

Copy2 of Supersonic Flow Over a Wedge - Verification & Validation

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

Verification & Validation

Verification

Adapt the Mesh

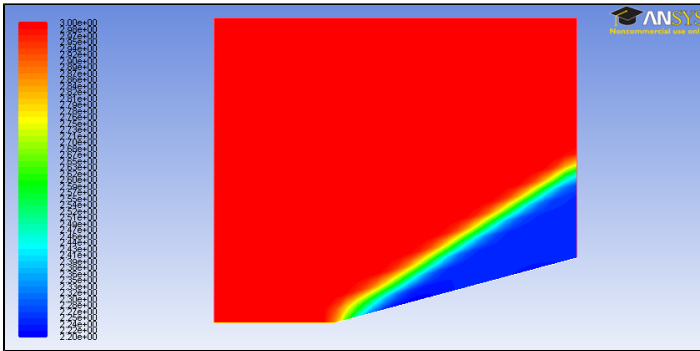
In order to test our simulation for convergence, we will refine the mesh. Refining the mesh will allow us to make sure that the results we are calculating are independent of the mesh. However, instead of refining the mesh everywhere (which would be wasteful, as most of the area of the domain far away from the shock has constant values), we will use our results to refine our mesh. Specifically, we are going to use the gradient of the pressure to determine where to refine the mesh.

Summary of the Above Video:

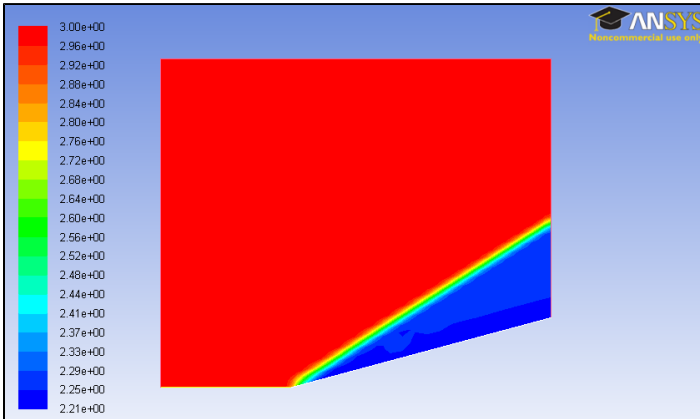
1. Display Current Mesh:
 - a. Mesh>Setup>Display (In newer versions, this is found under the Setting Up Domain tab, then Mesh > Display.)
2. In the Menu
 - a. Adapt>Gradient (In newer versions, this is found under Setting Up Domain tab, then Adapt > Mark/Adapt Cells > Gradient.)
 - i. uncheck Coarsen in Options Box
 - ii. select Gradient in Method Box
 - iii. Gradients of > Pressure > Static Pressure
 - iv. Compute
3. View Contours
 - a. Contours > Contours of Adaption > Existing Value
 - b. Compute
 - c. Display
4. Under Contours
 - a. Uncheck Auto Range
 - b. Change Min to 10000
 - c. Display
5. Under Refine Threshold enter 10000
 - a. Press Mark
 - b. Press Adapt
6. Display the New Mesh

Notice that the area surrounding the shock was refined. Now, re-initialize the solution, (Solution Initialization > Compute From Farfield > Initialize), and rerun the solution (you will also need to increase the number of iterations – I recommend 150).

Now, once again, plot the contours of the mach number. Below is a comparison of the mach number results from the original mesh and the refined mesh.



Original Mesh



Refined Mesh

The most striking difference between the two results is the thickness of the shock. Notice that for the refined mesh, the shock is less thick than for the original mesh. This shows that the refined mesh is converging towards the real case.

Comparison to Analytical Solution

In order to verify our simulation, we need to compare our results to either an analytical solution or an experiment. Below is a table comparing the values from the simulation with the calculations from the pre-analysis.

	Mach Number	Static Pressure (atm)	Shock Angle (degrees)
Theory Value	2.254	2.824	32.22
FLUENT Solution	2.243	2.803	34.99
Percent Difference	0.8%	0.7%	8.2%

As we can see from the table, we are getting a fairly good match between the computation and analytical approaches. From this we can build our trust in our simulation. Note that in this case, adaptive mesh refinement doesn't have a significant effect on the values behind the shock but makes the shock less smeared.

Save Project

Save the project using *File > Save*. Copy *wedge.wbpj* and the associated *wedge_files* folder to a flash drive. You will need both entities to resume the project.

Alternately, you can select *File > Archive* and save the project as one file called *wedge.wbpz*. When prompted, select the option to save *Result/Solution* also. You will then need to save only this file. This is also convenient to e-mail the project. Double-clicking on the *wedge.wbpz* file will resume the project. Note on some computers, double-clicking the *wbpz* file will open it in ANSYS AIM (a different application) rather than ANSYS Workbench. This is not right. The workaround is to start Workbench and then select *File > Open*.

Continue to Exercises

[Go to all FLUENT Learning Modules](#)