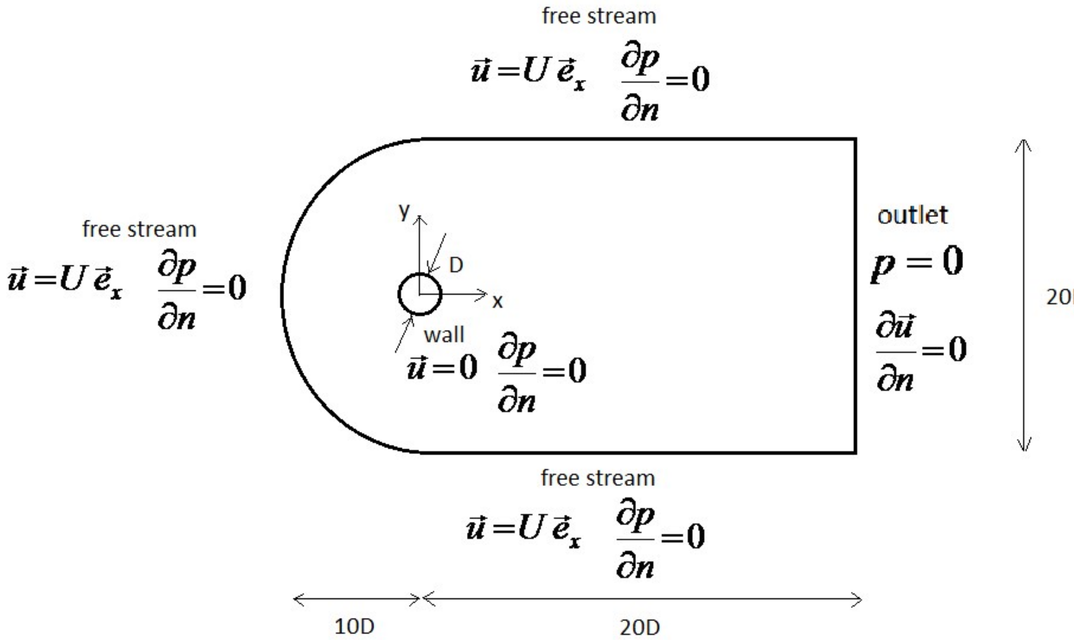
2nd assignment – Unsteady Flow around a Circular Cylinder

In this assignment the goal is solution of the flow around a circular cylinder, with the objective of determining the most efficient methods to capture the vortex shedding phenomenon. A schematic of the geometry and boundary conditions can be seen below.



1) Choose a directory to save the case and the case name.

2) Import the mesh. The mesh used in the assignment is in the GAMBIT format and is named “cylinder.neu”.

3) After importing the mesh use the “Mesh” menu to specify the “empty” boundary conditions, so that the flow is in the xy plane. You should specify also the “patch” boundary conditions. Notice that a selected boundary acquires a soft red color.

4) In the “Setup” menu you should choose the solver. Select a solver for unsteady, incompressible and turbulent flow. The pimpleFoam solver is going to be used for the simulation.

5) In the “Turbulence” menu select laminar flow.

6) In the menu “Transport properties” select the value of kinematic viscosity. According to the speed you intend to use in the free current, taking into account that the cylinder diameter in the mesh file is D = 1m, specify a kinematic viscosity so that Re = UD/ν = 100 + 10 × last digit of your USP number. So if the last digit of your USP number is 9, your Reynolds number is 190.

7) In the “Discretization” menu there are four submenus, for time discretization, spatial discretization (Convection), gradient calculation and interpolation. In the time discretization select the “backward” method. In the spatial discretization the linear

upwind scheme should be selected instead of the default upwind scheme. In the other submenus the default options are ok.

