**5th assignment – Turbulent Flat Plate with viscous sublayer solution**

This assignment is about the solution of the flow over a flat plate parallel to the current, comparing the results with the expression of the viscous sublayer and logarithmic law for a smooth plate. A schematic of the geometry and boundary conditions can be seen below.
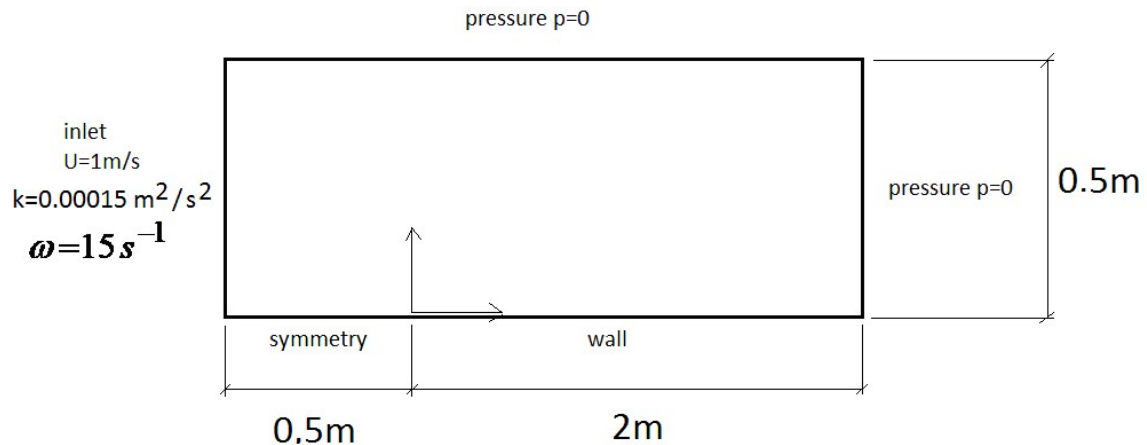


Fig. (1): Flow domain and boundary conditions.

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 "flatPlateViscousSublayer.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", the "wall" 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 turbulent flow. Choose the RANS $k$-$\omega$ SST.

6) In the "Transport properties" you should set the value of the kinematic viscosity. Set $\nu = 1 \times 10^{-6} \text{m}^2/\text{s}$, so that $\text{Re}_x = 1 \times 10^6$ for x=1m. This $x$ coordinate is equivalent to half the length of the plate.

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 for the velocity instead of the default upwind scheme. The turbulent quantities can be solved with the 1st order 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 1.0 relaxation for pressure and 0.3 relaxation for velocity. Set a 0.3 relaxation for the turbulent quantities. 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. Use the boundary conditions in Fig. (1) for $k$ and $\omega$.

In the "pressure" boundary a "fixedValue" zero pressure should be set associated to a "zeroGradient" boundary condition for the velocity, $k$ and $\omega$.

In the "wall" boundary a zero velocity is specified using the "fixedValue" boundary condition and a "zeroGradient" condition is used for pressure. Don't use openFOAM's no-slip condition for velocity (although in theory that's right, it doesn't work well). For $\omega$ and $\nu_t$ use the "standard wall function" boundary condition. Set $k = 0$ using the "fixedValue" boundary condition on the wall, because the viscous sublayer is being solved and you do not need a wall function for $k$.

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, pressure, $k$ and $\omega$ values from the free stream. The $\nu_t$ viscosity can be set to zero.

12) Ignore the "*Monitors*" menu.

13) In the "Run" menu you should specify the maximum number of iterations (around 10000) 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 can check the y+ of cells along the walls, which is important when using turbulence modeling. Use the "calculate" menu, "wall" option to check if y+ < 5, which is usual when using the $k$-$\omega$ model to solve the viscous sublayer.

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

17) On the upper right side, in "time", choose the iteration for wich you want the flow visualization. Click the "*apply*" button.

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

19) Choose a line between the points (1,0,0) and (1,0.03,0) and plot the $x$ velocity profile. A velocity profile should be obtained for comparison with the viscous sublayer and logarithmic layer expressions:

$$U^+ = y^+ \text{ para } y^+ < 5$$

$$U^+ = \frac{1}{\kappa} \ln y^+ + B \quad \kappa = 0.41 \quad B = 5 \text{ para } y^+ > 30$$

Save the $x$ velocity profile to a text file. The shear velocity is going to be obtained through the calculation of the wall shear stress in a further step.

20) Momentarily close Paraview. Go to the menu "calculate", option "wall" and calculate the "wall shear stress".

21) Open paraview again. Select the Wall instead of the internal mesh to visualize results. Remember to select the stress on the wall (wallShearStress) to be loaded.

22) On a line between the points 0.01,0,0 and 2,0,0, plot and save the wall stress data in the $x$ direction and the shear stress modulus. The stress in the $x$ direction will be negative, because of the way OpenFOAM calculates it. Use the module in the calculations that follow.

23) Use the stress at point (1,0,0) to calculate the shear velocity to plot the velocity profile for comparison with the viscous sublayer and log layer in step (19). Remember that:

$$u^* = \sqrt{\frac{\tau_o}{\rho}} \quad , \text{ where } \rho = 1 \, kg/m^3$$

24) Using excel or any other software calculate the friction coefficient $c_f = \dfrac{\tau_o}{\frac{1}{2}\rho U^2}$ as a function of $Re_x = \dfrac{U x}{v}$. Compare your results with the classical results from the Von Kármán integral equation for a 1/7 profile:

$$c_f = \frac{0.059}{Re_x^{0.2}}$$