Supersonic Flow Over a Wedge - Exercises

Author: Rajesh Bhaskaran, Cornell University

Exercises

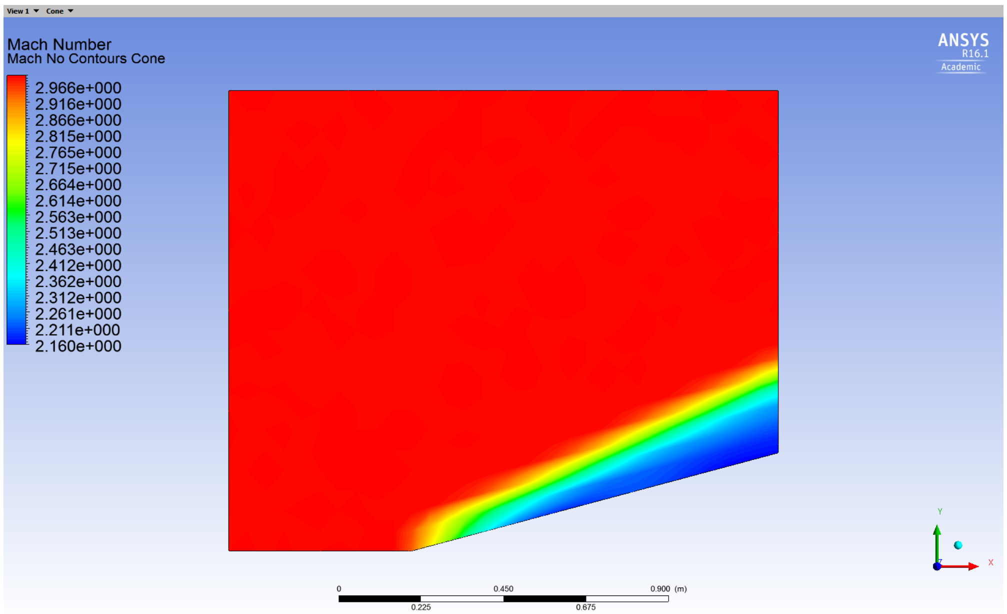
Supersonic Flow Over a Cone

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.

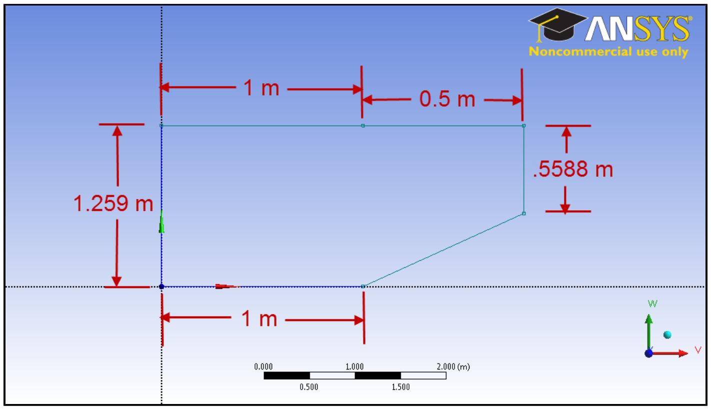
The result is displayed below. We did not change the mesh through adaption for this instead using the mesh generated in the tutorial. This needs be checked against the theoretical results for supersonic flow over a cone.

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



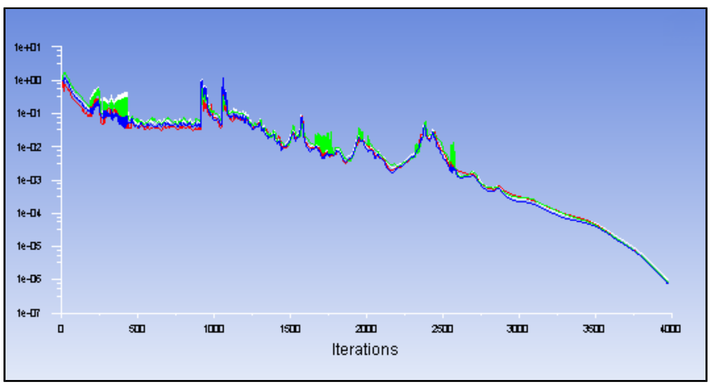
Separated Shock

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.

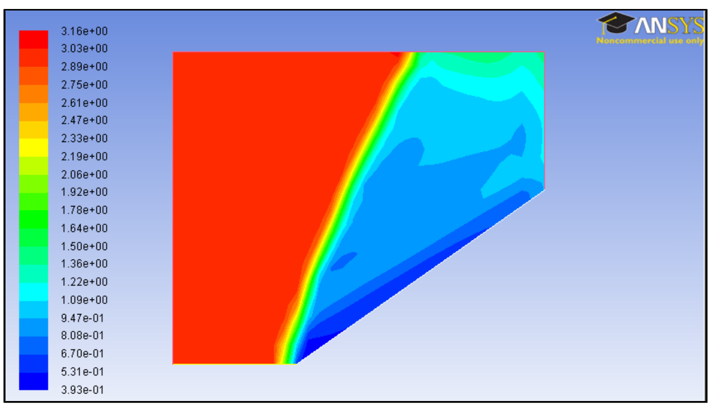


Once the geometry has changed, close the design modeler. We will have to re-calculate the solution, but we will want to change some factors affecting the solution. Usually, when you make an upstream change in ANSYS, the program will update all of the downstream data. We want to break this connection, so right click 4 Setup 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.

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