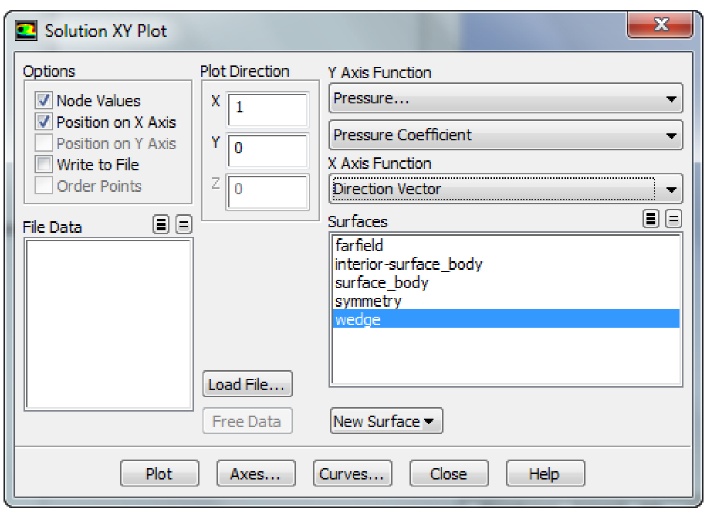
Now, change the ***Contours Of*** parameter to ***Pressure... Pressure Coefficient*** and press ***Display***.



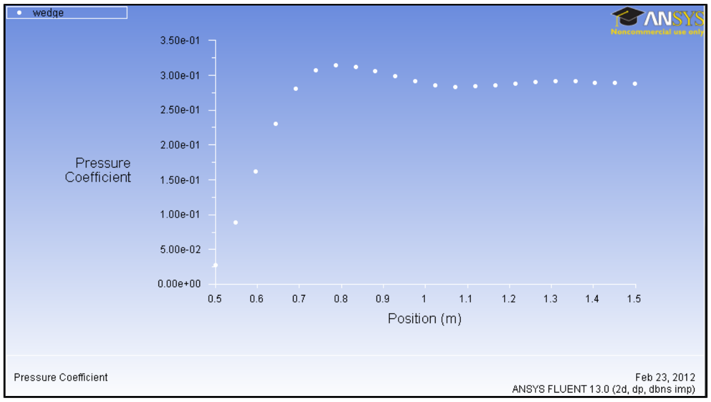
Next, change the ***Contours Of*** parameter to ***Pressure... Static Pressure*** and press ***Display***. When you have plotted the contours of static pressure, you may close the *Contours* window. Again, use the ***print information*** tool to view the pressure behind the shock. I received the range (282041.19, 286057.13). The average of this range is about 2.803 atmospheres. From the analytical solution, we found that the pressure behind the shock is about 2.824, a 0.7% difference.

Pressure Coefficient Along Wedge

In the *Outline* window under *Results*, select ***Plots***. In the *Plots* window, select ***XY plot*** and press ***Set Up...***. Change the ***Y Axis Function*** to ***Pressure... Pressure Coefficient***, and select the ***Wedge*** under ***Surfaces***.

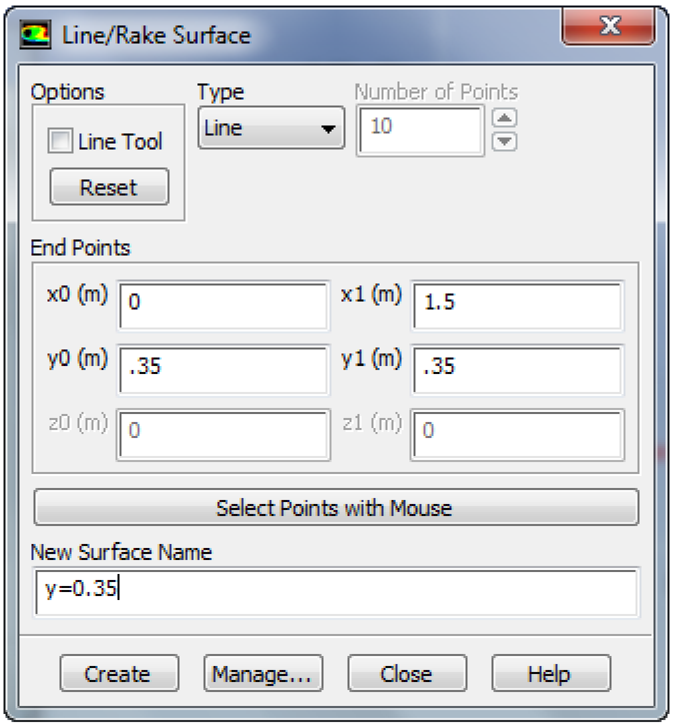


To see the plot, press ***Plot***

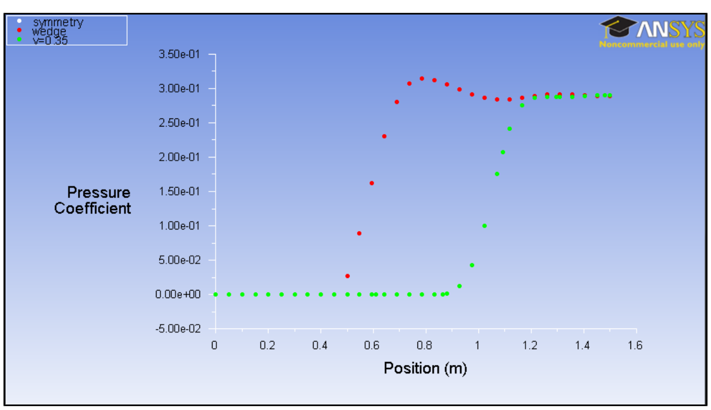


Measure Shock Angle

To measure the shock angle, we first need to create a line that we can plot the coefficient of pressure over. In the *Solution XY Plot* window, click ***New Surface > Line/Rake***. We will arbitrarily use the line, y = 0.35. Set the start point of the line as ***(0,0.35)***, and the end point of the line as ***(1.5,0.35)***. Name the line ***`y = 0.35`***. Now press ***Create***



Now, the new surface will appear in the list of surfaces. We want to plot the pressure coefficient across the wedge and the new line we created in order to determine the shock angle. In the list of surfaces, select ***wedge, symmetry,*** and ***y = 0.35***. Press ***Plot***.



We can then use the information from the plot to solve for the shock angle.

$$\theta_{shock} = \tan^{-1}\left(\frac{\Delta y}{\Delta x}\right) = \tan^{-1}\left(\frac{.35 - 0}{.97 - .47}\right) = 34.99^{0}$$

The shock angle from the analytical solution was 32.221 degrees. Because we took the value from a graph, a bit more error is to be expected.

Print Drag Coefficient

Next, we will have FLUENT report the drag coefficient. Close the *Solution XY Plot* window. In the *Outline* window, go to ***Reports***. In the *Reports* window under *Reports*, select ***Forces*** and press ***Set Up...***. Ensure ***Wedge*** is selected and press ***Print***. The drag coefficient will be printed.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules