Assignment #7 : CFD 2 — ME 412

due 10/26/2016 before midnight via Learning Suite — **50 possible points**

7.1 This week's assignment will give you an introduction to running CFD on a supercomputer. Read all the directions before proceeding so that you have a good idea of where you are headed. For various reasons I *highly* recommend submitting jobs to the supercomputer as early as possible. Don't wait until the last day.

(a) Request a supercomputing account if you don't already have one. **Make sure to request your account under this class, not as an account under my name.** I've obtained approval for the class as a whole, so if you request it correctly you should get automatic approval.

(b) We will analyze the the MD 30P30N multi-element airfoil (geometry available on the course website) using a 2D RANS simulation at the flow conditions specified below.

M_∞	0.2
α	8.23°
Re	9×10^6

Provide calculations and brief rationale for selecting significant CFD parameters: domain size and type, prism layer height, first cell height in prism layer, number of layers in prism layer, freestream boundary conditions (pressure/temperature), and anything else you think would be important to help someone understand/reproduce your approach.

(c) Watch the first 4 Getting Started videos for the Fulton Supercomputing Lab (Intro to FSL, Job Scheduler, BYU's Script Generator, Intro to SLURM Tools). Also review the following sections of written documentation available on the Documentation tab of the FSL website: Logging In, File Transfers, Storage, General Batch Information, Job Script Generator, SLURM Commands, Applications/Star-CCM+, Unix Tutorial.

(d) Using the supercomputer, perform a grid convergence study with the lift/drag ratio as your metric. Include a plot showing L/D versus your measure of grid size, **and** include a few screenshots of your final mesh. We have created a few submission files to help you with this (see tips on next page).

(e) Compare the pressure distribution **and** the lift coefficient from your grid-independent solution to the experimental data available here (tabulated data from this paper is available on our course website using this web tool to extract data from the plots). Show a plot of your pressure distribution, and report your lift coefficient.

(f) Discuss your findings.

See the next page for some helpful tips.

- You need three files:
  1. Submit script – We provided this for you (called submitstar), but in general one can be generated with the submit script generator on the FSL website (with some modification). This script tells the supercomputer what to do, how many processors to use, what commands to execute, etc. You need to modify this slightly to note the location and names of the other two files below, and possibly to change the number of processors/wall time. You submit your job using the command "sbatch script_name" at the command line of the supercomputer.
  2. Java macro – This file tells Star-CCM+ what to do. We provided a simple one for you (called run.java). The only thing run.java does is to "press run" on your simulation file and then save the results once it reaches the stopping criteria. You may also create your own java macro using the record macro feature. If you create your own you may want to review the [Automation/Simple Java Macros: Post-Processing Objects] tutorial. Creating your own can be helpful if you want to automate and parameterize things like changing the mesh size, stopping criteria, etc. (or you can manually make changes and retransfer files for use with the basic run.java script).
  3. Simulation file – This is the Star-CCM+ simulation file containing your geometry and any other setup your macro will start from.
- The version of StarCCM+ that you use to create your sim file must match the version of the module you load on the supercomputer. The current supercomputer version is 11.04.10. If you use the CAEDM Linux computers these should already match. If CAEDM Windows, they may not match. If you have installed on your own personal computer, as long as you downloaded the same version as noted in my email then you'll be fine, otherwise you'll need to reinstall.
- If you decide to create your own java macro, you could record the entire construction of your model, but it is generally easier if you only record the portion you want to change (e.g., the mesh generation and re-solve). Note that if you save anything in the Java macro, you will need to go in and edit the path. The call to the path should look like:

  ```
  resolvepath("filename.csv")
  ```

- Don't forget to initialize your solution.
- Don't use a "Velocity Inlet" boundary condition. That is for incompressible flow. Use the "Free-Stream" boundary condition instead. This will allow you to specify Mach number directly along with pressure and temperature (and thus density and viscosity for the Reynolds number).
- In general, one needs to experiment with different turbulence models. We will save you that time and tell you that for this particular problem we have found better results with the Spalart-Allmaras model.
- Limit the number of processors you use to 24 or less. We are a large group and so we need to be careful to not monopolize resources being used for research.
- Don't request more wall time than you actually need because it will take longer to start. You shouldn't need more than 1 hour per simulation—most meshes will solve in 10 minutes or less with 16 processors.
- Every run on the supercomputer produces a *.out file containing all the output from your run. If your simulation failed, you should look in the *.out file for clues as to what went wrong.