46115 - Turbulence Modeling

Assignment 2: RANS and LES of flow past a circular cylinder using OpenFOAM

Deadline: 2 December 2024

In this assignment, you will investigate fluid flow and vortex shedding past a circular cylinder at Reynolds number Re = 3900 by modifying an existing tutorial.

The domain is sketched below:

set D = 1
V = 1
change ν to change Re

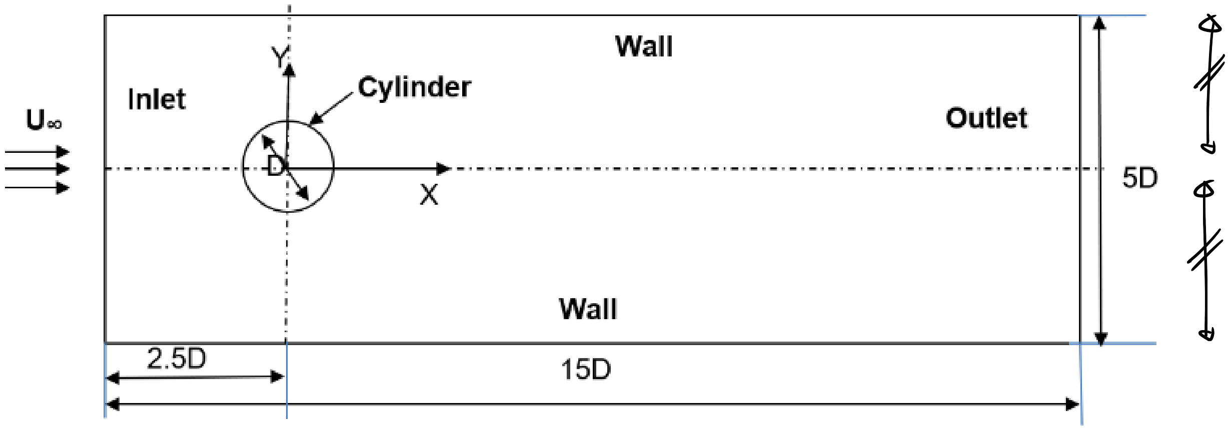

Figure 1. Sketch of the circular cylinder in uniform flow simulation

This domain setup is chosen for the assignment based on an available CFD simulation (at a different Reynolds number and using a different CFD Solver) so you can verify your numerical setup against the existing simulation [1].

a) You can get inspirations from the two references below to set up your simulations. So to begin with, read and summarize the references in 1-2 paragraphs each (what is it all about, what have they done and what are the key conclusions – a rather high level summary is ok.).
b) Start from the tutorial $FOAM_TUTORIALS/basic/potentialFoam/cylinder, edit the meshing dictionary, blockMeshDIct to match the needed dimensions (think of adding a few blocks and possibly changing the dimensions of the existing blocks) and prepare a full simulation setup. You would need systems, constant, 0 folder with appropriate contents including postprocessing tools.

2D MESH

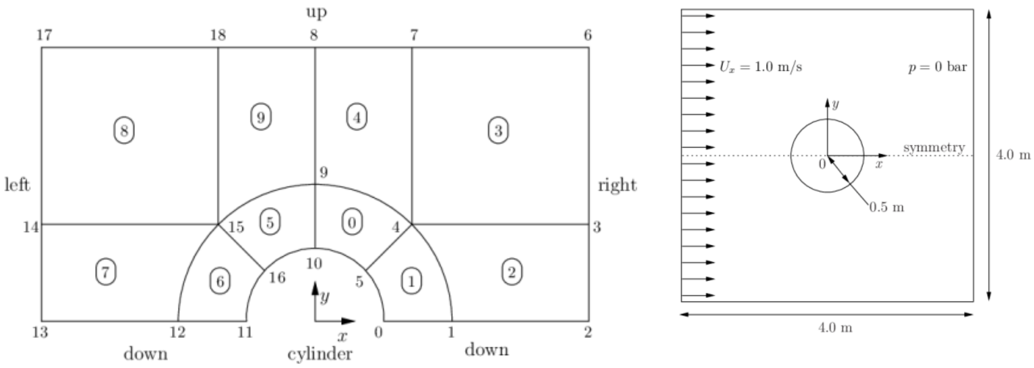

Figure 2. Sketch of the reference circular cylinder in uniform flow simulation: the blocks used for meshing. Note that this mesh needs to be adjusted to match figure 1

c) On the 2D mesh from the previous step, perform a k-epsilon and k-OmegaSST RANS model on the case. Present the flow field as well as force coefficients acting on the cylinder.

ask when going 3D

d) Turn this into a 3D mesh, use the recommended practice in wall resolution in the 3rd direction (z+ ~ 30[1]). Perform large eddy simulations on the case at laminar inflow conditions using Smagorinsky and the one equation (k) model and plot the flow fields and forces.

e) Repeat d only this time add 2% turbulence intensity using DFSEM.

f) Compare the above results against each other and ultimately against experimental measurements on mean pressure coefficient (Cp) and skin friction (Cf) on the cylinder, velocity and turbulence profiles (see reference 2 and reproduce their figures), on the cylinder and downstream. Plot y+ over the cylinder You are free to choose reference locations as long as experimental data are available for those points and that is gives a good picture of how the flow is developed across the domain.
   a. Report on the mesh and numerical setup
   b. How does each model compare against measurements. If you see a difference, try to explain the reason behind (simply saying there is a difference is not enough).

Upload the pdf report along with the "clean" case setups (only system, constant, 0, and your job script, and any postprocessing script you have used to postprocess your results, zipped). Refer to assignment 1 for guidelines on paper writing.

Please come and speak with me if you need my opinion on how to organize your teamwork.

Reference

[1] https://cfdflowengineering.com/cfd-modeling-of-flow-over-a-cylinder/

[2] Rajani, B.N., Kandasamy, A. and Majumdar, S., 2016. LES of flow past circular cylinder at Re= 3900. Journal of Applied Fluid Mechanics, 9(3), pp.1421-1435.

Appendix:

[1] You will naturally respect the overall meshing requirements for both RANS and LES.

Suggested task breakdown for an effective teamwork:

- Reading the references and getting insights for the numerical setup
- Modifying the existing blockMeshDict until a good mesh is obtained
- Establishing a base CFD setup for RANS
- Establishing a base CFD setup for LES
- Performing mesh sensitivity study – note that you can do many tasks in parallel. Both in preprocessing (manually, when preparing the cases) and in running cases (submitting multiple jobs)
- Postprocessing (new and existing tools/codes)
- Outlining the report
- Collaborative writing