

OpenFOAM assignment 2

See due date on Canvas

COE 347

There is absolutely no tolerance for academic misconduct. All assigned material is to be prepared individually. Submissions that are up to 24 hours late will be accepted with 50% penalty. Work submitted later than 24 hours after the deadline will not be graded.

Submit your homework electronically as a PDF via Canvas by 11:59pm on the due date.

If you are submitting a scanned copy of your handwritten notes, rather than a typeset document, please take the time to reduce the file to a manageable size by adjusting the resolution.

1 Objectives

In this assignment, you will simulate the flow around a circular cylinder. You will practice mesh generation and understand the effect of the quality of the mesh on the accuracy of the velocity field and its derivatives, i.e. stresses. You will also explore the dependence of the flow and drag on the Reynolds number. You will also have the opportunity to execute the simulations “in parallel“, i.e. using multiple cores on one or more nodes.

2 Resources

You may find useful information from various resources

- Class notes on the circular cylinder tutorial on Canvas
- Two journal papers (Williamson, Phys. Fluids 1988 and Park et al. KSME Int. J. 1998) on Canvas
- Tritton (Physical Fluid Dynamics), pp. 32-34.

3 Introduction

We shall consider steady and unsteady flow around a cylinder at various values of the Reynolds number, defined here as

$$\text{Re} = \frac{UD}{\nu}, \quad (1)$$

where U is the free stream velocity, D is the diameter of the cylinder, and ν is the kinematic viscosity of the fluid. For the sake of simplicity and without loss of generality, you can take $U = 1$ m/s and $D = 1$ m, so that $\text{Re} = \nu^{-1}$.

The Reynolds number is the only parameter that appears in the fluid flow equations for this particular flow, so that one may parametrize the solution with respect to the value of Re .

For very low values of Re , the flow is steady, falls in Stoke’s limit, and exhibits fore-and-aft symmetry. As the Reynolds number increases, the flow remains steady, but loses its symmetries and is no longer described by Stoke’s equations and the full Navier-Stokes are necessary. A recirculation region appears in the cylinder’s wake. As the Reynolds number increases further, the recirculation region becomes longer and

eventually the flow becomes unsteady in the wake region of the cylinder. Vortex shedding occurs, generating the well known “von Karman vortex street”.

We shall assume the flow to be strictly two-dimensional with x indicating the streamwise coordinate (oriented in the direction of the velocity in the free stream) and y the crosswise coordinate. The origin of the coordinate system is placed in the center of the circle, so that the point $(x, y) = (0, 0.5)$ is the “north pole” of the cylinder and $(0, -0.5)$ its “south pole”.

In what follows, one assumes that all lengths have been nondimensionalized by D , the cylinder’s diameter, all velocities by U , the free stream fluid velocity, and time by the *flow time* D/U . Thus, all quantities are assumed to be nondimensional.

For more details on the flow, see the slides discussed in class and posted on Canvas.

4 Preliminaries

Download the archive “circular-cylinder.tar.gz” from Canvas. Copy the archive to TACC or any other system you will be using for the assignment. Unpack the archive. Read the README file for a description of the contents.

5 Assembly of preliminary meshes

The goal is to produce two preliminary, coarse meshes using the OpenFOAM utility blockMesh.

The first mesh (mesh A) is used to simulate a low Reynolds number, steady flow around the cylinder. The second mesh (mesh B) is used to simulate a higher Reynolds number flow, which is unsteady and features vortex shedding.

Keep those requirements in mind when you select adequate domain sizes and shapes. Make sure the domain boundaries are at least 3 diameters away from the cylinder’s surface in all directions. You should be able to obtain meshes with $\approx 10^4$ cells.

Use the blockMeshDict template provided and supporting slides from Canvas to guide your work.

In your report:

- Include a schematic of the vertices, blocks, edges, and all other details that would allow to reproduce the two meshes.
- Attach (a) the two blockMeshDict files you used for mesh A and B; (b) the outputs from blockMesh; and (c) from checkMesh.
- Include figures (e.g. from Paraview) that provide both an overview of the mesh and its details, i.e. regions of higher resolution, as applicable.

