MNIS – Physical models for micro and nanosystems

Exercise 6: Fluid Flow Between Two Parallel Plates

WHAT ARE WE GOING TO LEARN:

-typical boundary conditions encountered during modeling of fluid dynamics

In this model, we will cover a simple example of fluid flow with a velocity field developing between two parallel plates. The purpose is to study some basic properties of laminar flow.

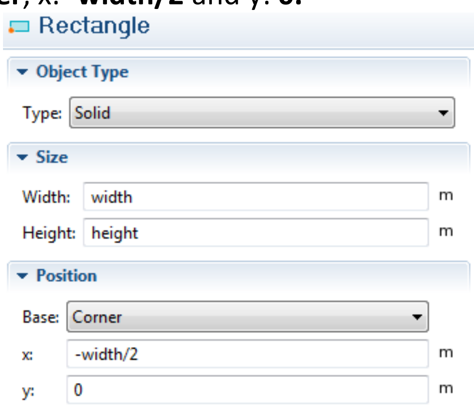
1. BUILDING THE MODEL

We will first start Comsol and define the type of geometry as **2D** and choose the physics package **Fluid Flow ► Single-Phase Flow ► Laminar Flow.** Choose **stationary study.**

We should now build the model. It will consist of a simple rectangle with a parameter-driven geometry and material properties. First, let us define the following parameters:

Name	Expression	Description
density0	1000 [kg/m^3]	
viscosity	0.01 [Pa*s]	
v0	2[cm/s]	
width	2 [cm]	inlet velocity
height	5 [cm]	
Re	(density0*v0*width)/viscosity	Reynolds number

We can build the model which will consist of a simple rectangle by right-clicking on **Component1 ► Geometry** and choosing **rectangle.** This will represent our tube through which the fluid is flowing. Under width add **width,** height should be **height.**
Under position choose **corner**, x: **-width/2** and y: **0.**



We can now proceed to the next steps. First, we must define material properties.

2. DEFINING MATERIAL PROPERTIES

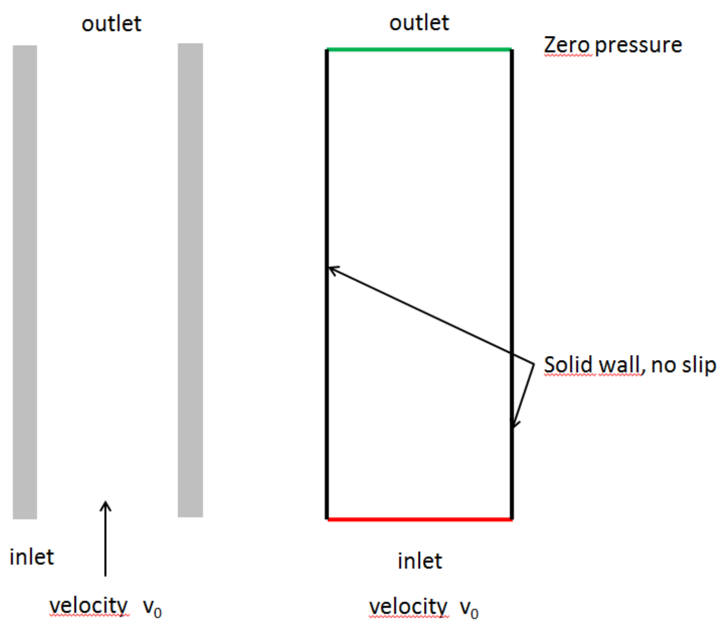
The fluid contained between the walls will be made of a non-standard fictional material with parameter-driven material properties. To do this, right-click on **Model ► Materials** and choose **Blank Material.** Under **material properties** choose **density** and **dynamic viscosity** and add them. Under **material contents** add the parametric values **density0** and **viscosity.**

Material Contents

	Property	Name	Value	Unit	Property group
☑	Density	rho	density0	kg/m³	Basic
☑	Dynamic viscosity	mu	viscosity	Pa·s	Basic

We can now move to the next step and define the boundary conditions. Assign the material you just created to the rectangle.

