McGill

ASSIGNMENT 3

TOPIC
Flow of viscous fluid between two parallel porous
plates with bottom injection and top suction using CFD
method with the help of COMSOL.

Vraj Bharatbhai Patel
McGill ID: 261022581
CHEE 631: Foundation of Fluid Mechanics

A. Significance of the geometry.

For this assignment I've chosen to do simulation of flow of viscous fluid between two parallel porous plates with bottom injection and top suction. The two dimensional steady laminar flow in channels with porous walls has numerous application in field of Science and Engineering through boundary layer control, transpiration cooling and biomedical engineering. This type of flow is very common in human body where this study is useful in drug transport to the nervous system, where we're dealing with this kind of flow and conditions.

B. The boundary conditions and assumptions.

Below are the assumptions that was taken during the study of the flow:

- Fully developed flow
- Laminar flow
- Reynolds number is very low
- Incompressible viscous flow
- Newtonian flow
- Uniform vertical flow

Boundary conditions:

For my simulations I've taken free and porous media flow in physics study and made the upper and lower boundary of the geometry as the leaking boundary where I've decided to take horizontal component as zero (no slip condition in x direction). Since this is the condition of the bottom injection and top suction, I've taken bottom plate's vertical velocity higher than top plate's vertical velocity. I've taken numerous approaches to solve this problem but there was only one way to solve this simulation problem. For this problem the boundary conditions are the most important because for some scenario the problem was not easy to converge because the problem is defined in a very low Reynolds number and in order to do experiments at low Reynolds number I choose to do simulation at very low velocity of the fluid.

I chose liquid water as the viscous fluid under the study. For the flow in the x- direction, velocity u is the function of y only. The flow is fully developed so, the velocity inside the two parallel plates is the average velocity. The inside the plates is pressure driven flow due to pressure gradient in the x direction there is a induction of flow in the x direction. The velocity component in the y direction is taken as constant only the x direction velocity is variable, and it changes with respected to y co-ordinates.

C.   Mesh independency based on either relative global error or a parameter reported

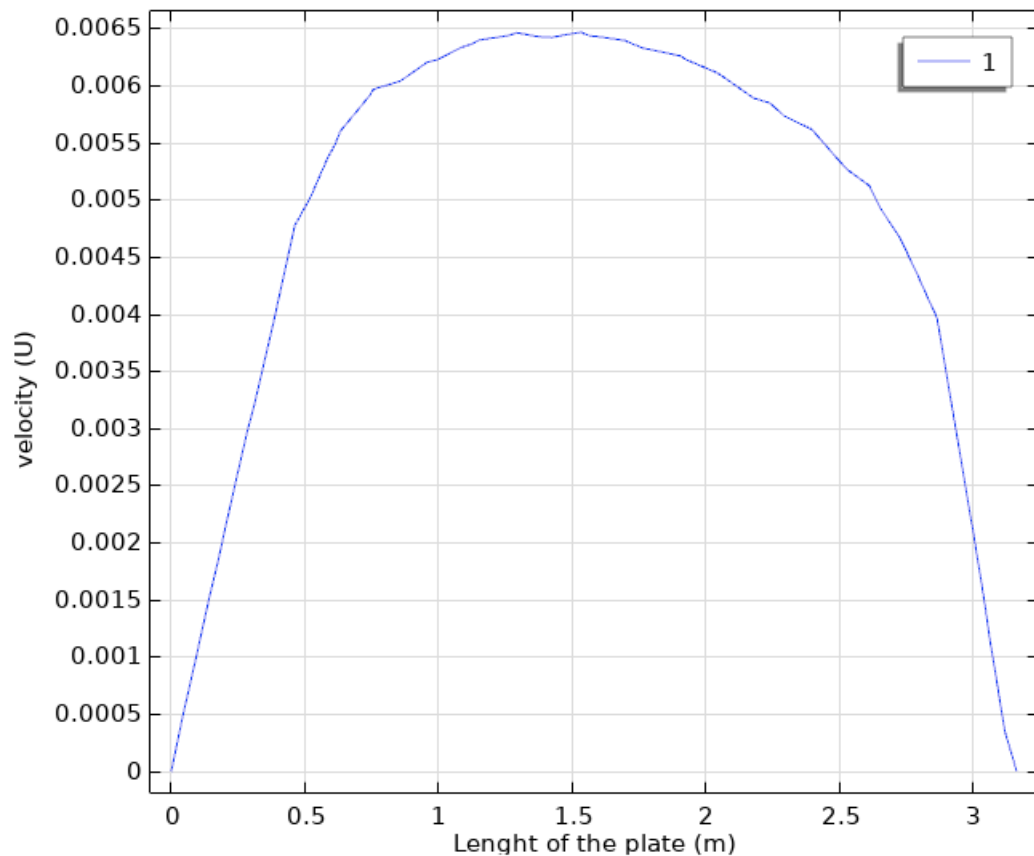For my simulation of the geometry, I tried for different mesh sizes and below are my results.

