FLUENT - Supersonic Flow Over a Wedge

**This page has been moved to https://courses.ansys.com/index.php/courses/supersonic-flow-over-a-wedge-using-ansys-workbench/
Click in the link above if you are not automatically redirected in 10 seconds.**

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

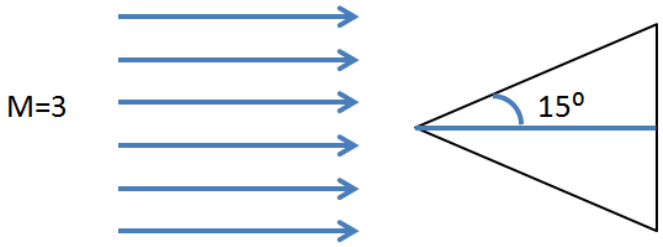
Supersonic Flow Over a Wedge

Created using ANSYS 13.0

⚠ This tutorial has videos. If you are in a computer lab, make sure to have head phones.

Problem Specification

A uniform supersonic stream encounters a wedge with a half-angle of 15 degrees as shown in the figure below.



The stream is at the following conditions:

$$\text{Mach Number } M_1 = 3$$

$$\text{Static Pressure } p_1 = 1\ atm$$

$$\text{Static Temperature } T_1 = 300\ k$$

Using FLUENT, calculate the Mach Number, static and total pressure behind the oblique shock that will be formed. Also, calculate the shock angle, pressure coefficient along the wedge and drag coefficient. Compare the FLUENT results with the corresponding analytical results.

Go to Step 1: Pre-Analysis & Start-Up

Go to all FLUENT Learning Modules