Key to Assignment 9
Assignment 9:
Consider the 2D device shown. Water (density 1 g/cm^3, viscosity 0.01 g/cm sec) comes in on the left-hand side with an average velocity of 1 cm/sec. The device is 1 cm long and 0.43 cm high. The slits removed are 0.01 cm high and 0.7 cm long. If the lower, left corner is at x = 0, y = 0 the inlet velocity is (in cm/sec) 13.95y–32.45 y*y. Use the 2D version of Comsol to solve the Navier-Stokes equation in this situation.

- Find the pressure drop needed to achieve this flow.
- Plot the results showing the streamlines and pressure (in color).
- Where is the flow fully developed as between two flat plates?
- What is the Reynolds number at the inlet? in each channel? Is the flow laminar?
- What fraction of flow goes through each channel?

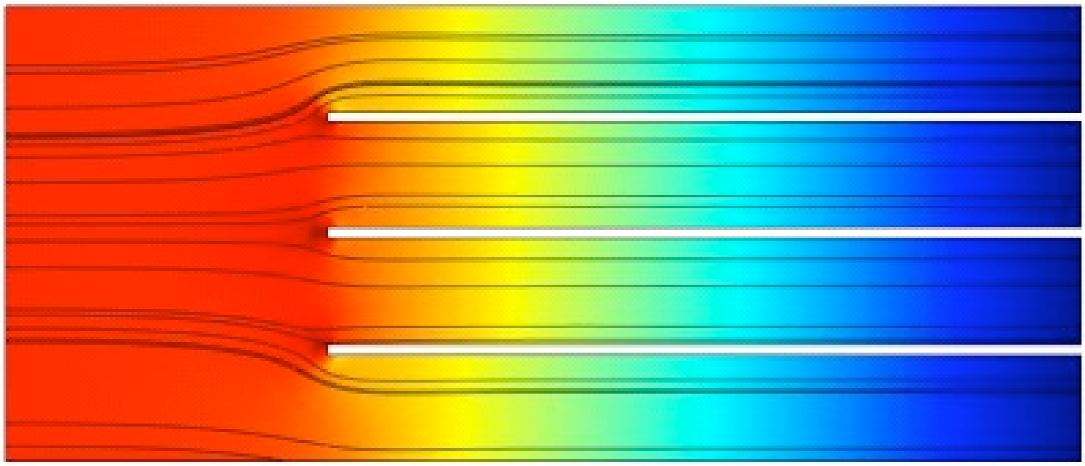
• If the flow is not identical in the different channels, what design changes do you suggest to make them identical? (Just describe the trends, you don't have to produce the design.)

1. Open Comsol Multiphysics, Chemical Engineering, Momentum, Laminar, Incompressible Navier-Stokes, Steady-state and click OK.
2. Under Options/Preference/Modeling change the units to cgs
3. Draw the geometry shown below. This was done by drawing one big rectangle and three small rectangles and making the domain be the big one minus the three small ones.
4. Click on the triangle to create a mesh. This one has 670 triangular elements.
5. Under Physics/Subdomain, set the density to 1 and the viscosity to 0.01.
6. Under Physics/Boundary Conditions, leave all the boundary conditions as no slip except for: left boundary, the velocity u is set to 13.95y–32.45 y*y, and v = 0. On the four outlet boundaries choose outlet and then chose the one that says p = 0 (not the one that also says no viscous stress).
7. Click the = to solve. It says there are 3363 degrees of freedom.
8. Post-processing – Plot Parameters. Under General, click surface and streamlines if they aren't already chosen. Choose the Surface Tab and set the variable to be plotted to p. I also clicked the Streamline tab and changed the color of streamlines to black. Click OK. The color scale indicates the pressure goes from 0 to 11.34 (in $dyne/cm^2$). It also shows that the flow is fully developed from x = 0 to about x = 0.1 and from x = 0.5 to x = 1 cm. The image is exported (shown below) by choosing File/Export, then Export.
9. Post-processing – boundary integration. To find the pressure at the inlet, we could put the cursor near there and the pressure appears near the bottom of the screen. Instead, calculate the average pressure at the inlet. It should be a constant, independent of y, but the inlet velocity is not specified to enough digits, so there are small variations. Under the boundary integration, put the variable p in the window (or use the pull down menu). Since we want the average over the region which is 0.43 long, put p/0.43 in the window. The answer is 9.773196. Do the same thing with the variable u, getting 0.429684. This is the value of the integral

$$\int_0^{0.43} u(y)dy$$

To get the average velocity, divide by 0.43 to get 0.999266 cm/sec. For the exit boundaries, the top and bottom boundary each have an integral of 0.102981 and the two inner ones have a value of 0.111868. This means the total flow rate out is 0.429698 (adding them up), which is within

0.000014 of the flow rate in. Thus, the mass balance is quite accurate. This also shows that 0.102981/0.429684 = 0. 2397 of the flow goes out the top and bottom outlets and 0.111868/0.429684 = 0.2603 of the flow goes out each of the middle outlets.



