

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

Problem Specification

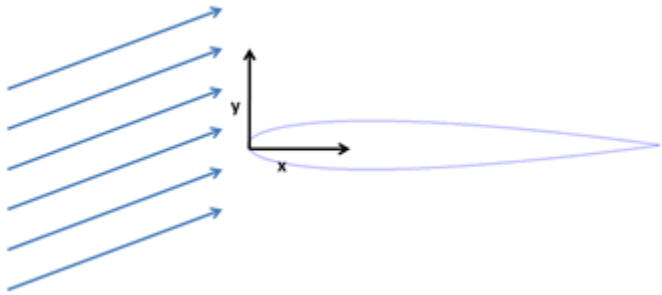
1. Pre-Analysis & Start-Up
 2. Geometry
 3. Mesh
 4. Physics Setup
 5. Numerical Solution
 6. Numerical Results
 7. Verification & Validation
- [Exercises](#)
[Comments](#)

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](#)