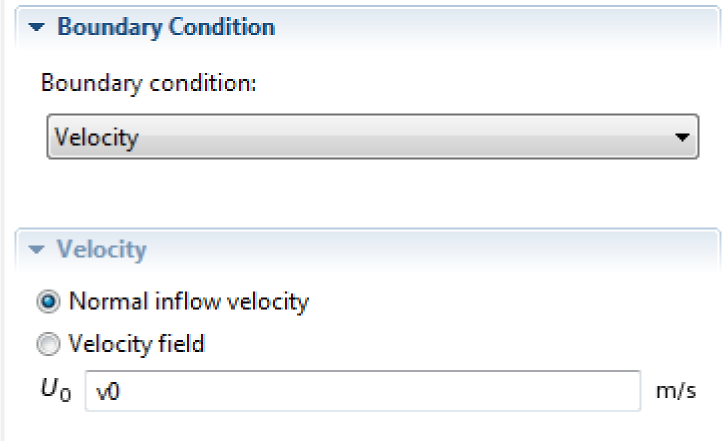
3. DEFINING BOUNDARY CONDITIONS



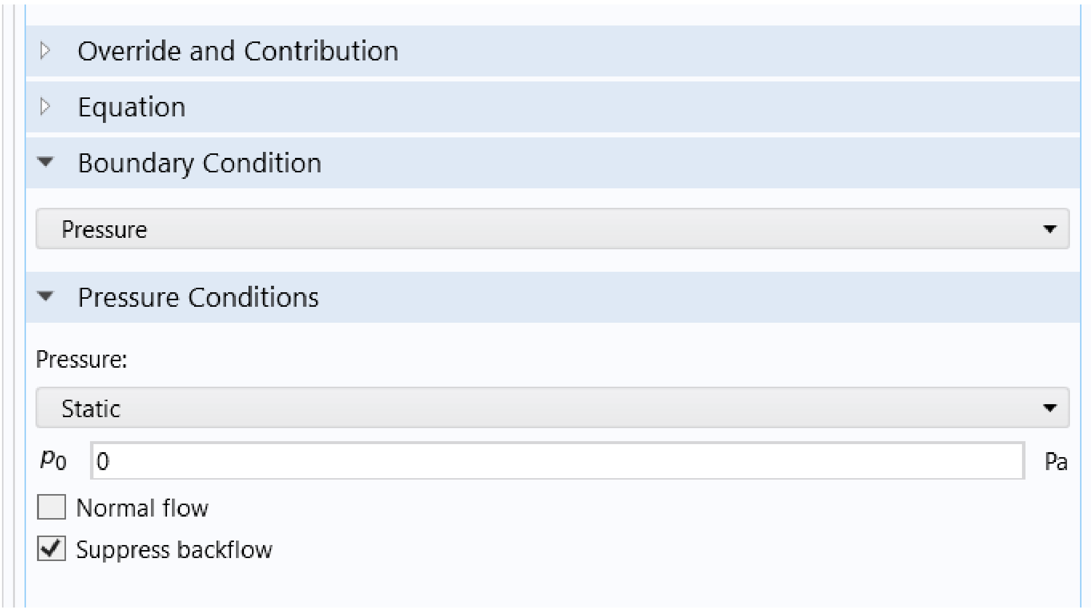
Here, we must define the boundary conditions. By default, Comsol assigns the no-slip solid wall boundary condition to all boundaries. We must define the boundaries that correspond to inlet and outlet of the pipe. Inlet is characterized by a velocity v0 and outlet with the absence of pressure (to be correct, the pressure there should be equal to the atmospheric pressure, but we will disregard that in this simulation).

Right click on **Model ► Laminar flow** and add **inlet** and **outlet.** Inlet should have under

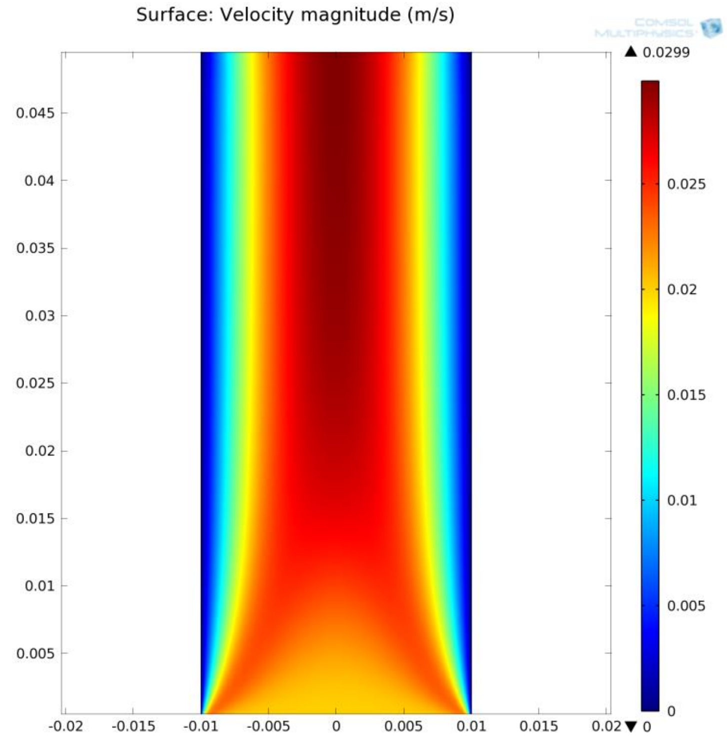
Boundary condition velocity. Add **v0** under **velocity:**



