

Open Source CFD course
Mid-term examination
Date: 2th April 2014

Name: _____
NEPTUN: _____
Signature: _____

1. Simulate a turbulent backward facing step flow using stationary Newtonian fluid approach and $k-\epsilon$ turbulence model. Geometry is shown in Figure 1 ($h = 0.1$ m, $L_c = 4h$, $L_s = 14h$, $D = 2h$). Construct a mesh by 4 blocks in blockMesh. The grading should be prepared with a 1.025 cell growth ratio, and 4 mm size of the wall adjacent cell in wall normal direction. Apply the fixed values of $U = 10$ m/s, $k = 0.375$ m²/s² and $\epsilon = 14.855$ m²/s³ at the inlet!

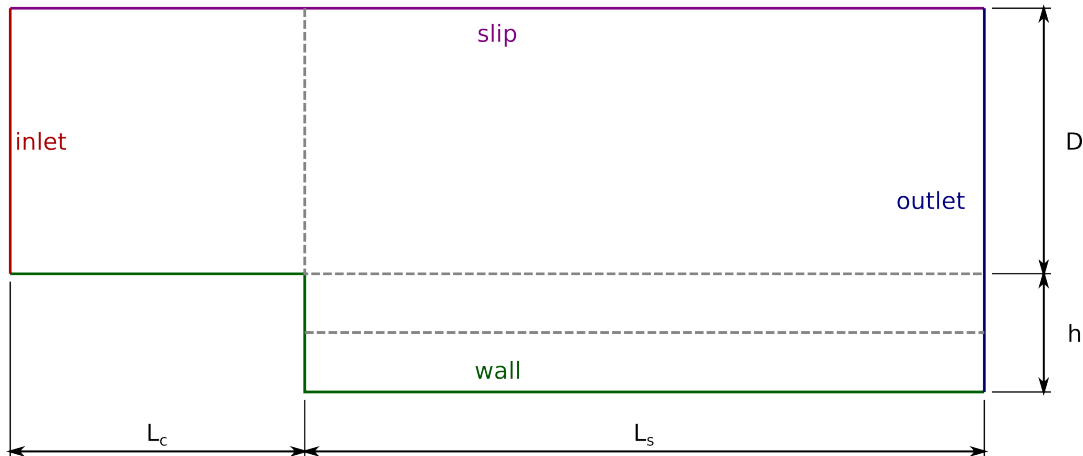


Figure 1. The geometry of the domain

Help: you can calculate number of cells and simpleGrading first/last ratios using these formula:

$$L = l_1 \frac{q^n - 1}{q - 1} \quad \text{and} \quad \log_y(x) = \frac{\ln(x)}{\ln(y)}$$

Visualize in paraFoam or gnuplot and save images in PNG file format:

- a.) mesh ___ / 10 p b.) convergence ___ / 10 p c.) velocity magnitude ___ / 10 p
d.) streamlines ___ / 10 p e.) vorticity ___ / 10 p

Show results to teachers when you are ready with these tasks!

Total: ___ / 50 p

Signature of teacher: _____

2. Where can you find simpleFoam.C?

___ / 5 p

3. What is the symbolic form of the basic equation that is solved in laplacianFoam.C?

___ / 5 p

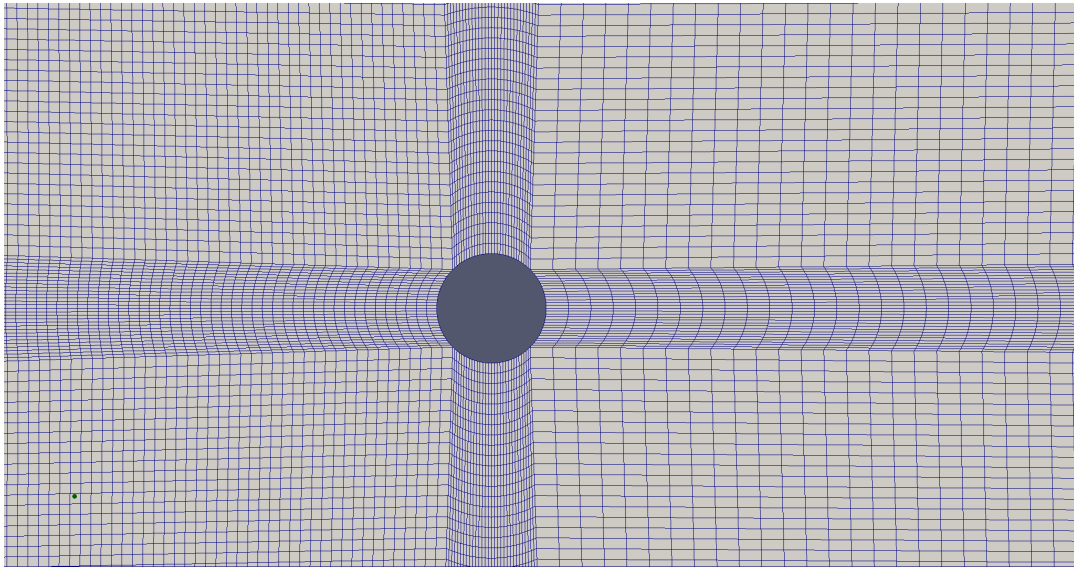
4. How does the Courant number change if you increase number of cells to double and increase time step to double?

___ / 5 p

5. If you study a flow and want to reduce the Reynolds number to the half, what do you change? List 3 possibilities!

___ / 5 p

6. The mesh below is not really good for CFD. How could you improve the mesh without significant cell number enhancement?



___ / 10 p

7. What are the dimensions of field epsilon in the simpleFoam tutorial called pitzDaily?

___ / 5 p

8. The cavity tutorial uses PCG solver (preconditioned conjugate gradient) to solve pressure equation. What other solvers can solve the pressure equation in OpenFOAM?

___ / 5 p

9. If your computer is limited in hard disk space, what setting should you change (in which file) to have less disk space used but still save at least one converged time step?

___ / 5 p

10. Suppose you have a temperature field in your OpenFOAM case named as T. How do you calculate the average of the temperature over the patch "outlet"?

___ / 5 p

Total: ___ / 100 p

Signature of teacher: _____