1. Coarser:

For the coarser mesh size my solution did converge but the error in my solution very high and variance is more.
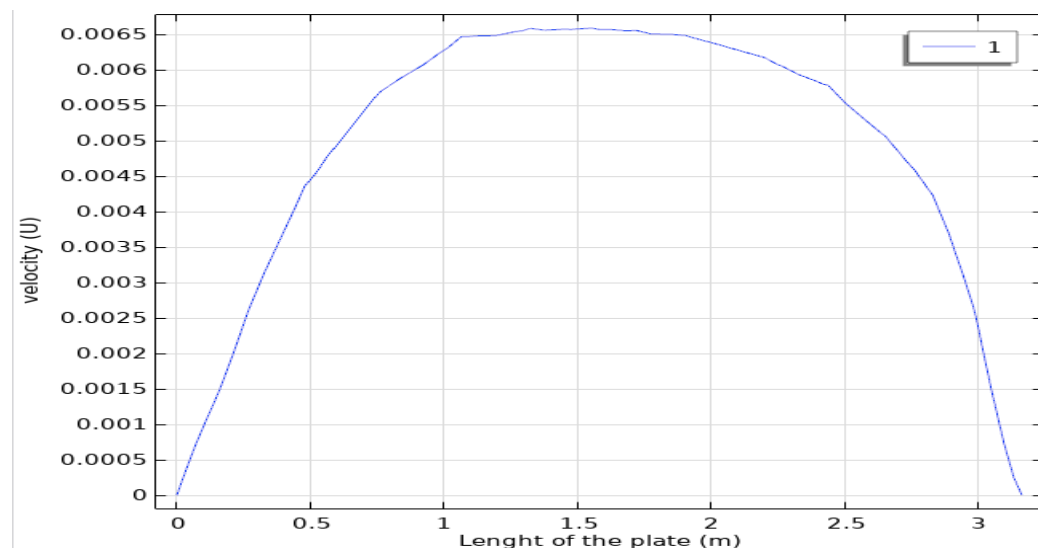
2. Coarse:

In this mesh size the solution also converges but variance is also high, the result is as same as that of coarser mesh size.


3. Normal and fine:

In this mesh size the problem was failed to converge, so in order to find the better solution I increased the mesh size but for that size also the solution didn't converge.

4. **Finer:**

This mesh size gives the perfect solution of the problem and also from the plot we can say that the graph has very low variance so we can say that finer is the best mesh if we increase the mesh size to extra fine then also the solution would be same as that of finer but the computational time and memory increases, so finally I chose finer mesh size for my simulation.
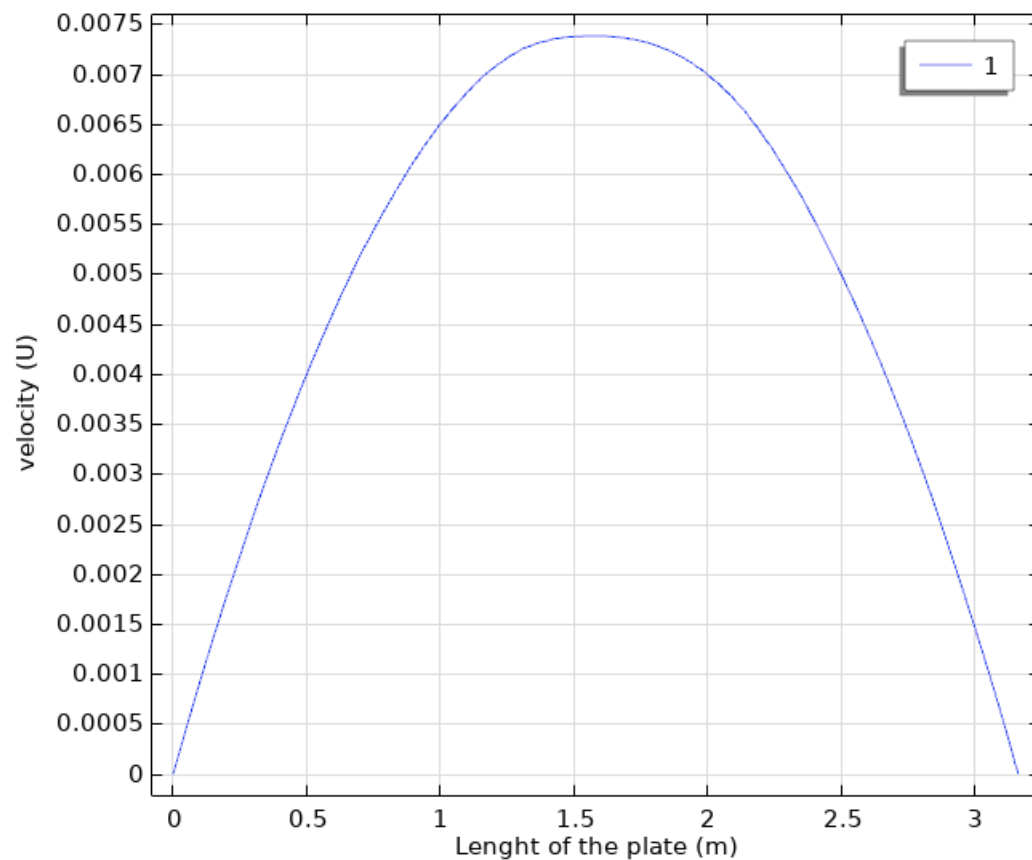
D. Results of the comparison to experimental data or closed form data:

In the research paper from which I've done the simulation, it has a closed form solution which was derived by applying the continuity, momentum and Navier stokes equation with some initial assumptions. The result shows the relationship between velocity in the x direction and how it changes with the distance between two plates.

$$\frac{u(y)}{u_{max}} = \frac{2}{Re}\left[\frac{y}{h} - 1 + 2\frac{1 - e^{-Re\left(1-\frac{y}{h}\right)}}{1 - e^{-2Re}}\right]$$

Here, Re is the Reynolds number which is defined by,

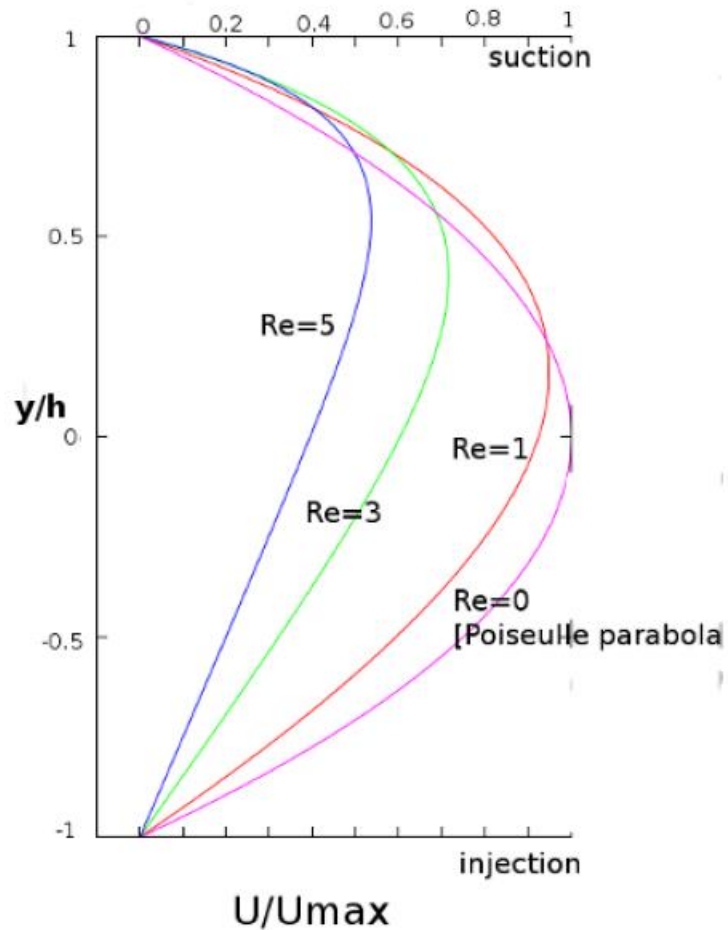$$Re = \frac{\rho vh}{\mu}$$

Where,
ρ = density of the fluid
v = velocity component in the y direction
h = distance between two porous plates
μ = viscosity of the fluid

The experimental data with different Reynolds number shows the below graph which is taken from the paper.

