

MEEG332 - Fluid Mechanics II

Spring 2019

CFD Coding Assignment #1

Due Date: 5/20/2019

Please reach out to: jkuehl@udel.edu or abatista@udel.edu for any questions.

For this coding challenge you are to perform a computational fluid dynamics (CFD) calculation. Meaning: you will do all the major steps of for a CFD run: geometry, meshing, solving, visualization, and analysis. Details below.

Preparation

1. If you have access to a windows computer you should be able to download Ansys Fluent as a student (the entire suite). Here is the link: <https://www.ansys.com/academic/free-student-products>

You'll want to click the "ANSYS Student" box. If you have windows 7,8, or 10, you can download version 19.2 of Ansys. Under "Prior Releases" version 18.2 should work also with Windows 7 or 8 or if you have any trouble with v19.2 of Ansys.

2. Choose to do either a high-speed project, or a low-speed airfoil project.

3. Depending on your project choice follow one of the below tutorials from Cornell

3.1 Low-Speed - Flow Over an Airfoil Tutorial

<https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Flow+over+an+Airfoil>

3.2 High-Speed - Supersonic Flow Over a Wedge Tutorial

<https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Supersonic+Flow+Over+a+Wedge>

4. Perform your CFD Calculation. Below are the requirements for each case.

4.1 Low-Speed - Flow Over an Airfoil Project

Perform an external, viscous, CFD calculation for any airfoil you desire (except the NACA 0012 from the Cornell tutorial) at a 4 degree positive angle of attack. The freestream velocity must be between: $(100 \text{ m/s} < u < 250 \text{ m/s})$.

4.2 High-Speed - Supersonic Flow Over a Wedge

Perform an external, viscous, supersonic CFD calculation for a wedge, cone, or cylinder you want except the 15 degree wedge from the Cornell tutorial, or any other wedge that is between the angles of 11 and 19 degrees. Example acceptable geometries are: 10 degree wedge, 20 degree wedge, 15 degree cone. The freestream Mach # must be between: $(1 < M < 4)$.

5. Constraints

There are certain constraints your run must include, some are just to simplify the calculation.

Gas: air, Density and Pressure: sea level conditions, Use: Metric units (meters, kilograms, seconds), Laminar viscous flow conditions.

Deliverables

The below are the deliverables for the project. It doesn't have to be a report, but everything below should be included. For instance, all of the below can be turned in as a single Word or PDF file.

1. Identify and provide a CAD Model of your geometry. This can simply be the geometry created in Ansys or any other CAD software you used.

2. The mesh. Remember these are external flows. So, the only wall boundary conditions should be the airfoil, wedge, cone, etc...

3. Solving

3.1 A list of your initial, boundary, and solver conditions.

3.2 Solver output. I.e. Residuals, # of iterations, etc...

4. Visualization

4.1 Contour plots of: u, v velocity, Mach #, and pressure distributions

5. Specific Project Requirements

5.1 Low-Speed Project Only

5.1.1 Identify the airfoil and the freestream velocity you used.

5.1.2 Perform a Lift over Drag (L/D) calculation and comparison to a benchmark (I.e. a comparison to the airfoiltools.com data for your airfoil is acceptable)

5.1.3 Plot the height above the wall (y) vs velocity (u direction) from the wall to the freestream (velocity profile) of the airfoil or the high-speed geometry. Three locations should be provided towards the top of the airfoil upstream of the separation point, near the separation point, and downstream of the separation point.

5.2 High-Speed Project Only

5.2.1 A well-defined shock in your contour plots. Meaning, too wavy, too thick, or no shock at all, are examples of not well-defined.

5.2.2 Provide the angle of geometry and the angle of your shock, then provide a comparison of the shock angle from your CFD calculation and the theoretical Theta-Beta-Mach diagram.

5.2.3 Plot the height above the wall (y) vs velocity (u direction) from the wall to the freestream (velocity profile) of the airfoil or the high-speed geometry. Three locations should be provided towards the leading edge of the geometry, roughly the middle, and towards the end.

5.2.4 Temperature contour plot

Note: All provided results should be correct with respect to the fluid's flow conditions.