MNIS – Physical models for micro and nanosystems

Exercise 7: Fluid Flow Past a Cylinder

WHAT ARE WE GOING TO LEARN:

-time-dependent fluid dynamic modeling

In this model, we will cover a simple example of fluid flow with a velocity field developing behind a cylindrical obstacle.

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 **Time Dependent** study and choose **done.**

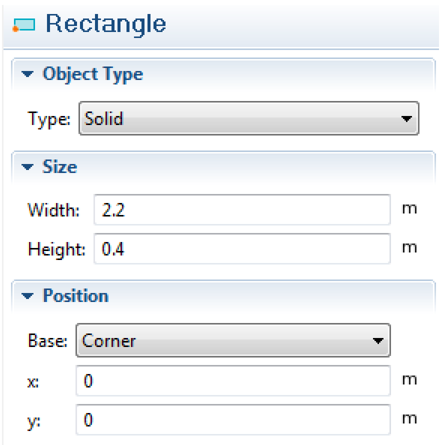
We should now build the model. It will consist of a simple rectangle and a cylinder. Let us first define the following parameters:

Name	**Expression**	**Description**
rho0	1 [kg/m^3]	
mu0	0.001 [Pa*s]	
Umax	0.002[m/s]	Max inlet velocity
width	0.5 [m]	
height	5 [m]	
Re	(rho0*Umax*width)/mu0	Reynolds number

This should result in a Reynolds number Re = 1.

We will also create a step function which will simulate a gradual increase of the inflow velocity. In the Model Builder window, right-click **Global Definitions** and choose **Functions ► Step**. Go to the **Settings** window for Step. Locate the Parameters section. In the **Location** edit field, type 0.1.

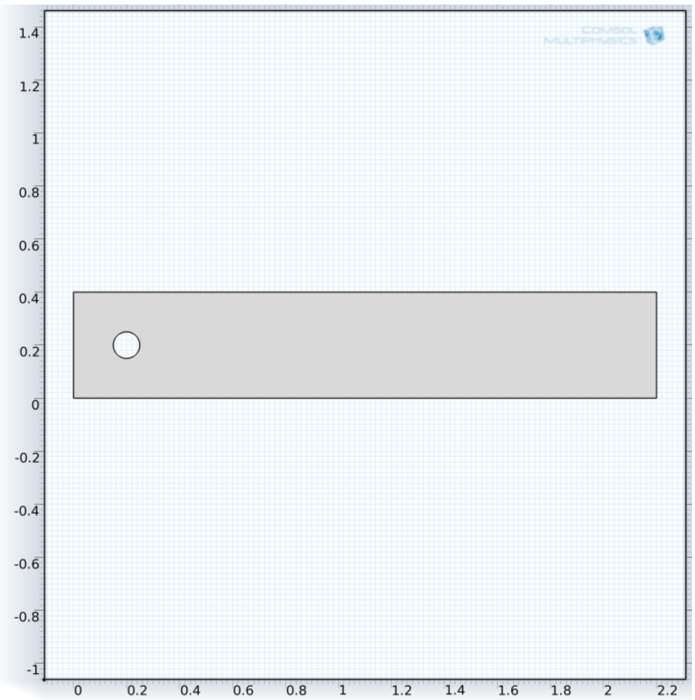
We can now build the model which will consist of a simple rectangle by right clicking on **Component1 ► Geometry1** and choosing **rectangle.** Under width add **2.2,** height should be **0.4.** Under position choose **corner**, x: **0** and y: **0.**



In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**. Go to the **Settings** window for Circle. Locate the **Position** section. In the x edit field, type **0.2**. In the y edit field, enter **0.2**. Locate the **Size and Shape** section. In the **Radius** edit field, type 0.05. Choose **build all.**

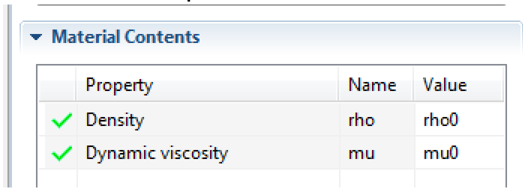
We now need to build the composite object, remove the cylinder which acts as an obstacle from the rectangular fluid flow. Right-click on **Geometry 1,** choose **Booleans and partitions ▸ difference.** Under **settings ▸ objects to add** add the rectangle. Under **objects to substract** add the circle. Choose Build all.

The end result should look like this:



2. DEFINING MATERIAL PROPERTIES

You now need to define the material properties. Right click **Model** ► **Materials** and choose **Blank Material.** Under **material properties** choose **density** and **dynamic viscosity** and add them. Under **material contents** add the parametric values **rho0** and **mu0.**

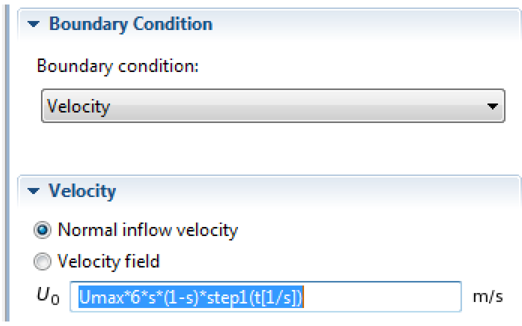


We can now move to the next step and define the boundary conditions. Assign the material you just created to the rectangle with the hole.