The outlet should have under boundary condition **Pressure, Pressure: static.**

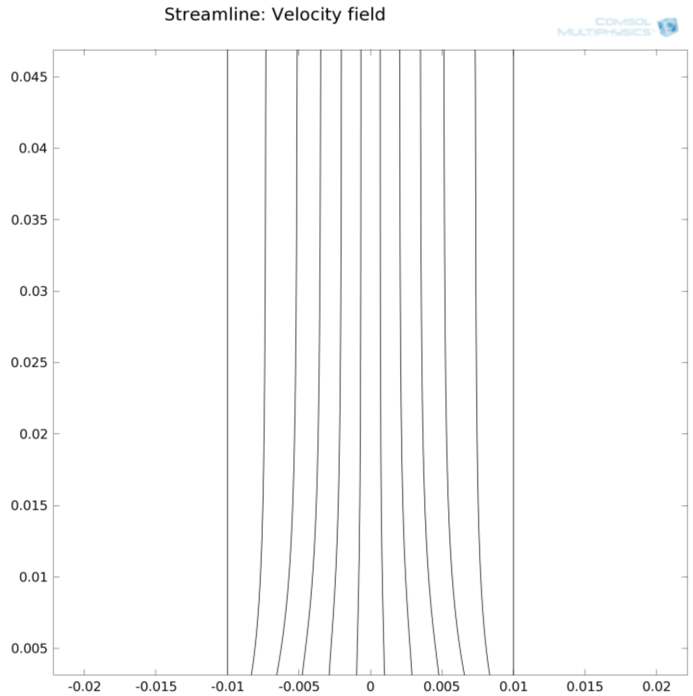


The default mesh size is a bit to coarse for this problem, so you can refine to the element size **finer**. We can now start solve and if everything is OK we should get this kind of velocity distribution:



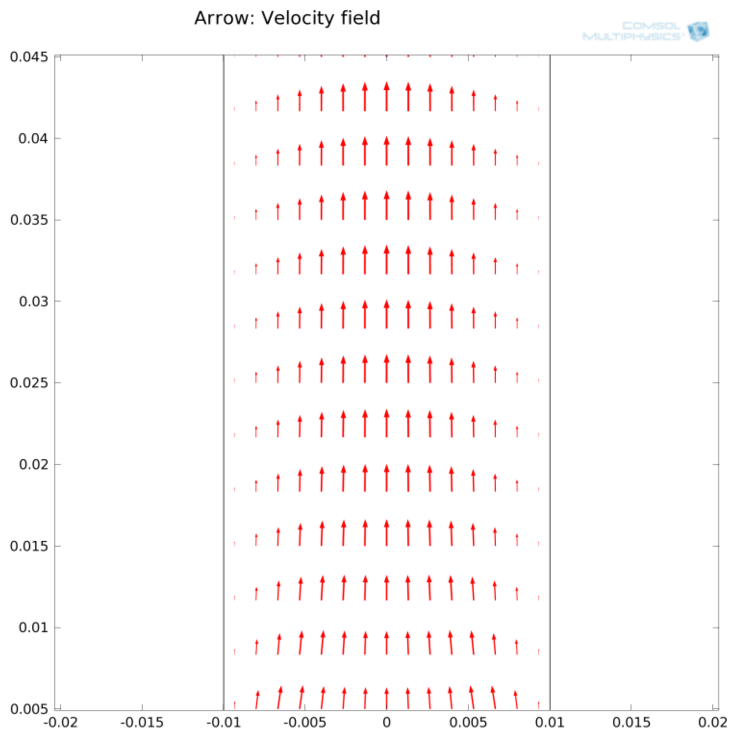
4. PLOTTING RESULTS

We can now look with more attention into some fine details of the result. Let us create two new 2D plot groups. In the first one, create a **streamline** type plot. Select the inlet (boundary 2 in my case) in the **selection** section of **settings**. The result should look something like this:



We can see how the streamlines are parallel close to the outlet, a hallmark of laminar flow.

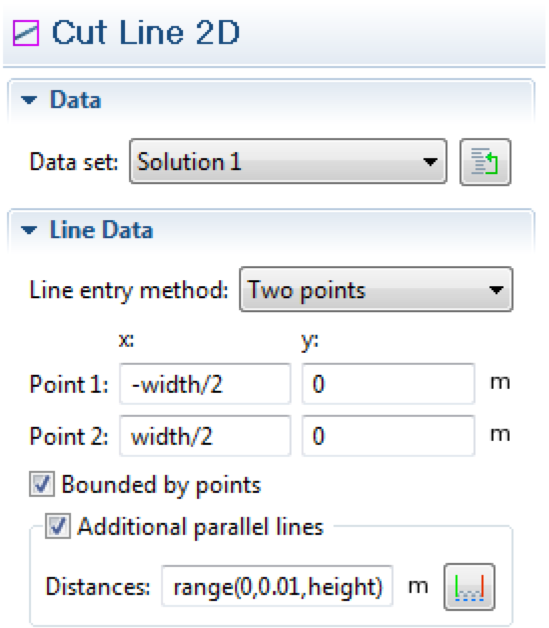
In the second plot group, we can create an **arrow surface** plot. The result, shown on the next figure show a velocity distribution typical of laminar flow in a cylinder, called Poiseuille flow.

