FLUENT - Flow over an Airfoil

This page has been moved to https://courses.ansys.com/index.php/courses/flow-over-an-airfoil/
Click in the link above if you are not automatically redirected in 10 seconds.

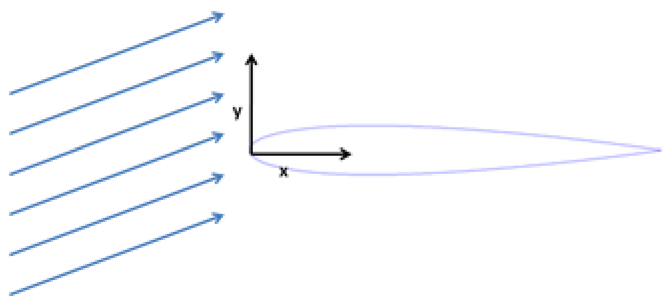
Author: Benjamin Mullen, Cornell University

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