# Finite element modeling and Navier-Stokes

## I. READING

- Chapters 6 and 7 (again!)

## II. INTRODUCTION

In this homework we will use Comsol to model some 2D flow problems. The point is to get you a little more familiar with finite element modeling and gain a little intuition about flow problems.

Feel free to explore things that seem interesting to you. Keep what you turn in simple. Create a plot or two and provide a little explanation of what the results mean. Keep your geometry scaled so that the length scale is 1 unit, the density is 1 unit, and the viscosity is the inverse Reynolds number. For reasons we will discuss in class you can expect the software to break down once the Reynolds number exceeds a few hundred or maybe 1000, depending on the problem. In all cases keep the problem 2D and steady flow.

You can work on these problems with a partner and turn in one joint assignment if you would like.

## III. PROBLEMS

1. Create a 2D channel which is 2 units high (in $y$) and has a length of 20 units (in $x$). Set the inlet corner for the rectangle at (0,-1). Specify the inlet $x$ velocity field at $x = 0$ to be $u = 1 - y^2$. This parabolic velocity field is an exact solution to the Navier Stokes equations. Set the outlet condition at $x = 20$ to be at a pressure of zero. Set the viscosity (Reynolds number) to 1 initially. When you run this simulation, it should be pretty simple and show the parabolic velocity profile the whole length of channel. Use the simulation to measure the pressure drop across the length of this channel. Create a plot of pressure along the channel's centerline as a function of $x$. Change the Reynolds number and see how the total pressure drop changes (holding everything else constant).

2. Adjust the channel to be 4 units high and move the corner to coordinate (0,-2). Specify the inlet velocity field at $x = 0$ to be $u = (4 - y^2)/8$. This velocity field is an exact solution to the Navier Stokes equations and has the same flow rate as the previous problem. Recompute the pressure drop. How does the required pressure for a given flow rate scale with the channel height? From the previous two problems, can you figure out the formula for the channel resistance $\Delta P = QR$ where $Q$ is the total flow rate (4/3 in this case) $\Delta P$ is the total pressure drop and $R$ is a function of length, height, and viscosity.

3. Now add the two channels of the previous problem in series. Make the small channel suddenly expand into the larger one. Plot streamlines; it looks best to adjust the "Streamline Positioning" to "uniform density" with spacing of around 0.01. Plot the pressure as a function of $x$ along the centerline. Increase the Reynolds number from 1, 10, 100, to 200 (above this the solutions may be suspect). Pay attention to the change in streamlines. Notice that at low Reynolds number, we could add the pressure drops as though the problem were resistors in series. At higher Reynolds number this assumption will break down.

4. Reverse the direction of the flow (you can just swap which boundary is specified as the inlet and outlet and keep the geometry the same). Specify the inlet velocity on the large channel to be $u = (4 - y^2)/8$. Repeat the same numerical experiments as the previous problem. Comment on the resistor in series analogy.

5. Consider flow over a cylinder. Place a cylinder with a diameter of 1 inside a 2D channel. Center the cylinder at the origin. Make the channel height go from plus and minus 3. Make the length sufficient such that entance/exit effects are not important (20 units should be OK). Set the inlet velocity on the left to be $u = (9 - y^2)/9$ - our parabolic velocity profile with a velocity of 1 along the centerline. Plot the streamlines at different Reynolds numbers. Look up the result for flow over a cylinder in a uniform free-stream flow and see if your results make sense in general.

   Compute the total force acting on the cylinder. Compute the total force, by selecting "derived values", "line integration". Select the surface of the cylinder. Under "expression" look for "total stress, x-component". Adjust the Reynolds number and recompute the drag. Start low, (like 0.01) and then increase up until a few hundred (where the solution will start to break down). Make a plot of drag coefficient versus Reynolds number. Compare your solution to that of a cylinder in a free stream which is easily found online.