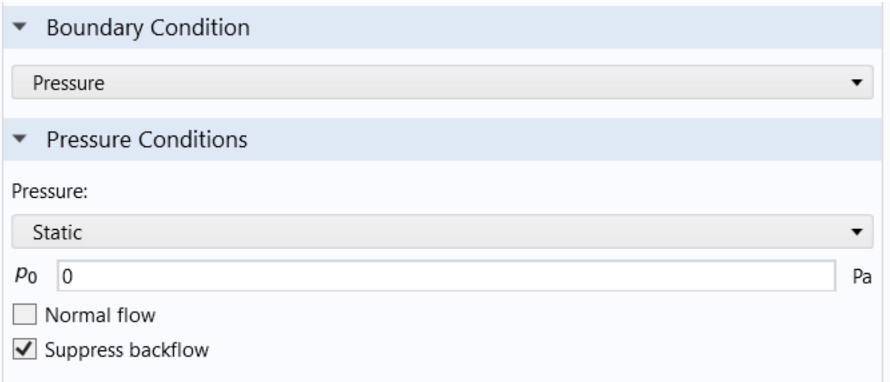
3. DEFINING BOUNDARY CONDITIONS

Here, we must define the boundary conditions. By default, Comsol assigns the no-slip solid wall boundary condition to all boundaries. We have to 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 atmospheric pressure, but we will disregard that in this simulation).

Right click on **Model** ► **Laminar flow** and add **inlet** and **outlet.** First inlet should have under **Boundary condition normal inflow velocity.** Add **Umax*6*s*(1-s)*step1(t[1/s])** under **velocity.** This will make sure that we have the Poisseule flow right at the start of the tube and that the liquid velocity will gradually turn on as a function of time, so that the simulation can adapt at the start:

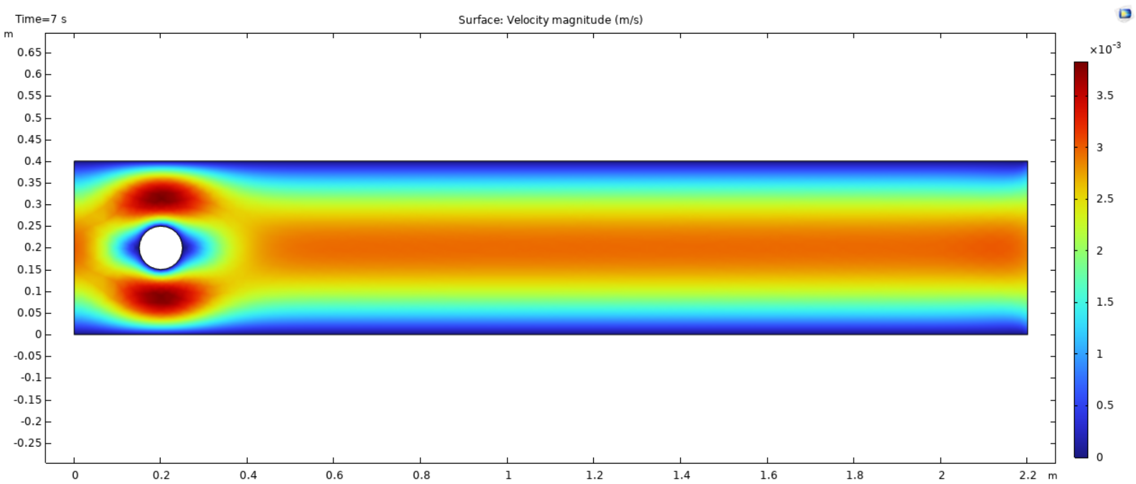


The outlet should have under boundary condition **pressure, static, 0.**



We now need to define the study parameters. In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**. Go to the **Settings** window for Time Dependent. Locate the **Study Settings** section. In the **Times** edit field, type **range(0,0.2,3.4) range(3.5,0.02,7)**. This will make the simulation go faster in the beginning, when nothing interesting is happening and slower once vortices can develop. In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**. Expand the **Study 1 ▸ Solver Configurations** node, then click **Time-Dependent Solver 1**. Go to the Settings window for Time-Dependent Solver. Click to expand the Time Stepping section. From the Steps taken by solver list, select Intermediate. In the Model Builder window, right-click Study 1 and choose Compute.

After some time, you should see the solution that looks something like this (for time=7):



4. PLOTTING RESULTS

We can now also create an animation of the simulation results. Go to **Results ▸ export** and right-click on **animation.** Here, you can tweak various parameters and export the results of your simulation as an .avi or .gif file. To display the animation in the simulation window, add player under **export.** This will animate the results in the graphics window.

Another useful plot is the one where you can trace trajectories of particles that flow with the fluid. To do this, in the **Model Builder** window, right-click **Velocity (spf)** and choose **More Plot Points ▸ Particle Tracing with Mass**. Go to the Settings window for **Particle Tracing with Mass**. Click to expand the **Mass and Velocity** section. In the **Mass** edit field, type **4*pi/3*1e-9**. Find the **Initial velocity** subsection. In the **x component** edit field, type **u**, in the **y component** edit field, type **v**. Locate the **Particle Positioning** section. In the **y** edit field, type **range(0.1,0.05,0.3)**, in the **x** edit field, type 0.

Click to expand the **Release** section. From the Release particles list, select **At intervals**. Select the **Start time** check box. In the associated edit field, type **3.6**. In the **Time between releases** edit field, type **0.4**. Click to expand the **Coloring and Style** section. Find the Line style subsection. From the Type list, select **None**. Find the **Point style** subsection. From the **Type list**, select **Point**. Find the **Point motion** subsection. From the **when particle leaves domain** list, select **Disappear**. Click the Plot button and start the **player.** You could get something like this (for Re>100; for Re=1, particles will not move much because of their low initial velocity):