4th assignment – Solution of the flow through a backward facing step

This assignment is about the flow solution in a backward facing step, characterized by an adverse pressure gradient. The simulation must be carried out with the standard k-ω model from Wilcox. The Reynolds number based in the H height of the step is $U_o H/\nu = 44000$. The turbulence intensity at the domain inlet is I=3.9% and the turbulence length scale is L=0.07*2H. Using U_o = 1m/s and H = 1m, the turbulence kinetic energy is k = 0.0022 m^2/s^2 and the specific dissipation is ω = 0.62 s^{-1} at the inlet. The kinematic viscosity is ν = 0.0000227 m^2/s.

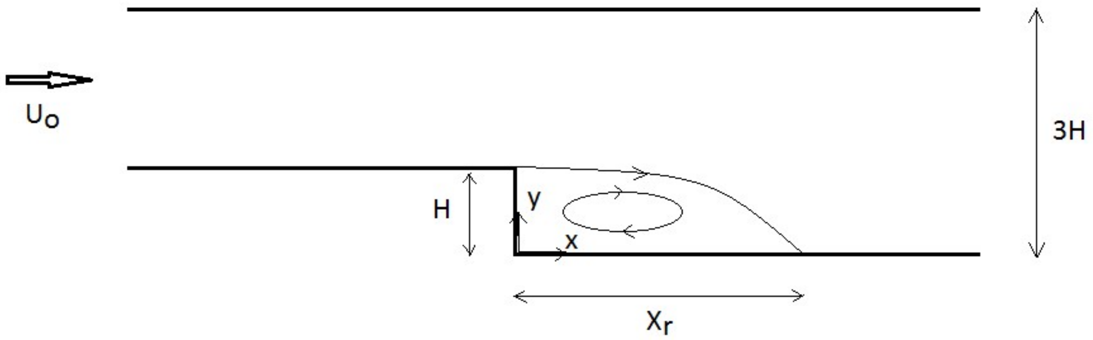

Fig.1) Flow domain description.

It is expected to obtain a length of the recirculation region $X_r \cong 7H$. In addition to verifying the length of the recirculation region, the velocity profiles obtained in the simulations for x/H = 5.33 must be compared with the experimental results from the Figure below, obtained from http://ta.twi.tudelft.nl/isnas/report94-24/node8.html.

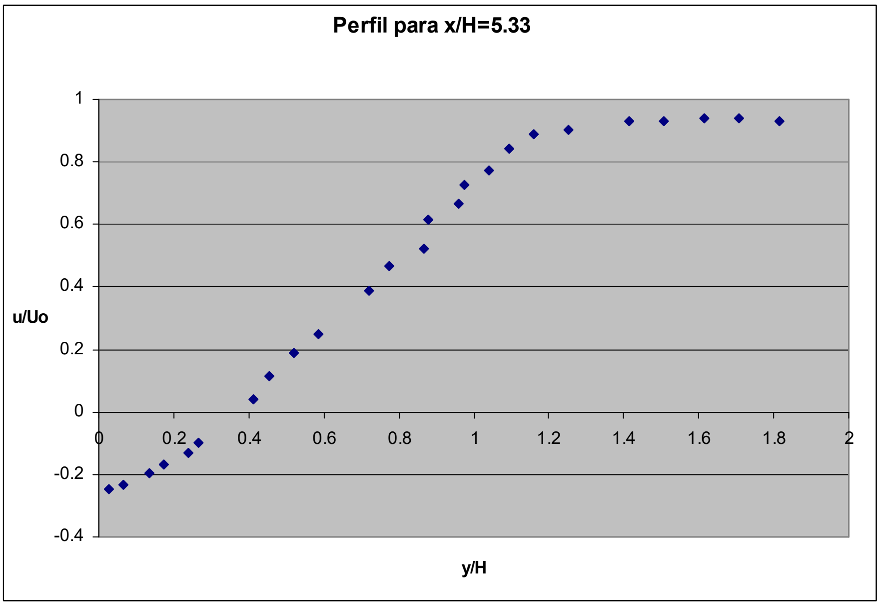

Fig.2) Experimental velocity profile for x/H = 5.33. From http://ta.twi.tudelft.nl/isnas/report94-24/node8.html.

A mesh file in .neu format and an Excel spreadsheet with the data from the graph in Fig.2 will be provided..

Roadmap for the simulation:

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 “backStep.neu”. In this mesh H=1m.

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 “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 standard k-ω turbulence model.

6) In the “Transport properties” you should set the value of the kinematic viscosity. Set ν=0.0000227 m^2s, so that Re_H = 44000.

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 1^{st} 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 previously discussed for k and ω.

In the “outlet” boundary a “fixedValue” zero pressure should be set associated to a “zeroGradient” boundary condition for the velocity, k and ω.

In the “wall” boundary a zero velocity is specified using the “fixedValue” boundary condition and a “zeroGradient” condition is used for pressure. Do not use openFOAM's no-slip condition for velocity (although in theory that's right, it does not work well). For k, ω and ν_t use the “standard wall function” boundary condition.

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, k and ω values from the inlet. The ν_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, go to the “calculate” menu and use the “wall” option to calculate the wall shear stress on the walls.

16) 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 (5.33,0,0) e (5.33,2,0) and plot the x velocity profile. Save the data to compare this velocity profile with the experimental profile from the excel spreadsheet.

20) close paraview and call it again.

21) Choose to visualize the results on the wall right after the “step”.

22) In the “*filters*” menu, in “*alphabetical*”, choose “*plot over line*”.

23) Choose a line between the points (0,0,0) e (50,0,0) and plot the x component of the wall shear stress. Locate the end of the separation bubble, that is, the point where the shear stress equals zero.

24) Save the profile of the wall shear stress to accurately locate the end of the separation bubble. Compare the result with the experimental data ($X_r \cong 7H$).