The Reynolds number at the inlet is defined using the average velocity and height between the channels. It is

$$Re = \frac{\rho < u > H}{\mu} = \frac{1 \times 0.999 \times 0.43}{.01} = 42.9$$

At the outlets it is

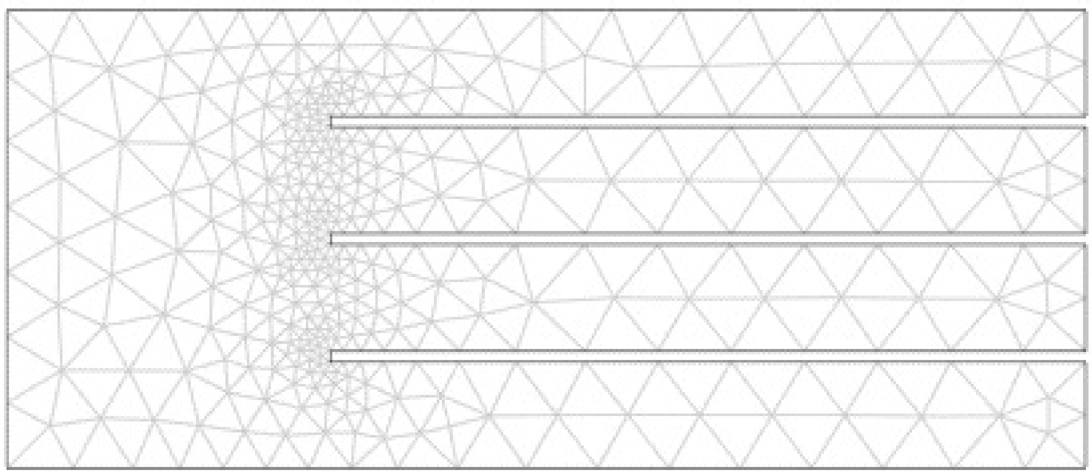
$$Re = \frac{\rho < u > H}{\mu} = \frac{1 \times 0.103}{.01} = 10.3 \text{ and } \frac{1 \times 0.112}{.01} = 11.2$$

Since the flow is a bit less for the channels on the outside, we could increase the flow there by making them slightly larger than the innermost channels.

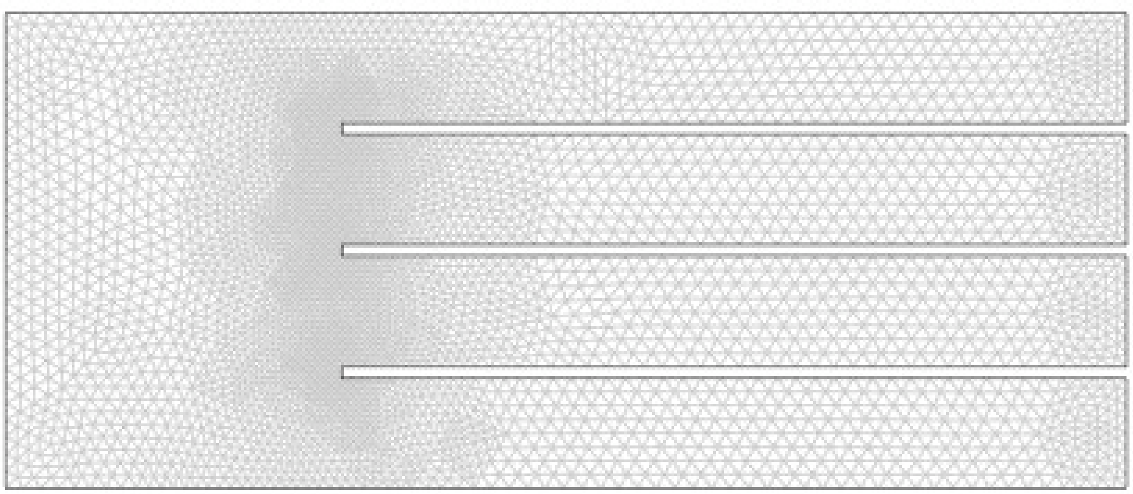
One feature of the flow is that the peak pressure is 11.3 whereas the pressure in is only 9.77. The reason is that there is a small stagnation pressure at the tips of the obstructions. If you use the Mechanical Energy Balance, you'll find that where the kinetic energy goes down the pressure goes up. This is smoothed by the viscous dissipation, of course. But at the tip, the velocity goes from something to zero, so the pressure goes up slightly. If the flow rate were decreased so that the Reynolds number was about 1, there would be very little pressure increase at the tip because the inertial effects would be so small.

To assess the accuracy, the mesh was refined once and the problem resolved. This was done again. The results are shown in the table. Clearly the mass balance is even better as the mesh is refined, and the inlet pressure is almost the same. The only thing that changes is the peak pressure. Thus, if that were of concern, you need to use the refined mesh. For that finest mesh, the mesh is shown in the figure. Notice that the elements are very small near the obstructions.

	No. elements	degrees of freedom	Flow rate in	Flow rate out	Inlet pressure	Peak pressure
mesh1	670	3363	0.429684	0.429698	9.773196	11.3
mesh2	2680	12753	0.429681	0.429676	9.775313	12.8
mesh3	10720	49623				12.91



mesh with 670 elements



mesh with 10,720 elements