8) In the solution menu there are four submenus, “Solvers”, “Pimple”, “Residuals” and “Relaxation”. In the Pimple algorithm menu, choose 20 “outer correctors” and 1 or 2 “correctors”. If you are using two correctors, you are running a PISO scheme with 20 iterations per time step. If you choose only one “outer corrector” and the “consistent” button is checked, you will be running a SIMPLEC cheme. If you choose only one “outer corrector” and the “consistent” button is unchecked, you will be running the SIMPLE scheme. In the residuals menu lower the residuals to 10^{-5}. In the relaxation menu, use velocity relaxation equal to 0.7. If you have chosen PISO or SIMPLEC, leave 1.0 as pressure relaxation. If you have chosen SIMPLE, use 0.3 for the pressure relaxation.

9) Ignore the “*Operating Conditions*” and “*cell zones*” menus.

10) In the “*Boundary Conditions*” menu the velocity and pressure boundary conditions in the “patch” boundaries should be stipulated. In the free stream boundary the velocity components should be set using the “fixedValue” boundary condition, and a Neumann boundary condition for pressure can be set using the “zeroGradient” boundary condition. In the outlet boundary a “fixedValue” zero pressure should be set associated to a “zeroGradient” boundary condition for the velocity. In the “wall” boundary a zero velocity is specified using the “fixedValue” boundary condition and a “zeroGradient” condition is used for pressure.

11) In the “*Initial Conditions*” menu you should specify the initial guess for the iterative solution of the velocity and pressure fields. Specify the same velocity and pressure values from the free stream.

12) In the “Monitors” menu set the forces monitoring. Choose the boundary where you will calculate the forces (wall) and check the box to calculate the force coefficients. In the mesh, the cylinder has a length of 1 m and a diameter of 1 m, which means that the area used to calculate the non-dimensional drag and lift coefficients is 1 m^2. You should also specify the correct velocity (the same from the free stream). Check the directions of lift and drag forces. The drag force has to be in the direction (1 0 0) and the lift force has to be in the direction (0 1 0).

13) In the "run" menu, in "time control", set the duration of the simulation in seconds in order to have at least 40 cycles of vortex shedding (note that you may need more than that to have the amplitude of the lift force stabilized) . The time-step value is determined by the expected Strouhal number. As for a circular cylinder $St = fD / U \approx 0.2$, UT / D will be 5. So, if you choose a time-step = 0.05 , you will have about 100 time steps per vortex shedding period, which is a reasonable resolution.

14) Still in the “run” menu, under “output”, in the “write control” option set the frequency, in seconds, with which you want to save the solution. You must have at least five files per vortex shedding period to have a good movie. Check the “clean old result files” button if you want to specify how many solution files you want to keep so as not to clutter your hard drive. Remember that it is very easy to fill a hard drive saving a very long transient flow view.

15) Save the case using the floppy disk button and run. Don't worry about high residuals, since for transient problems simFlow only shows the residuals at the beginning of the time step, without graphically recording the drop during the iterations.

16) After the final time for the transient simulation is reached you should post-process your results using paraview in the “Postprocessing” menu. Click the paraview button.

17) In the upper right corner, next to “time”, you will see a button to show the preview movie. Choose to plot velocity magnitude contours and click the button to see the vortex shedding visualization.

18) After visualizing the vortex shedding, close paraView and go to the “postProcessing” folder inside the simulation directory. There will be the folder “forces” that contains the coefficients in the text file “coefficients.dat”. Plot this file with Excel or gnuplot. The 1st column is time, the 3rd is the drag coefficient and the 4th is the lift coefficient. Calculate the Strouhal number. According to your Reynolds number, compare with the formula proposed by Williamson (1989):

$$St=-\frac{3.3265}{\mathrm{Re}}+0.1816+1.6\times10^{-4}\times\mathrm{Re}$$

20) Repeat the simulations and fill in a box similar to the one below, making the combinations of time and space discretization methods. Which combination gives the best result for the Strouhal number?

time	**space**	**Strouhal**
Backward (2nd order)	Linear upwind (2nd order)	
Backward (2nd order)	Upwind (1st order)	
Implicit Euler (1st order)	Linear upwind (2nd order)	
Implicit Euler (1st order)	Upwind (1st order)	