



## TFES Lab (ME EN 4650) Computational Fluid Dynamics

---

Textbook Resource: Section 5.5 (pp. 208–223) from Pritchard, 8<sup>th</sup> ed.

### Objectives

- (i) Use ANSYS Workbench, a commercial software package for solving fluids engineering problems, to simulate the turbulent flow around a two-dimensional airfoil.
- (ii) Set up a computational domain and mesh, with fine grid spacing near the object and coarser grid spacing near the edges of the domain.
- (iii) Observe how the residuals as well as the drag and lift coefficients vary as a function of the number of iterations performed for different angles of attack of the airfoil.
- (iv) Visualize the resulting numerical solution using contour plots of the velocity and pressure, as well as pathline plots.
- (v) Validate the numerical solution by comparing the calculated drag and lift coefficients with that obtained from experimental data.

### Computational Fluid Dynamics (CFD) Software

ANSYS Workbench is the general name for the collection of simulation tools sold by Ansys, Inc. of Canonsburg, PA. The University of Utah has a site license to run ANSYS Workbench in the CADE Lab and the Engman Lab (both located in WEB, 2nd floor) and MEK Computer Lab (in the basement of MEK). You will need to remotely access the computers that run ANSYS Workbench in the labs. Consult the Help File posted on CANVAS for how to do this.

ANSYS Workbench simulates fluid flow using a computational fluid dynamics (CFD) program called Fluent. There are a number of different numerical techniques available to perform CFD. Fluent utilizes the *finite-volume* method, in which the flow domain is discretized into many small volume elements. Conservation of mass and momentum is applied to each volume element. In addition, each volume element is coupled to its nearest neighbors through surface forces (i.e., pressure and shear stress). Boundary conditions must be specified at the edges of the domain; and, an initial guess of the velocity everywhere in the domain must be supplied in order to start the solver. The solver then performs a series of iterations in order to determine the velocity and pressure distribution inside the domain. Iterations continue until the convergence tolerance is achieved as set by the user. In this method, convergence means that the velocity field ceases to change substantially between successive iterations.

The accuracy of the numerical simulation depends heavily upon how the domain is discretized. This is referred to as the grid or the “mesh”. The mesh spacing must be fine enough so that changes in fluid properties do not vary significantly across the volume elements comprising the mesh. The accuracy of the numerical simulation also depends on the model used to represent the behavior of the turbulence in the flow. It is often too computationally expensive (i.e., the solver takes too long to reach convergence) to directly solve for the turbulent motions in the flow. So, these are typically modeled using a combination of

algebraic and differential equations that contain an array of adjustable parameters. There are many different types of turbulence models. Selecting the appropriate model for any given flow can be challenging and usually requires research into the primary literature (i.e., published journal articles). Even if one is confident in the selection of a turbulence model, tuning the parameter values in that model can be a nearly impossible task. Therefore, the default parameter values are generally utilized; even though these will not likely be optimized for the flow under investigation.

## Two-Dimensional Airfoil Mesh

In the present lab, the computational mesh for flow over a two-dimensional airfoil has been generated using free software called **Construct2D**. Mesh generation for CFD simulations generally requires a lot of time investment. In fact, people working in CFD tend to spend most of their time creating and refining the computational mesh. An important aspect of performing CFD simulations is to determine the finest mesh required in order to obtain mesh-independent results. This means that if you refine your mesh further (i.e., make the volume elements smaller), the simulation results do not change appreciably. Several different meshes have been created for you (and are available for download from CANVAS) in order to explore how mesh spacing affects the simulation results.

## Laboratory Procedures

This lab assignment consists of TWO parts:

- (i) During your lab section, complete the provided *Tutorial* of high Reynolds number flow around a two-dimensional airfoil. Note, you may need more than 80-minutes to finish the tutorial if this is your first time using **ANSYS Workbench**.
- (ii) Modify the procedures in the *Tutorial* appropriately to complete the following two CFD simulations on your own of flow over a NACA 0012 airfoil at a chord Reynolds number of  $Re_c = 1.5 \times 10^5$ . All other settings and parameters should be the same as those used in the *Tutorial*.
  - (a)  $\alpha = 5^\circ$  (angle of attack, pre-stall condition),  $n = 400$  (mesh with 400 nodes along the airfoil)
  - (b)  $\alpha = 12^\circ$  (angle of attack, post-stall condition),  $n = 400$  (mesh with 400 nodes along the airfoil)

## Required Plots

- 1a. In a single figure, include two images of your computational mesh. The top image should be of the entire domain and the bottom image should be zoomed in near the airfoil. These should be for the case of  $n=400$  nodes on the airfoil with an angle of attack of 0 deg.
- 1b. In a single figure, include three convergence plots for the case with an angle of attack of 12 deg. The left plot should be the residuals (continuity, x-velocity, y-velocity, k, omega) as a function of number of iterations; the middle plot should be the drag coefficient as a function of number of iterations; and, the right plot should be the lift coefficient as a function of number of iterations.

- 1c. In a single figure, include two contour plots of the velocity magnitude for the case with an angle of attack of 12 deg. The top plot should be of the entire domain; and, the bottom plot should be of the region zoomed-in near the hexagon.
- 1d. Provide a plot of the pathlines near the airfoil for the case with an angle of attack of 12 deg.
- 1e. Provide a contour plot of the pressure coefficient near the airfoil for the case with an angle of attack of 12 deg.
- 1f. In a single figure, include two plots side-by-side of the drag and lift coefficients as function of angle of attack showing the experimental data. Overlay your simulation results for angles of attack of 5 deg and 12 deg. Use the Matlab figure files posted on CANVAS under the “Resources” row.
- 1g. Provide a table comparing the lift and drag coefficients from the simulation at angles of attack of 5 deg and 12 deg with those obtained from experimental data at a similar Reynolds number. Also include the percent difference between the simulation values and experimental values.

## Short-Answer Questions

- 2a. Discuss how the CFD simulation results vary with mesh spacing. Use the results from the tutorial to answer this question. Be sure to be specific in your response. [4–6 sentences]
- 2b. Discuss what is “validation” in the context of CFD simulations and why it is necessary. [2–4 sentences]