

Supersonic Flow Over a Wedge - Numerical Results

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

Numerical Results



This page goes through the post-processing steps for the Supersonic Flow Over a Wedge in the *CFD-Post* post-processor included with ANSYS Workbench. For instructions on performing the post-processing using the classic FLUENT post-processor, see [this page](#).

Mach Number Contours

The following video shows how to make a plot of the Mach number contours using *CFD-Post*. If the videos below don't appear in your browser, try reloading the webpage.



To properly send the additional quantities to CFD-Post, you need to do the following steps in the right order. First initialize your solution, then select the additional quantities as shown in the above video, and finally, run the calculation. (If you already ran the calculation and opened CFD-Post, you can update which quantities are transferred by selecting them and then running at least one iteration.)

Summary of the above video:

1. Some calculated parameters are not by default carried over into CFD-Post. We are interested in such quantities (i.e. Mach Number). To manually transfer a customized selection of quantities:
 - a. Click File > Data File Quantities...
 - b. Under Additional Quantities, select Static Pressure, Total Pressure, Mach Number, and Total Temperature.
2. Post-processing will be done in CFD-Post. To open, double-click *Results* in Workbench.
3. We are interested in viewing contours of Mach Number in CFD-Post
 - a. Select Insert > Contour > Name > Mach No.
 - b. Under Details of Mach No, select Locations > symmetry 1.
 - i. Variable > Mach Number > No Contours = 101
4. Turn on the mesh in the graphics pane
 - a. Check the box next to symmetry 1 in the Outline tree > Click Render > Show Mesh Lines.
5. Turn off the mesh by deselecting symmetry 1 in the outline tree
6. Save a copy of the figure in the graphics pane
 - a. Select the camera icon in the toolbar.

Pressure Contours

The following video shows how to make a plot of the pressure contours.

Summary of the above video:

1. Turn off the Mach Number contours in the graphics window
 - a. Uncheck the box next to Mach no in the Outline tree.
2. We are interested in viewing contours of pressure in CFD-Post
 - a. Select Insert > Contour
 - b. Name > Pressure Contours
 - c. Under Details of Pressure contours, select Locations > symmetry 1.
 - i. Variable > Pressure.
3. To increase the number of contours to 101
 - a. Under Details of Pressure contours, scroll down to # of Contours. Type "101".

Velocity Vectors

To plot the velocity vectors, follow the steps in the video below.

Summary of the above video:

1. We are interested in viewing velocity vectors in CFD-Post.
 - a. Select Insert > Vector
 - b. Type "Velocity vectors" under Name in the Insert Vector dialogue box that appears.
 - c. Under Details of Velocity vectors, select Locations > symmetry 1.
2. To make the velocity vectors more visible, turn off the pressure contours
 - a. Uncheck the box next to Pressure contours in the Outline tree

Plot Mach Number Variation Along $y=0.4$ m

First, we'll create a line at $y=0.4$ m. Then, we'll plot the Mach number variation along this line using the "Chart" facility in CFD-Post. These steps are shown in the video below. Note that you can export the data to a csv file using the *Export* button next to *Apply* in the Chart menu. You can then open the csv file in Excel to determine the values of Mach number, pressure etc. behind the shock to, say, 4 significant digits.

Summary of the above video:

1. Insert line $y = 0.4$ m in CFD-Post.
 - a. Select Location > Line > Name Line 1
 - b. Under Details of Velocity vectors, type Point 1 coordinates (0,0.4,0) and Point 2 coordinates (1.5,0.4,0).
2. Plot Mach Number along the newly created line $y = 0.4$ m
 - a. Select Insert > Chart
 - b. Type "Mach no along Line 1" under Name
 - c. Under Details of Mach no along Line 1, select the Data Series tab. Select Location > Line 1.
 - i. select the X Axis tab. Select Variable > X.
 - ii. select the Y Axis tab. Select Variable > Mach Number.
3. To increase the number of samples along line $y = 0.4$ m to 100.
 - a. Double click on Line 1 in the Outline tree.
 - b. Under Details of Line 1, type "100" in Samples.

[Go to Step 7: Verification & Validation](#)

[Go to all FLUENT Learning Modules](#)