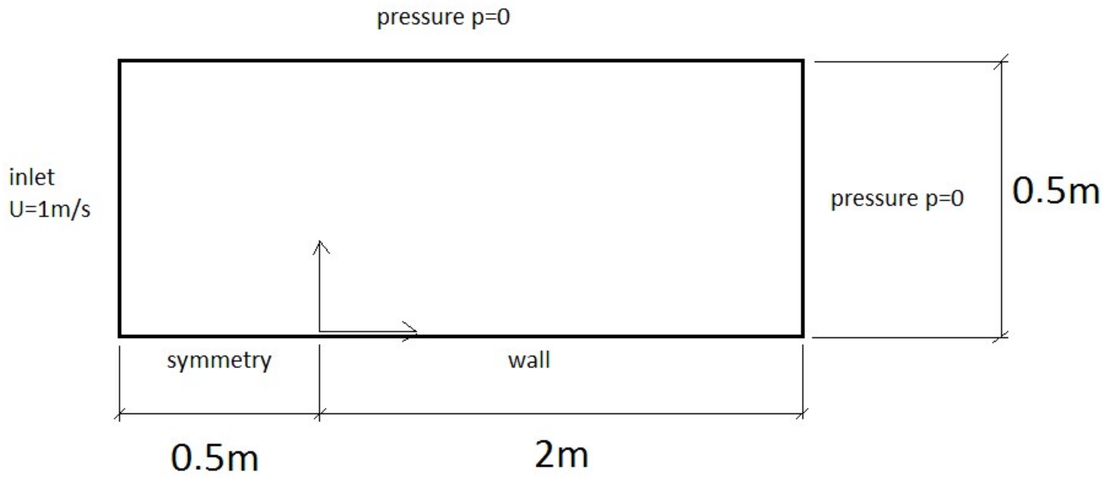
1st assignment – Blasius Laminar Boundary Layer

The assignment is about the solution of the flow on a flat plate parallel to the current (zero pressure gradient), with comparison of the results with the Blasius solution. A scheme of the geometry and boundary conditions can be seen below.



Notice that the plate has a length L = 2m and the velocity of the incident flow is $U = 1\,\mathrm{m/s}$. In this case, the viscosity shall be chosen so that $\mathrm{Re_L}$ = UL/ν = 10000 multiplied by the last digit of your USP number (if it is zero, multiply by ten). As an example if the last digit of your USP number is two, $\mathrm{Re_L}$ = 20000 and ν = 0.0001 m^2/s.

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 “flatPlate.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 “symmetry” and “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 steady, incompressible and turbulent flow. The simpleFoam solver is going to be used for the simulation.

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

6) In the “Transport properties” menu you should specify the kinematic viscosity value for your Reynolds number $\mathrm{Re_L}$.

7) In the “Discretization” menu there are four submenus, for time discretization, spatial discretization (Convection), gradient calculation and interpolation. In the time discretization the simulation should be selected as steady. 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 for solvers, Simple algorithm options, residuals and relaxation. Lower the residuals values to 10^{-5}. If using the consistent SIMPLE algorithm opt for a 0.75 relaxation for pressure and 0.75 relaxation for velocity. The other options can be left with their default selections.

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 "inlet" 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 "pressure" 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) Ignore the "*Monitors*" menu.

13) In the "Run" menu you should specify the maximum number of iterations in the "Time Control" submenu, and the frequency to save files of the solution fields in the "Output" submenu.

14) Save the case using the floppy disk button and run.

15) After convergence you should post-process your results using paraview in the "Postprocessing" menu. Click the paraview button.

18) On the upper right side, in "time", choose the iteration for wich you want the flow visualization.

19) Click the "*apply*" button.

20) In the "*filters*" menu, in "*alphabetical*", choose "*plot over line*".

21) Select a line between the points (1 , 0 , 0) and (1 , 0.1 to 0.2 , 0). Notice that for this x position $Re_x=Re_L/2$. The ideal *y* coordinate for the 2º point should be set according to the boundary layer thickness.

22) If all went well... you will see the boundary layer velocity profile. In the "*file*" menu, in "*save data*", the profile data can be saved to a text file that can be opened using Excel.

You can recall that the Blasius solution can be found in the literature (as an example, in "Fluid Mechanics" from Frank White) . The Blasius solution is given by a $f'(\eta)$ function, where $f'=u/U$ and $\eta = y\sqrt{\frac{U}{\nu x}}$.

Write a small report Comparing your results with the $f' \times \eta$ profile from Blasius. Compare also the boundary layer thickness from your solution, given by the y coordinate where u=0.99U with the Blasius solution:

$$\frac{\delta}{x} = \frac{5}{\sqrt{\mathrm{Re}_x}}$$

Below you can see the Blasius profile results:

η=y*((U / (ν x))^1/2)	u/U
0.0	0,00000
0,2	0,06641
0.4	0,13277
0.6	0.19894
0,8	0,26471
1.0	0,32979
1,2	0.39378
1.4	0,45627
1.6	0,51676
1,8	0.57477
2.0	0,62977
2,2	0.68132
2.4	0,72899
2,6	0.77246
2,8	0,81152
3.0	0,84605
3,2	0,87609
3,4	0,90177
3,6	0,92333
3.8	0,94112
4.0	0,95552
4,2	0,96696
4,4	0,97587
4.6	0,98269
4,8	0,98779
5.0	0,99155
∞	1