Independent Research Project Plan

Integration of CFD Modelling Framework IC-FERST for Industrial Application in BP:
Automation of pre- and post-processing using Python

Dongzhen Li
Supervisors/advisors: Asiri Obeysekara, Andre Nicolle, Chris Pain

June 28, 2019

1 Introduction and background

1.1 Turbulent flow around a body

Computational Fluid Dynamics (CFD) is widely utilised in academic research as well as industry. A popular aspect of CFD is focusing on the flow around bluff bodies. [1] A familiar example for engineers is the blowout preventer (BOP) underwater. The BOP can be seen as a bluff body and the water body flows around that can be seen as a turbulent flow, so that it is important to know the actual conditions like the pressure load level and the flow patterns around the body to keep a steady and efficient working environment.

In real industry problems, there will be many complicated circumstances with different body shapes, and there are also respectable researches and experiments about those shapes, but during over the last half century, the properties in the wake of the flow around a circular cylinder have become the most work carried out. [3]

1.2 Flow past a circular cylinder

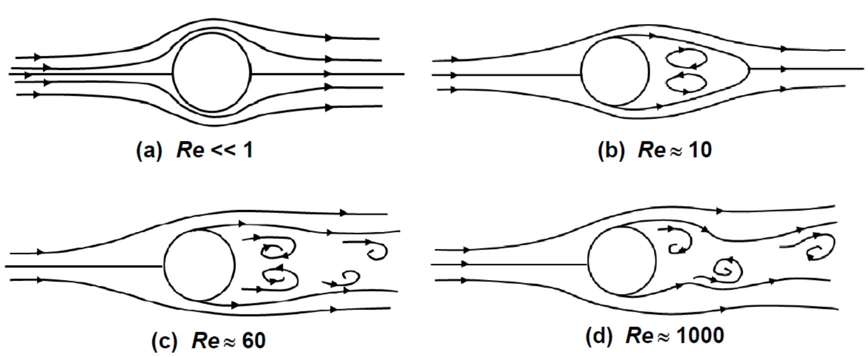

Figure 1: Flow pattern generated around a circular cylinder. [6]

Flow past a circular cylinder is a classic case, because it is seen as a simplified benchmark problem for many industry cases. According to [][][], modelling and programmes, such a case can validate the modelling code and then extended and modified to a more general cases.

Namely flow past a circular cylinder is a good start to this project. It is a representative of the unsteady three-dimensional flow containing separation, reattachment, free shear layer instabilities so that it is an appropriate example for three dimensional disturbances and transition from laminar to turbulent state of flow in the wake of the cylinder. Along with the varying Reynolds number, flow past a circular cylinder reveals distinct flow patterns into different flow regimes. [5]

Regime	Reynolds number	Flow characteristics	Experiments
Steady	$0 \rightarrow\sim 5$	Attached boundary layer	Kovasznay 1949 Coutanceau & Bouard 1977 Tritton 1959,1971
	~ 5	Flow convectively unstable	
	$5 \rightarrow\sim 40$	Symmetric, attached twin vortices	
	~ 25	Flow absolutely unstable (parallel stability theory)	
Unsteady Laminar	$40 \rightarrow 150$	Stable vortex street, decaying downstream	
	~ 90	Oblique vortex shedding	
Transitional	$150 \rightarrow 300$	Transition to turbulence in the wake. Fully turbulent wake in $40 \sim 50D$ downstream	Williamson 1989
Sub-critical	$300 \rightarrow 2 \times 10^5$	Transition in the free shear layers	Cantwell & Coles 1983 Roshko 1954a
	$\geq 10^4$	Most of the shear layers is turbulent Base pressure insensitive to Re	
Critical	$2 \times 10^5 \rightarrow 5 \times 10^5$	Lower transition in C_D from ~ 1.2 to ~ 0.3 Near wake width decreases to less than $1D$ Separation moves to rear of cylinder Laminar separation, transition, reattachment and turbulent separation of boundary layer	Achenbach 1968 Norberg 1987 Delaney & Sorensen 1953
	$5 \times 10^5 \rightarrow 3.5 \times 10^6$	Upper transition in C_D from ~ 0.3 to ~ 0.7 Near wake width increases (stays less than $1D$) Separation point moves forward	
Post-critical	$\geq 3.5 \times 10^6$	Turbulent cylinder boundary layer Regular vortex shedding ($St \simeq 0.27$), $C_D \sim 0.7$ Transition precedes separation, no reattachment	Roshko 1961 Shih *et al.* 1992

Figure 2: Summary of cylinder flow regimes. [2]