6 Obtain a solution for $Re = 20$ and 110

Use the two meshes (A and B) that you obtained above to compute two preliminary solutions:

- A steady solution for $Re = 20$ on mesh A.
- An unsteady, periodic solution for $Re = 110$ on mesh B. For this unsteady case, make sure that the flow has “settled” and repeats periodically.

In your report:

- Include three contour plots for each solution. One for u/U (x component of velocity), one for v/U (y component), and one for pressure $p/\rho U^2$. For the unsteady case, choose an instant in time during the period.
- For steady case, include a plot of the streamlines to provide a qualitative overview of the flow. Show the detail of the recirculation region.
- For the unsteady case, include a plot that shows the time history of u/U , v/U , and $p/\rho U^2$ sampled at $(x, y) = (5.5, \pm 0.5)$ versus normalized time $t/(D/U)$. Use adequate values of the x-axis and y-axis range so that the time history of the flow quantities is clear.

7 Improving the mesh

Now that you have obtained two, preliminary solutions on meshes A and B, take a good look at the solutions and modify/change the meshes to improve the results with respect to the following:

- Resolution, i.e. how many cells in key regions of the flow, e.g. boundary layers, recirculation region, separation point, vortex street, etc.
- Domain size, i.e. how big of a domain should you consider in each region, i.e. forward, wake, cross-wise.

Consider the following aspects:

- Is the extent of the domain in front of the cylinder sufficiently large? Is the flow perturbed by the presence of the cylinder up to the left most domain boundary? Or is there a region ahead of the cylinder where the flow is uniform, just like a free stream?
- How about the domain in the wake of the cylinder? Make sure that the recirculation region and at least a few vortices in the unsteady case fall inside the wake region, rather than on the outlet, i.e. the rightmost boundary of the domain.
- Consider the crosswise extent of the mesh also, i.e. the top and bottom surfaces. Again, is the flow near those surfaces different from a uniform free stream flow? Is the presence of the surfaces perturbing the flow?

The paper by Park et al. contains some useful discussions on the crosswise extent of the domain that guarantees a solution that is independent of the crosswise domain size.

- Finally, is the extent of the inner block that wraps around the cylinder adequate? Is it too close to the cylinder, so that the transition from the inner block to the outer blocks falls in a region of interest?

In the remainder of the assignment, you will have to generate meshes and solutions that show clearly that the data you are reporting are *independent* of the mesh used.

In your report:

- Include a table that provides the details of each mesh you used to produce the results you are reporting. Clearly label each mesh in the table. In the table, include pertinent domain sizes and total number of points.
- Attach all blockMeshDict and relevant outputs from blockMesh and checkMesh for each mesh you tested.

8 Unsteady flow: Strouhal number of the vortex shedding

Consider the case $Re = 110$. You need to identify the frequency of the vortex shedding f , which is reported as a Strouhal number

$$St = \frac{f}{U/D} = fcn(Re). \quad (2)$$

Such frequency is a function of the Reynolds number.

Browsing the resources (Tritton, journal articles), find the value of the St number for $Re = 110$ for an unconfined circular cylinder.

You need to perform a simulation that reproduces the frequency of the vortex shedding.

You can use carefully placed “probes” to record the history of velocity and pressure at selected locations near the cylinder's surface and in its wake and process the time varying signals in order to obtain a frequency.

Make sure that your solution captures the location of separation on the cylinder's surface. Use the results in Park et al. to confirm that separation is occurring at the correct location on the cylinder's surface.

Use various meshes and time step sizes to make sure that the frequency you obtained is *independent* of the mesh (both resolution and domain size) and time step size.

In your report:

- Include a description of the procedure you used to obtain the St number. Be specific with the details so that the procedure may be reproduced. Use figures as needed.
- Include a table that lists the mesh used (indexed by the label for each mesh), the time step used, and the resulting St number
- Comment on your findings.

9 Steady flow: velocity in the boundary layer, separation length, and rate of strain at the cylinder's wall

Now consider the steady flow at $Re = 20$. You will need to report on the velocity in the boundary layer, the length of the separation region, and the rate of strain tensor at the cylinder's wall.