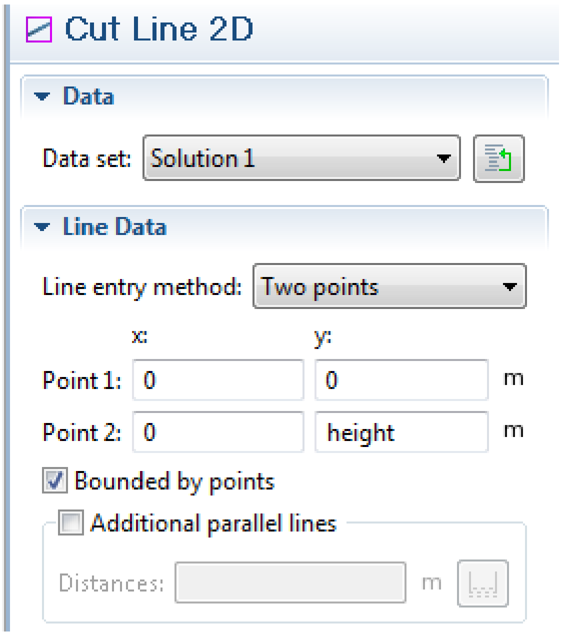


This type of a flow is characterized by velocity = 0 close to the walls (no slip condition) and a parabolic dependence of the velocity as a function of distance from the walls.

It is also interesting to look into line plots of flow velocity across the flow and along the flow direction. Let us create for this two new data sets, both of **cut line 2D** type. The first one consists of an array of lines across the tube, under a 90° angle with the fluid flow, starting at 0 going until height and in steps of 0.01 [(range(0,0.01,height)]. The second one will be a usual line going along the fluid flow.

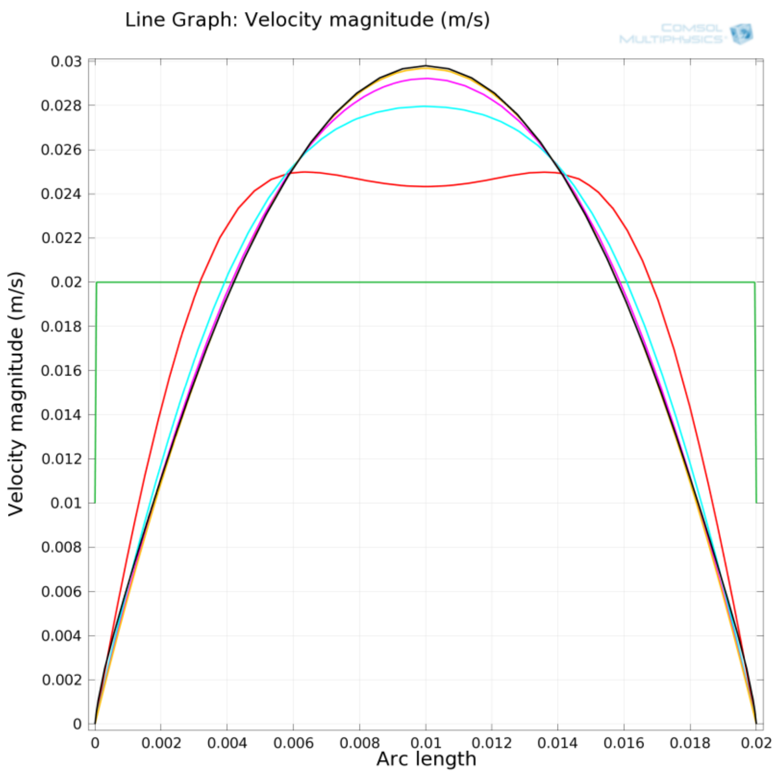
They should have these parameters:





You will also need to create the two new 1D plot groups with the corresponding line graphs.

In the direction perpendicular to the fluid flow, you should get something like this:



As you can see, the velocity profile approaches the parabolic profile as you move away from the inlet. The area under the curves should be the same – this is the consequence of the fact that the liquid is incompressible. You could try repeating the same simulation for different values of the Reynolds number (for example 10x lower and 10x higher). For higher Re, it will take longer for the Poiseuille flow to develop.

This can be also visualized by looking at the plot of the fluid flow velocity along the tube: