

# Solutions to Navier-Stokes

## I. EXPERIMENT COMPARISON

In the following problem we will make use of the simple solutions to the Navier Stokes where we assume long channels where the flow doesn't change along the length of the channel. In this case, the flow velocity only changes across the channel. In all the problems here we construct the solution as the sum of Couette and Poiseuille flow.

We will demonstrate a simple experiment in class of a solid cylinder dropping through a liquid filled tube. Outside of class, some data points were recorded and are reported in Table 1. In the experiment we can vary different parameters such as the fluid and the geometry.

$D_{cyl}$ (mm)	$L_{cyl}$ (mm)	$M_{cyl}$ (g)	$D_{tube}$ (mm)	$\mu$ (Pa-s)	$\rho$ (kg/m <sup>3</sup> )	V (cm/s)
24.43	100	66.4	25.8	0.0009	1000	2.05
23.34	100	61.2	25.8	0.0009	1000	7.4
22.43	100	56.1	25.8	0.0009	1000	14.2
24.38	200	132.5	25.8	0.0009	1000	2.05
24.43	100	66.4	25.8	0.05	1210	0.042
23.34	100	61.2	25.8	0.05	1210	0.177
22.43	100	56.1	25.8	0.05	1210	0.412

TABLE I Experimental data for a cylinder dropping in a tube.

Observe the experiment, and try to quantitatively explain the behavior you see. This exercise is closer to how a "real" problem is stated rather than the usual book problems. A few things to guide you, though ultimately all we are after is a simple expression that explains the overall behavior and a comsol simulation of the full problem. We will work through a large part of this problem in class.

- Start with a 2D approximation. This assumption will speed up and simplify your ability to compute things analytically and with comsol. We will provide 2D analog of the problem in Comsol for you to validate against. We formulate the problem where you set the speed of the shuttle and you are trying to compute the total force on the object. You may want to play with the Comsol simulation for a little bit, just to experiment and get some intuition. Work out the 2D problem analytically assuming a sum of Poiseuille and Couette flow. You can now compare Comsol results to your analysis on a one to one basis to make sure it is correct.
- Using your 2D analysis, assume a very thin gap and come up with an analytical relationship that explains the data. In assuming a thin gap, you ignore the curvature of the gap and just multiply the stress in the gap by  $2\pi R$  to get the total viscous stress.
- Conduct a 3D simulation in Comsol and compare the comsol results to both the 2D model. In the 3D simulation, you will want to use axisymmetric mode, as it will be faster to solve. Also, note that the experimental data points with water are pretty high Reynolds number and will have difficulty solving. You should just use Comsol for the higher viscosity data. Also when computing the force on the object: Right click "derived values", select "Integration", then "line integration". Click the top and bottom surface of the shuttle. In "expressions" add  $2\pi r p$ . Right click again on "derived values" and select "Integration", then "line integration". Click the side of the shuttle that is running parallel to the tube wall. In expression click "add expression". In the confusing menu of options, Find "component", "laminar flow", "Auxiliary variables", "Viscous stress", "viscous stress Z component". The simulation will allow you to confirm your model for a few sets of parameters.
- Once you are convinced your model works, you can use the analytical solution to compare to the experimental data. It should look pretty reasonable.

## II. PROBLEMS

1. Read through the slider bearing problem in section 8.6 of the book.

Now consider that the block has three regions of height and the total length is  $3L$ . From  $0 < x < L$  and  $2L < x < 3L$  the height is  $h$ . In the middle from  $L < x < 2L$  we will consider two cases where the height is either  $h/2$  or  $2h$ . Assume the total pressure difference across the block is zero, as in the problem in section 8.6.

- Without writing a single equation, sketch the velocity field under the block and pressure as a function of length for this problem. This plot need not be quantitative. Try to reason whether the block experiences a force perpendicular to the wall, and if so is it attractive or repulsive?
  - Compare your educated guesses to a 2D steady state Comsol simulation. Note when doing the Comsol simulation you need to shift the reference frame so you are riding on the block at constant velocity and the lower wall is stationary.
2. Flow over a cylinder in free stream of uniform flow. Go back to your previous Comsol model where we looked at flow patterns over a cylinder in 2D flow. Now we will compute the drag coefficient. Adjust the Reynolds number to 1; just to get a nice and stable solution to start with. Adjust the boundary conditions. The inlet should be uniform flow with x velocity of 1. Outlet on the right should be constant pressure. The upper and lower boundaries should have a "moving wall" with a velocity of 1. Make the region of flow that you are solving go from  $-5 < y < 5$  and  $-10 < x < 20$ . Place the cylinder with a diameter of 1 at  $x = 0$  and  $y = 0$ . Adjust your comsol simulation to compute the net force on the object (We can show you how to do this in class - but add "derived values" in results, select line integration, select the walls of the cylinder, click on replace expression, under component 1, laminar flow, auxiliary variables, total stress - select normal stress x-component. Convert this number to a predicted drag coefficient - being careful of the units. You can either run different Reynolds numbers by hand, or learn to set up a parametric sweep (again, we can show you). Either way, you will want to space your points in Re logarithmically. Compute drag as a function of Reynolds number using the stationary solver from Re=0.01 to Re=200. Compute drag as a function of Reynolds number using the transient solver from Re=50 to Re=2,000. The transient drag will fluctuate in time, you can just take an average value. You don't need to go overboard - just select 10 values of Reynolds number or something along those lines. Create a single plot of drag coefficient versus Reynolds number and compare to the experimental data. You can look up the result online to compare.