Flow over an Airfoil

Created using ANSYS 14.0

Problem Specification

In this tutorial, we will show you how to simulate a NACA 0012 Airfoil at a 6 degree angle of attack placed in a wind tunnel. Using FLUENT, we will create a simulation of this experiment. Afterwards, we will compare values from the simulation and data collected from experiment.

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

Go to all FLUENT Learning Modules