FLUENT - Supersonic Flow Over a Wedge- Step 6

Site Under Construction

Please bare with us as we update this site to include instructions for the newest version of FLUENT.

Step 6: Analyze Results

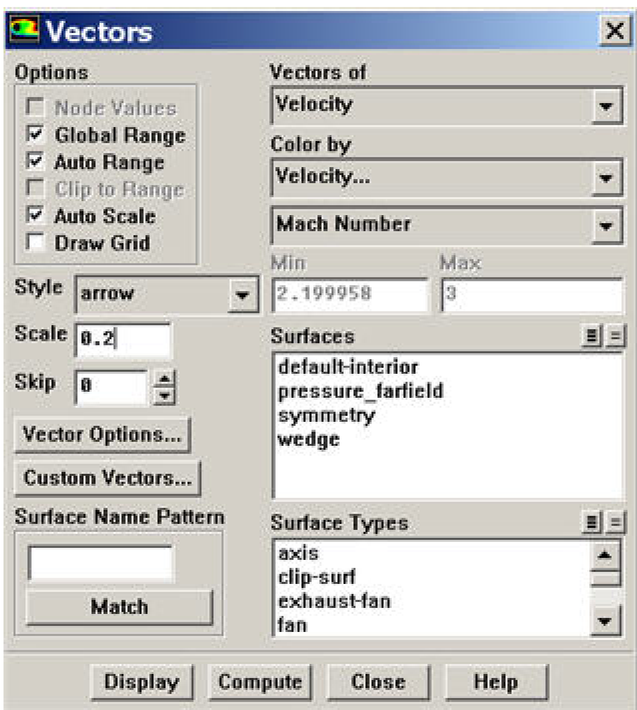
Plot Velocity Vectors

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

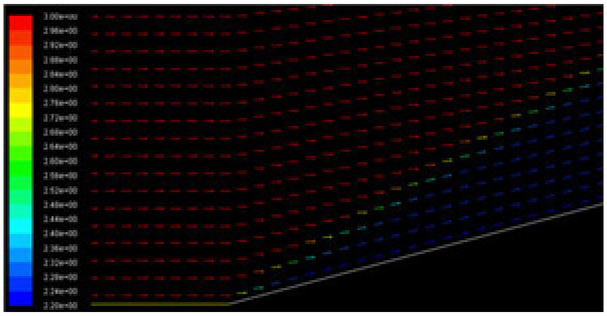
Display > Vectors

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.



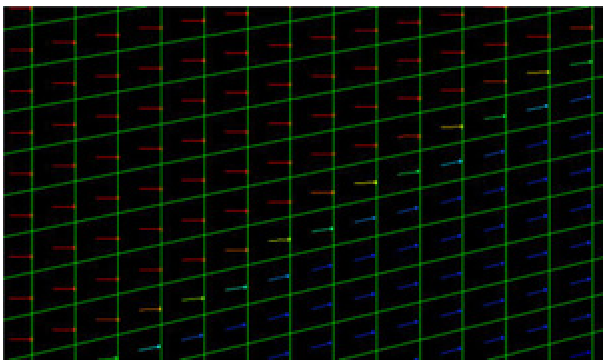
Higher Resolution Image

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 iimage by opening this file in an image viewer.

Let's investigate how many mesh cells it takes for the flow to turn. Turn on the mesh by clicking on the ***Draw Grid*** checkbox in the *Vectors* menu. In the *Gri d Display* menu that pops up, click ***Display***. This displays the mesh in the graphics window. ***Close*** the *Grid Display* menu. Click ***Display*** in the *Vectors* menu. Zoom in further as shown below.



Higher Resolution Image

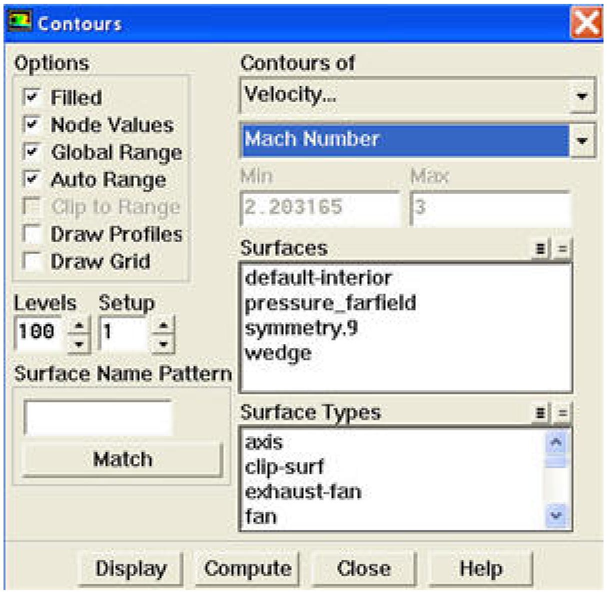
We see that it takes 2-3 mesh cells for the flow to turn. According to inviscid theory, the shock is a discontinuity and the flow should turn instantly. In the FLUENT results, the shock is "smeared" over 2-3 cells. In the discrete equations that FLUENT solves, there are terms that act like viscosity. 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

