FLUENT - Supersonic Flow Over a Wedge- Step 6

- Problem Specification 1. Pre-Analysis & Start-up 2. Geometry 3. Mesh 4. Setup (Physics) 5. Solution 6. Results
- 7. Verification & Validation

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*.

Vectors			×
Options	Vectors of		
Node Values	Velocity	Velocity	
🗹 Global Range	Color by		_
Clip to Range	Velocity	Velocity Mach Number	
Auto Scale	Mach Numbe		
	Min	Max	
Style arrow	• 2.199958	3	
Scale 8.2	Surfaces		II.
Skip B	default-interi pressure_fai symmetry wedge	or ffield	
Surface Name Patter	m Surface Type	s	≡ =
	axis clip-surf exhaust-fan	2	1

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.

Options	Contours of		
Filled	Velocity Mach Number		
 Node Values Global Range 			
Auto Range	Min	Max	
Clip to Range	2.203165	3	
Draw Profiles	Surfaces		=
evels Setup 100 ÷ 1 ÷ Surface Name Patte	default-interio pressure_far symmetry.9 wedge	or field	
	Surface Types	5	=
Match	axis clip-surf exhaust-fan fan		< >

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.

💶 Reference Values 🛛 🔀
Compute From
•
Reference Values
Area (m2) 1
Density (kg/m3) 1 . 176674
Depth (m) 1
Enthalpy (j/kg) 844043.4
Length (m) 1
Pressure (pascal) 101325
Temperature (k) 300
Velocity (m/s) 1041.263
Ratio of Specific Heats 1.4
Reference Zone
fluid
OK Cancel Help

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