As the Figure 1 and Figure 2 above shown, the flow at small Reynolds number between 5 to 47 stays at a steady and laminar status. When it is getting closer to 47 and continuing up to a value of Re of about 190, there will be indication that alternate vortices shedding from the cylinder surface. In this stage, the vortices perform as quite regular, which is known as the von Karmen vortex street. If the Re keeps getting larger, the instability appearing in the two-dimensional flow will lead to disturbance in the far wake region in a three-dimensional way. [5]

Re = 3900 is a lower sub-critical value for the flow, which will remain laminar properties but have transition and separation characteristics in the free shear layer of the wake and turbulent eddies shed periodically along the wake. [5]

1.3 Analysis methodology

The flow around a circular cylinder at a sub-critical Reynolds number, which has become a benchmark for more application about complex flows in real technical issues. This benchmark is with simple geometry but can have complicated flow phenomena, including laminar separation with no fixed separation point, transition to turbulence in the thin shear layers, that are separating, and shedding of large-scale vortices [7]. [4]

About the problem turbulent flow around bluff bodies, engineers are interested in (1) the pressure distribution on the bodies, which is required in the design process with regard to failure under wind loading, (2) the velocities around the bodies, which is required for the assessment of pedestrian comfort and pollutant dispersion. [4]

From the research results of [5] [4] and which they have mentioned, the analysis about the simulation model of flow past a circular cylinder can be visualized as follows:

(1) the pressure distribution on the bodies:
Surface pressure coefficient distribution by computing the surface pressure coefficient along with the angle θ**.**

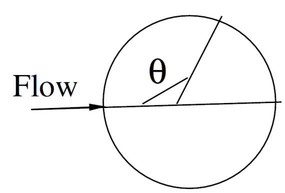

Figure 3: Schematic diagram of the angle θ. [5]

(2) the velocities around the bodies:
Mean streamwise velocity along wake centreline: $U_{CL} - x/D$

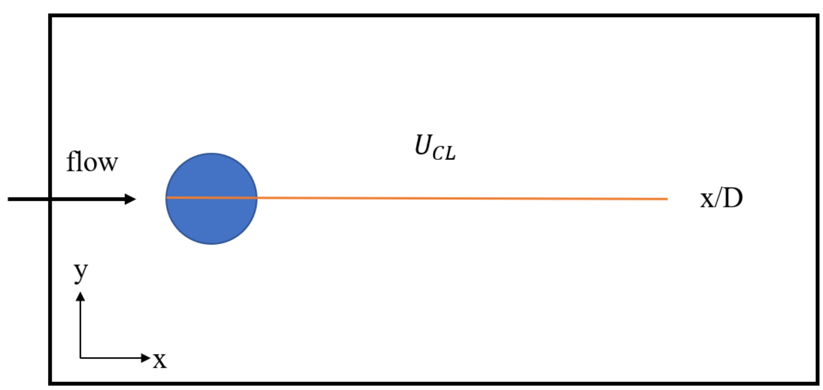

Figure 4: Schematic diagram of $U_{CL} - x/D$.

In both the near wake region and the farther wake region, for comparison reasons, it is sensible to choose three different distances x/D at each, like 1.06, 1.54, 2.02, and 4, 7, 10.

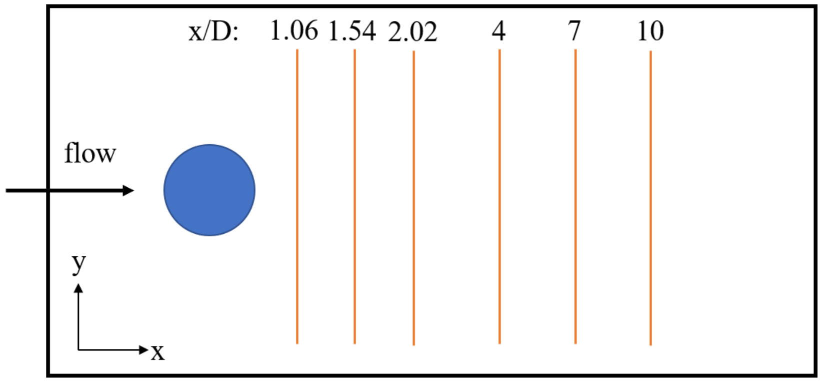

Figure 5: Schematic diagram of the choices in near and farther wake regions.

