Assignment #1: Dyn. Similarity
## due 9/7/2016 before midnight via Learning Suite

ME 412

50 possible points

Objective: Extend your understanding of dynamic similarity. Gain experience defining engineering problems from real-world problem statements, and be introduced to Star-CCM+ (Focused on Learning Outcome 1, supporting Outcome 5).

Project: Read the project guidelines. Start brainstorming topics that you are interested in. Do some preliminary reading to see if your topic is project-worthy. Start looking around for someone to partner with.

1.1 In aerodynamic wind tunnel testing it can be difficult to simultaneously match the full scale Mach number and Reynolds number. Learn about when this problem might occur, what some of the solutions are, and what the advantages and disadvantages of these approaches are. Describe your findings.

1.2 Computing: Two-dimensional flow around a cylinder sounds simple, but actually has some rather complex Reynolds Number dependent behavior. Figure 9.21 in your text shows some of the interesting behavior, which we will discuss in more detail when we get to viscous flows. For now, we are going to use dimensional analysis to analyze case (C) known as a von Karman vortex street, which occurs around a Reynolds Number of 100. Watch this youtube video, which shows the phenomenon in CFD and in a water tunnel (incidentally this video is a pretty good example of an appropriate project for this class).
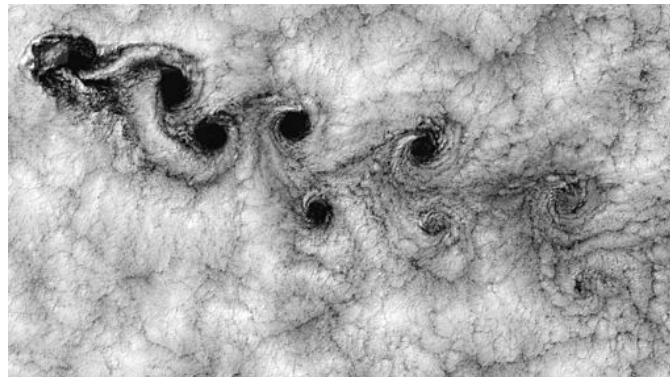
Satellite images of a von Karman vortex street in the clouds behind an island off the Chilean coast.

We want to determine how the shedding frequency ω changes as a function of Reynolds number using experimental data. An experiment was conducted using a rectangular water tunnel with an adjustable flowrate. A dye was injected so that the vortices could be observed and counted. For separate experimental runs the flowrate and cylinder diameter were varied. In each case, a timer was used to find the shedding time for 10 vortices.

An Matlab script (with data from the student companion site for Fundamentals of Fluid Mechanics) is provided that gives the temperature of the water (T), the width of the channel (b), the volume flowrate (Q), the water depth (y), the cylinder diameter (D), the number of vortices shed (N), and the time it took to shed the vortices (t). You should plot the shedding frequency as a function of freestream velocity, and create a *second* plot showing these relationships using nondimensional parameters.

After plotting the experimental data, overlay your second graph with the equation proposed in this journal paper (which you can find online): "A new Strouhal–Reynolds–number relationship for the circular cylinder in the range $47 < \mathrm{Re} < 2 \times 10^5$". Briefly comment on your findings.

1.3 Open-ended: If it started raining and you wanted to get under a shelter, would you get less wet by walking or running? Why? As a specific example: if you needed to cross a football field in the rain, how many ounces of water would your clothes absorb?

1.4 CFD: Complete the "Introduction to Star-CCM+" tutorial and turn in some plots showing your work. No discussion in needed for this task.

Star-CCM+ is installed on the CAEDM computers both on Windows and Linux (I recommend becoming familiar with Linux). You can use it locally or login remotely. It is under Program Files in the CD-adapco folder on the Windows machines. Searching in the Windows menu for "star" is the easiest way to access it. It is also available on Linux through the command line or in Applications. It can be run from the terminal using the command "starccm" on the Linux workstations. In the Applications drop down menu it is under CAEDM Apps. Tutorials are available under the help menu. Note the URL of the tutorial when launched in your browser. This is the path you will need to follow (up to doc) in order to locate files mentioned in the tutorial. Under "Resources" on the course website, there is a pair of videos to help you with launching the program and finding the tutorials.

Star-CCM+ can also be installed on your individual computers. We will provide directions later in the semester if you would like to pursue this option.

Later in the semester we will learn more about how Computational Fluid Dynamics (CFD) works and how to use it effectively, but for now we will focus on tutorials to get some experience. In these first tutorials it can feel like you are just clicking buttons. Understanding will come in time, but please do ask lots of questions.