Numerical Methods in Thermo-Fluid Dynamics II

Summer Semester 2021

DELIVERABLE TASK I:

Large Eddy Simulation of flow over a circular cylinder at Re= 3900

Given: Monday, 23/04/2021

Deadline: Monday 31/05/2021

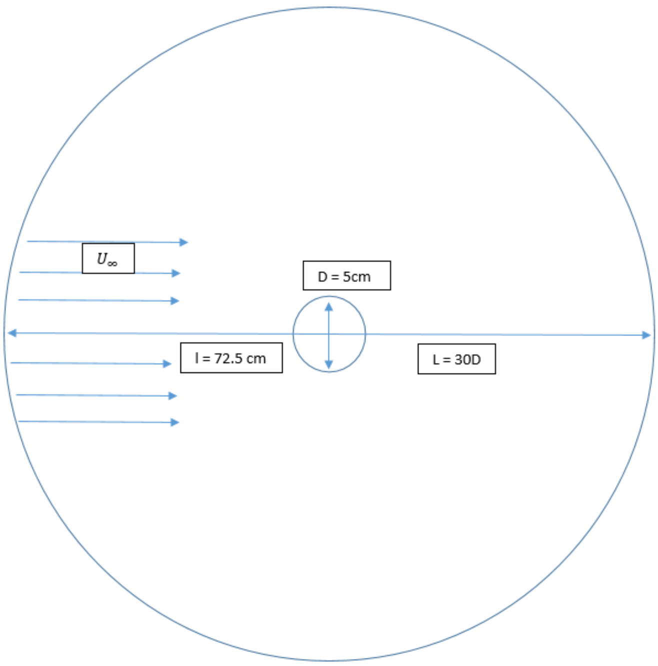
Institute of Fluid Mechanics
Department of Biochemical Engineering, Technical Faculty
Friedrich-Alexander University Erlangen-Nuremberg

Md. Ashfaqul Bari, Dr. Manuel Münsch

Simualtion of Turbulent flow over a square cylinder

The objective of the first deliverable task is to simulate flow over a circular cylinder by using PIMPLE method with OpenFOAM (pimleFoam solver)with Large Eddy Simulation(LES). Smagorinsky Subgrid scaler model will be used. Pimple algorithm is a combination of SIMPLE and PISO which enables to use much bigger time step size. We will perform a full three-dimensional simulation

The computational domain is as follows:



A circular cylinder of diameter $D = 5$cm is placed inside a circular domain of length $L = 30D$. The origin of the domain is located at $(0, 0, 0)$ The entrance length (distance between inlet and the closest cylinder wall) is selected to be $l = 72.5$ cm.

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 $Re = U_\infty l_{char}/\nu$. The velocity in the $x-$ and $y-$ directions are denoted as u and v, respectively. For this task $\nu = 1.0023 \times 10^{-6}$
A group of 4 students will complete the task and submit a combined report. Present your result on 31st May,2021

Tasks

The Deliverable Task I should contain the case folder of OpenFOAM (20p) and a written report (80p) describing the following results for $Re = 3900$:

- (10p) **LES**: summarize equations solved by openFoam for LES and Smagorinsky model.
- (10p) **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? (3 different mesh is provided)
- (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 (all the figure should contain proper labeling unit):
  - (4p) the velocity contour on the complete domain.
  - (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)
  - (4p) instantaneous streamwise velocity stream line for the complete domain and give explanation.
  - (4p) mean streamwise velocity stream line for the complete domain and give explanation.
  - (4p) wall shear stress (τ_w) along the cylinder wall.
  - (4p) pressure coefficient ($C_p = (P - P_\infty)/0.5\rho U_\infty^2$) along the cylinder wall.
  - (4p) friction coefficient ($C_f = \tau_w/0.5\rho U_\infty^2$) along the cylinder wall.
  - (4p) lift coefficient ($C_l = F_l/0.5\rho A U_\infty^2$)
  - (4p) drag coefficient ($C_d = F_d/0.5\rho A U_\infty^2$)
  - (4p) what is Strouhal Number S_t? calculate the vortex shedding frequency.
- Explain your results and make your final conclusions about the Deliverable Task.

Deadline: 31st May,2021 email: md.ashfaqul.bari@fau.de