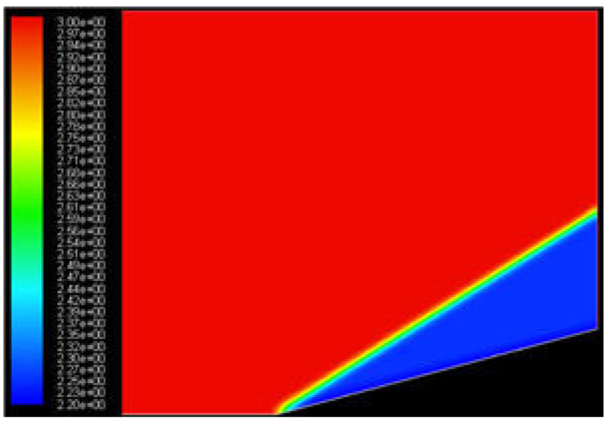
Let's take a look at the Mach number variation in the flowfield.

Display > Contours

Under ***Contours of***, choose ***Velocity..*** and ***Mach Number***. Select the ***Filled*** option. Increase the number of contour levels plotted: set ***Levels*** to `100`.



Click ***Display***.



Higher Resolution Image

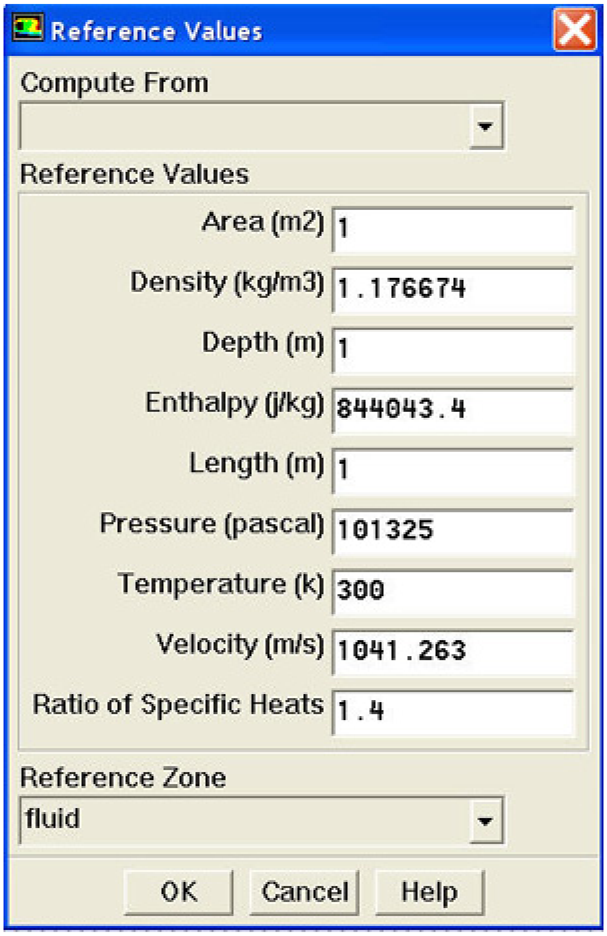
We see that the Mach number behind the shockwave is uniform and equal to 2.2. Compare this to the corresponding analytical result.

Plot Pressure Coefficient Contours

Let's set the reference values necessary to calculate the pressure coefficient.

Report > Reference Values

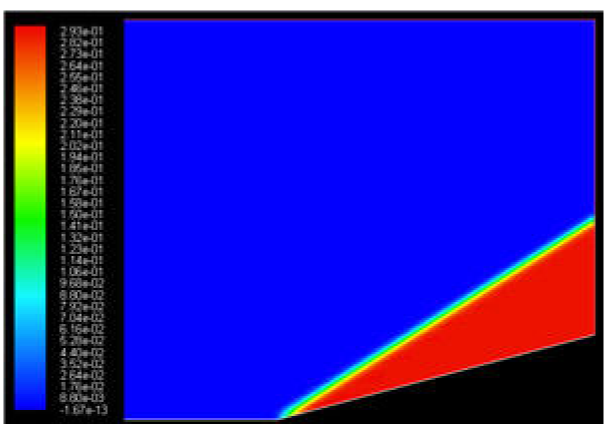
Select ***farfield*** under ***Compute From***.



The above reference values of density, velocity and pressure will be used to calculate the pressure coefficient from the pressure. Click ***OK***.

Display > Contours...

Select ***Pressure...*** and Static Pressure from under ***Contours Of***. Then select ***Pressure Coeffient***.



Higher Resolution Image

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.

Go to Step 7: Verify Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules