Comsol Guide Stenosis Exercise

Laboratory Exercise in Computational Fluid Dynamics

- Launch Comsol Multiphysics. (If needed right click on the icon and run as administrator)
- Specify Space dimension
- Choose **Fluid flow**→ **Single-phase flow**→ **Laminar flow (spf)** -> then click the forward blue arrow
- Choose **Stationary study ->** then click the finish line flag
- Load the geometry: click on the (Geometry 1) tab and select import. (select the .dxf file option -> Browse-locate and select the geometry file you will work with)
  - ➢ Important!!!! In the import options tab: Chose form solids

  After specifying these options click build all.

- Click on the laminar flow tab and change the problem into the "**incompressible**" mode
- Click on the fluid properties tab and input the density and viscosity values after changing the input in to "user defined" instead of "from material"
- Right click laminar Flow tab and add one inlet and one outlet + define the vertices that correspond to each one of them by choosing them and clinking the (+) plus button . The inlet is the one closest to the stenosis.
- Define the values in the inlet and the outlet. Inlet is a velocity field and the outlet is pressure, no viscous stress.
- Click on the mesh tab and use the default values to generate the computational mesh. -> Click build all
- Click on the study tab and the click on the equation button to launch the solver.
- After the solver finished you will automatically see some results 1) the velocity magnitude surface plot and 2) the pressure contours plot.

  Proceed with the post processing and the question on the exercise sheet.

  (more details be found on the post processing guide)