Numerical Fluid Mechanics II

Summer Semester 2018

DELIVERABLE TASK I: Laminar flow over a square cylinder

Given: Friday, 20/04/2018
Deadline: 17/05/2018

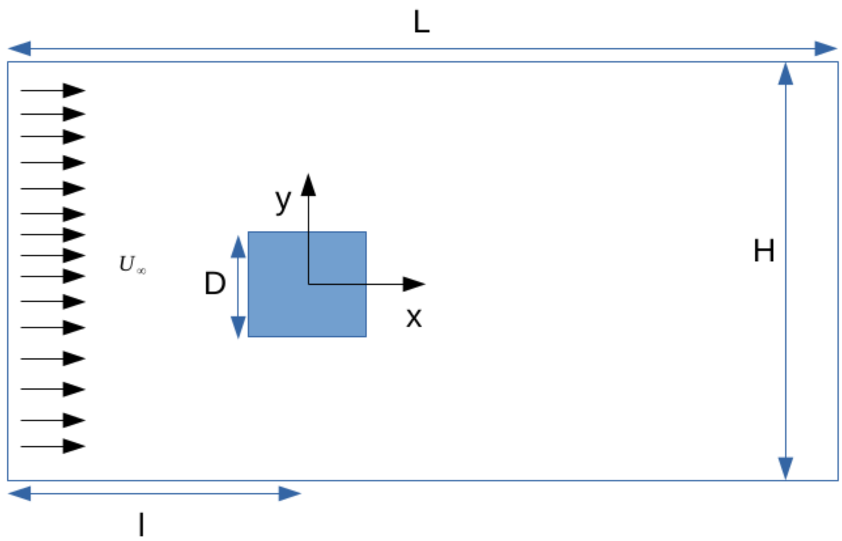
Institute of Fluid Mechanics
Department of Biochemical Engineering, Technical Faculty
Friedrich-Alexander University Erlangen-Nuremberg

Md. Ashfaqul Bari, Dr. Manuel Münsch

Simualtion of laminar flow over a square cylinder

The objective of the first deliverable task is to simulate the laminar flow over a square cylinder by using PIMPLE method with OpenFOAM (pimleFoam solver). Pimple algorithm is a combination of SIMPLE and PISO which enables to use much bigger time step size. We will perform a two-dimensional simulation, i.e. assuming that the channel has infinite width.

The computational domain is as follows:



A square cylinder of side length $D = 4$cm is placed inside a rectangular domain of length $L = 50D$ with a blockage ratio $B = 1/8$. $B = D/H$ where H is the height of the channel. The entrance length (distance between inlet and center of the block) is selected to be $l = 0.25L$.

The boundary conditions are:

- No-slip boundary at walls
- Uniform streamwise velocity profile at the inlet
- Neumann boundary condition at the outlet

The Reynolds number is typically defined as $Re = UH/\nu$. The velocity in the $x-$ and $y-$ directions are denoted as u and v, respectively.

A group of 4 students will complete the task and submit a combined report.

Tasks

The Deliverable Task I should contain the case folder of OpenFOAM (20p) and a written report (80p) describing the following results for $Re = 50, 150$ ans 310 :

- (20p) **Grid convergence study**: perform grid convergence study by computing Grid Convergence Index (GCI). Compute the re-circulation length L_r behind the cylinder and comment on which mesh you would choose for your simulations and why? (500×80 for your first grid)
- (10p) **Run pimpleFoam efficiently**: using the mesh chosen above, change the relaxation factor of pressure and velocities. Generate a table regarding the number of outer iterations with different relaxation pairs. The case folder with code having the improved relaxation factor is the one to be submitted. (maximum $3-4$ pair)
- (10p) **Theory**: What is vortex shedding? how vortex shedding occurs.
- (40p) **Results and interpretation**: with the mesh and relaxation factors chosen above perform the simulations and Generate a figure for each of the following quantities for each Re:
  - (4p) the velocity contour on the complete domain. (1 figure for each Re)
  - (4p) streamwise u and cross streamwise v along the centerline ($y = 0$) behind the cylinder and give explanation. (2 figure side by side 1 for u 1 for v each figure should contain comparison for 3 Re)
  - (4p) streamwise velocity u at $x = 0, 4$and 8 and give explanation. (3 figure for 3 Re)
  - (4p) cross flow velocity v at $x = 0, 4$and 8 and give explanation. (3 figure for 3 Re)
  - (4p) wall shear stress (τ_w) along the cylinder wall. (1 figure containing comparing 3 Re)
  - (4p) pressure coefficient ($C_p = (P - P_\infty)/0.5\rho U_\infty^2$) along the cylinder wall. (1 figure containing comparing 3 Re)
  - (4p) friction coefficient ($C_f = \tau_w/0.5\rho U_\infty^2$) along the cylinder wall. (1 figure comparing 3 Re)
  - (4p) lift coefficient ($C_l = F_l/0.5\rho A U_\infty^2$) (1 figure comparing 3 Re)
  - (4p) drag coefficient ($C_d = F_d/0.5\rho A U_\infty^2$) (1 figure comparing 3 Re)
  - (4p) what is Strouhal Number S_t? calculate the vortex shedding frequency.
- Explain your results and make your final conclusions about the Deliverable Task.

Bonus task (20p)

Perform $3D$ simulation for $Re = 310$ with $W = D$ and 20 grid points and compare your $2D$ simulation. Is there any difference between the results and if there is explain the reasons.

Deadline: 17th May,2018 23:59 **email: md.ashfaqul.bari@fau.de**