

Airfoil CFD Validation Using a NASA-provided Mesh



UnderConstruction

Created using ANSYS 14.5

Learning Goals:

In this exercise, you will learn to verify and validate the airfoil results from FLUENT by comparing with NASA CFD results (verification) and experimental results (validation). In the process, you will learn to:

- Import mesh in plot3d format.
- Split boundaries in Fluent by creating region registers.
- Apply Spalart-Allmaras turbulence model.

Problem Specification:

Consider the same problem encountered in the main tutorial.

A tremendous amount of effort has been put into the numerical study of this case of turbulent flow around the NACA 0012 airfoil and results have been reported on the NASA site for comparison. For a more detailed description of this problem, click the link below.

http://turbmodels.larc.nasa.gov/naca0012_val.html

In this case study, we will be solving the same problem using ANSYS FLUENT version 14.5. We set the inflow angle at 10 degrees. At the end of this case study, we will compare our results with the reported results as a step for validation. Published results can be found on this page below:

http://turbmodels.larc.nasa.gov/naca0012_val_sa.html

Getting Started:

We will use the exact mesh that was used in the NASA study. The mesh grid for the flow space around the airfoil are given in the link below:

http://turbmodels.larc.nasa.gov/naca0012_grids.html

The meshing files are provided in different formats (plot3d format and CGNS format) at different meshing resolutions. In this tutorial, we will be using the medium 2d grid (449*129, 257 points on airfoil surface) in plot3d format as an example. The meshing file we will be using can be downloaded [here](#).

Instructional Videos:

Please watch the following videos to model the flow in FLUENT.

To start, make sure you have saved your downloaded mesh file to a known location on your computer. If you are not able to launch FLUENT directly (as shown in the first video), you can still start a project through workbench. Open workbench and click and drag the *Fluent* icon under *Component Systems* into *Project Schematic* to create a standalone system. Double click on the *Setup* cell to launch FLUENT and proceed with the video instructions once in FLUENT.

The following video shows you how to import the NASA grid into FLUENT.

The following video shows you how to probe for desired points in the mesh which will be used later to split up the flow domain into region registers.

The following video shows you how to create region registers using the points obtained in the previous video. We will then split the mesh boundaries up according to these registers.

The following video shows you how to set up the physical conditions for our model flow problem. Unlike the main tutorial problem which uses viscous-laminar governing equations, we will now employ the Spalart-Allmaras turbulence model. We will also change to a pressure-based solver. Be aware that we will also change our viscosity value to adjust our Reynolds number to 6E6.

There are a few numerical solver specifications necessary before we can run the calculation. The following video shows you what specifications will likely make for a verifiable, converged solution.



UnderConstruction