Supersonic Flow Over a Wedge - Exercises

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

Exercises

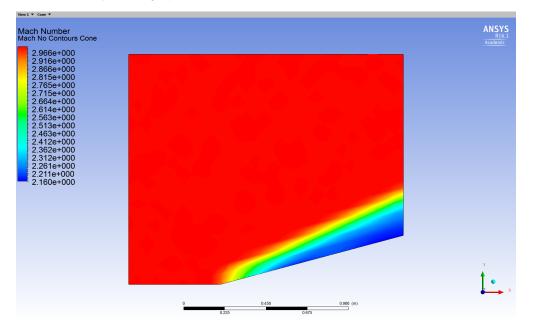
Supersonic Flow Over a Cone

Change the geometry from a wedge to a cone. What do you expect to change?

In the Outline window, click on General under Problem Setup. Under 2D Space select Axisymmetric. We also need to change the boundary condition for the symmetry to an axis. Click on Boundary Conditions in the Outline window. In the Boundary Conditions window, under Zone, select Symmetry. Change the Type to Axis. Now, reinitialize the solution, then run it again for 100 iterations.

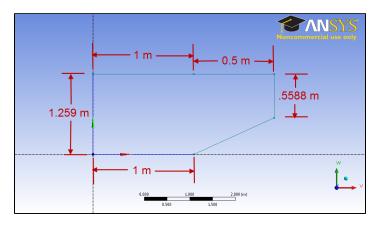
The result is displayed below. We did not change the mesh through adaption for this instead using the mesh generated in the tutorial. This needs be checked against the theoretical results for supersonic flow over a cone.

To look at the theory describing supersonic flow over a cone, see NASA's website.



Separated Shock

Next, we will alter the geometry to achieve a separated shock. Close FLUENT and open the Design Modeler. We want to increase the angle of the wedge above its critical angle. We will increase the angle to 35 degrees. Change the geometry's dimensions to match that of the diagram below.



 \checkmark

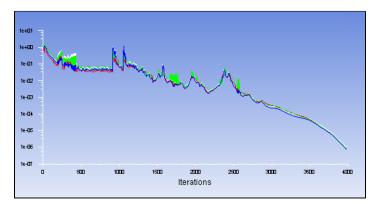
Once the geometry has changed, close the design modeler. We will have to re-calculate the solution, but we will want to change some factors affecting the solution. Usually, when you make an upstream change in ANSYS, the program will update all of the downstream data. We want to break this connection,

4 💓 Setup so right click

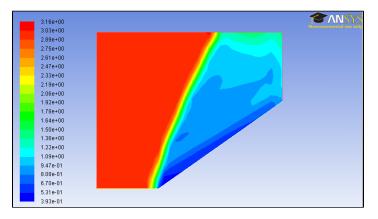
and select Reset. We will have to input the boundary conditions again, but that shouldn't take long - and will end up saving us time when we calculate the solution inside of the FLUENT environment.

Update the mesh by clicking Generate Mesh . Close the mesher, click 3 Mesh Next, open up the mesher by double clicking **Update Project** , then once again double click 4 Setup \checkmark

Update Project , then once again double click Setup Re-enter all of the data from Step 5 (here is link for reference). This time, set the Courant Number to 1.0. This will make the solution more stable, but it will solve more slowly. Run the solution again, this time with 5000 iterations.



Plot the contour plot of the mach number to see how the shock has changed.



Go to Comments

Go to all FLUENT Learning Modules