You will need to repeat the simulations with various meshes to demonstrate convergence of the results with respect to mesh resolution and the size of the computational domain. A way to show convergence is to plot the quantity of interest for various solutions on different meshes.

Consider the following quantities:

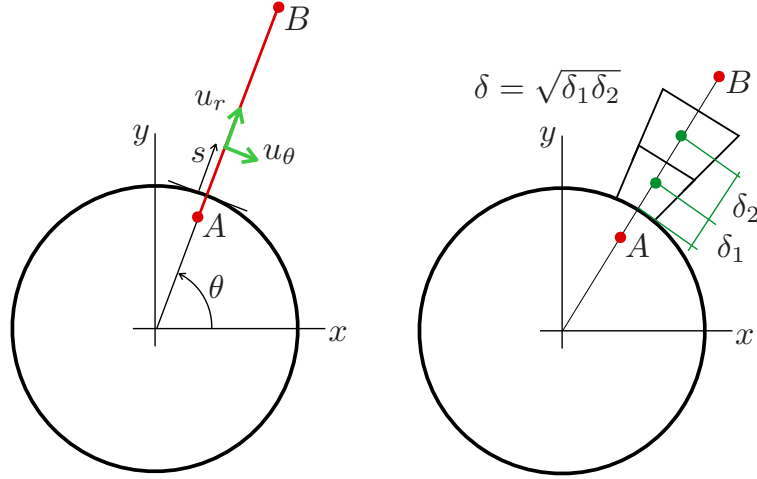


Figure 1: Schematic of the solution sampling strategy and the definition of $\delta = \sqrt{\delta_1 \delta_2}$.

- **Radial and tangential velocity normal to the cylinder's wall.** Extract the u and v components of velocity along a line at various angles θ using the sampleDict file and sample utility. See Fig. 1 for an example.

For $\theta = \{\pi/4, \pi/2, 3\pi/4\}$, compute u_r and u_θ , i.e. the radial and tangential components of velocity in the local coordinate system along the sampling segment, as a function of s , the distance from the cylinder's wall.

- **Rate of strain tensor at the wall.** Use the velocity data to compute the two components of the rate of strain tensor evaluated at the wall, at locations $\theta = \{\pi/4, \pi/2, 3\pi/4\}$:

$$e_{rr} = \frac{\partial u_r}{\partial r} \quad (3)$$

and

$$e_{r\theta} = \frac{r}{2} \frac{\partial}{\partial r} \left(\frac{u_\theta}{r} \right). \quad (4)$$

Note that in the above, r is the radial coordinate.

- **Length of the recirculation region.** Define the length of the recirculation region L as the distance between the rightmost point on the cylinder's surface along the x axis and the location where u is zero.

Normalize the length by the cylinder's diameter, L/D .

In your report:

- Include plots of u_r and u_θ versus the distance s at the three tangential locations requested. For each plot, show the results from various meshes to demonstrate convergence. Show clearly the velocities in the near wall region.
- Include a table of the values of L/D and e_{rr} and $e_{r\theta}$ at the wall for each mesh you considered. Compare L/D to what you expect based on the results in Park et al.
- Include plots of e_{rr} and $e_{r\theta}$ at the wall as a function of $1/\delta$ for various meshes. δ is a measure of the linear size of the mesh near the wall, defined as in Fig. 1. Use a combination of logarithmic and linear scales and make sure the plots are clear.
- Compare and contrast the convergence of the velocity components and that of the two rate of strain components with respect to the mesh resolution.

10 Extra credit (20 points) - Drag coefficient for the circular cylinder

Define as F the force on the cylinder per unit length in the spanwise direction. Then, the drag coefficient C is

$$C = \frac{2F}{\rho U^2 D}. \quad (5)$$

For Re up to about 50, $C \propto Re^{-1}$. See a discussion in Tritton (Physical Fluid Dynamics), pp. 32-34 on the drag on a cylinder as well as a plot showing C vs. Re .

Consider $Re = 20$. Your goal is to compute the nondimensional drag coefficient C and compare to the data in Tritton or in Park et al.

In your report:

- Include a detailed explanation on how you are computing the force F based on the solution
- Any intermediate results and derivations
- The comparison between your data and the data from other sources