Simulation results:

In the simulation, I tried for different velocities as a boundary condition but it was really difficult to find the right velocity which can give an actual satisfying solution to the problem. The ill conditioned boundary conditions doesn't converge to the solution.
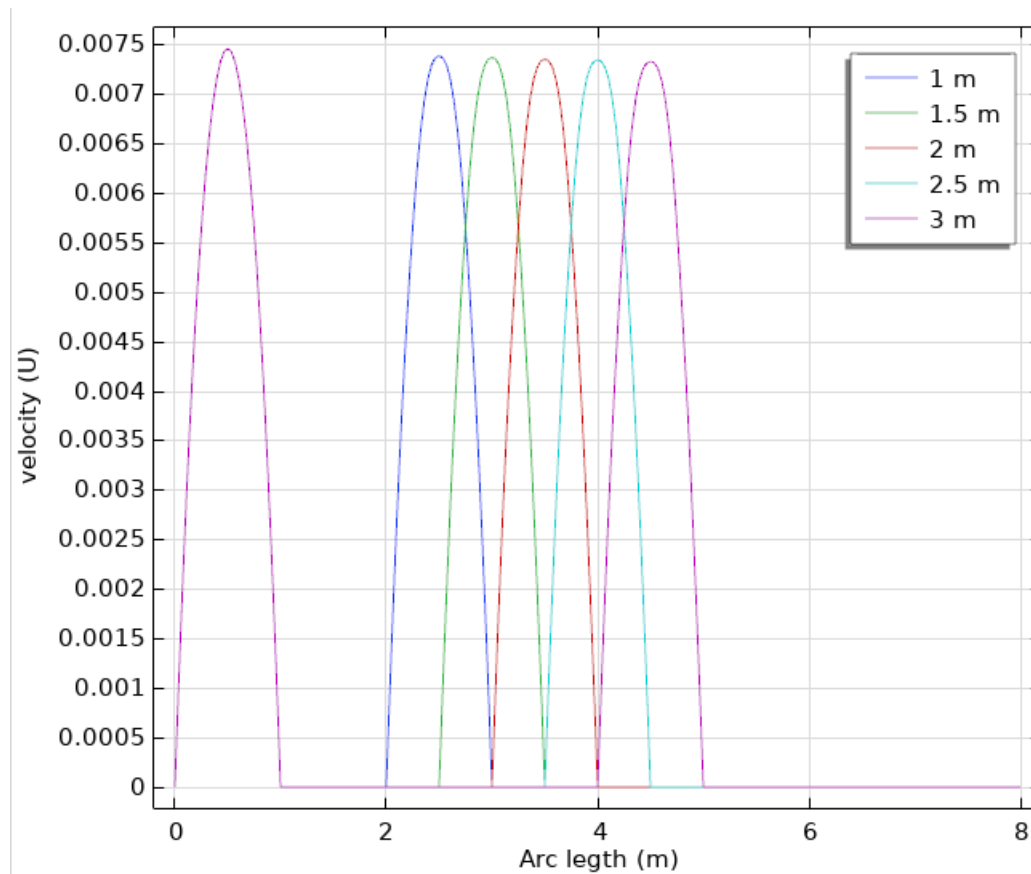
Since it was difficult to find the solution of the problem, in my simulation to change the velocity was really difficult so I changed the length of the porous plates to carrying out the experiments for different Reynolds number.

Comparison:

If we compare the actual results with the simulated result, we can see that the simulated result is no where nearer to the actual results, the actual result has the closed form solution which has something similar to

parabolic velocity profile and from the result it self we can distinguish the difference between the injection and suction but from the actual simulation we can't tell a difference between those two source and sink. Also the simulation result has symmetric velocity profile which shows that there is very low effect of source and sink which affects the flow between two porous plates. The main reason for this is the very low Reynolds number. For some values of velocities solution failed to converged, the main issue for me for this simulation was to come with a boundary conditions which converge as well as give some results but for those values the simulated solution doesn't really fit with the actual results.

E.   Discussion of flow physics and potential differences between the simulation and experimental and/or closed form solution.

Here, in this flow we have two different types of velocity one is in the direction of x and another is uniform cross flow velocity. The flow in the x direction is pressure driven also at the porous plates we've no slip condition. The flow is assumed to be steady state. The fluid is incompressible and viscous which is passing through rectangular cross section.

In actual simulation due to very low values of velocities it was really difficult to find the solution. COMSOL uses iterative methods to find the solutions meanwhile, in the closed form solution we apply mathematical fundamentals to solve the problems. So, lesser values of parameters in the COMSOL can affect convergence of the solution. Also, we need to know more about boundary conditions which really affects the solutions in the COMSOL. In my research paper there we no numerical values for the parameters were given. So, I need to come up with the different parameters values and see how this affects the results for this problem.

In conclusion, simulated result fails to get the same kind of result as that of in the actual result. And there are numerous reason for that. As per my

observation, in simulations, boundary conditions are the most important parameter to consider because under-defining or over-defining the boundary conditions affects convergence results.

Extra:

I've also tried one another geometry for the same problem. Where instead of using the leaky boundary I have putted porous medium at the wall and I'm passing my fluid from that porous medium. I'm attaching that file with my submission file. It's called '**porous simulation**'.

References:

1) http://www.ptep-online.com/2014/PP-36-14.PDF