At those 6 positions, there are several quantities can be the output:
Transverse profiles of mean streamwise velocity: $\bar{u}/U - y/D$
Transverse profiles of mean cross-stream velocity: $\bar{v}/U - y/D$
Transverse profiles of the streamwise component of the resolved mean turbulent stress: $\bar{u'}^2/U^2 - y/D$
Transverse profiles of the crossflow component of the resolved turbulent stress: $\bar{v'}^2/U^2 - y/D$
Transverse profiles of mean Reynolds shear stress component: $\bar{u'v'}/U^2 - y/D$

1.4 Research code to commercial use

As the basic knowledge and its importance mentioned above, flow past a bluff body or a circular cylinder is a popular aspect for both academia and industry. For those advanced research code in that field, like IC-FERST, it is state-of-the-art but not integrated with pre- and post- processing well, considerable to implement the automation of simulation and its processing for better commercial engineering application purposes.

2 Objectives

The proposed work in this project is related to a research project between Imperial College and BP to evaluate advanced CFD codes IC-FERST developed in Imperial College by Applied Modelling and Computation Group (AMCG). These codes are well-suited to utilise the increasing computational resources and power through advancement in high performance computing (HPC). Due to the HPC context of the industrial partner, this project is aimed to integrate the pre- and post- processing automation by python scripts, for the most convenience to users running the whole procedure by scripts instead of manual operations on each software interfaces. The flow chart shows the integration development:

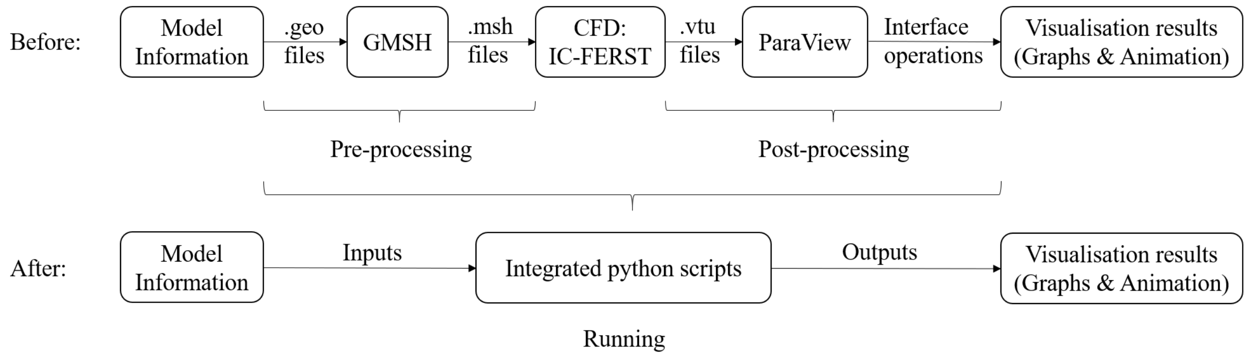

Figure 6: Flow chart of existing and after development working procedures.

GMSH is a mesh generator to generate the geometry (*.geo) and the mesh (*.msh) for pre-processing. As the flow past a circular cylinder has a simple geometry, the post-processing will become the focusing part here. ParaView is a popular choice for visualising and analysing in engineering problems, and the vtk library in python can be used to run ParaView functions by scripts. Vtk analyses the .vtu files such as scalar fields, vector fields and nodal positions, into NumPy arrays using python. Some useful vtk scripts list as follows:

vtk scripts		
function	code	notes
import the vtktools module into python	import sys sys.path.append('<<fluidity/tools directory path>>') import vtktools	

reads in a .vtu file	ug=vtktools.vtu('filename.vtu')
	
	#extract all files use this inside a loop:
	ug=vtktools.vtu('run_123_'+str(i)+'.vtu')
		"ug" is a vtu object

Get a list of all fields	ug.GetFieldNames()	A list of all contained scalar and vector fields
Extracting scalar and vector quantities	p=ug.GetScalarField('Pressure') uvw=ug.GetVectorField('Velocity')	Pressure: NumPy array called p; Velocity: the array uvw
Add a (new) field to the vtu object	psq=p**2 ug.AddScalarField('PressureSquared', psq)	For instance the square of the pressure as an extra field
Write the changed vtu object which also contains the new field PressureSquared	ug.Write('<<new vtu file name>>') #By not specifying a filename, the original vtu file will be overwritten ug.Write()By not specifying a filename, the original vtu file will be overwritten	The new vtu file can then be opened in Mayavi or Paraview, and the new field will be available for visual analysis.
Overview of all possibilities of vtktools.vtu	help('vtktools.vtu')	

Figure 7: Some useful scripts in vtk.

This project is established on a specific case the problem flow past a circular cylinder as a start point. After implementing in 2D circumstance for validation of the code, then transferring into a 3D simulation, more generalisation of the code will be concentrated on the tools.

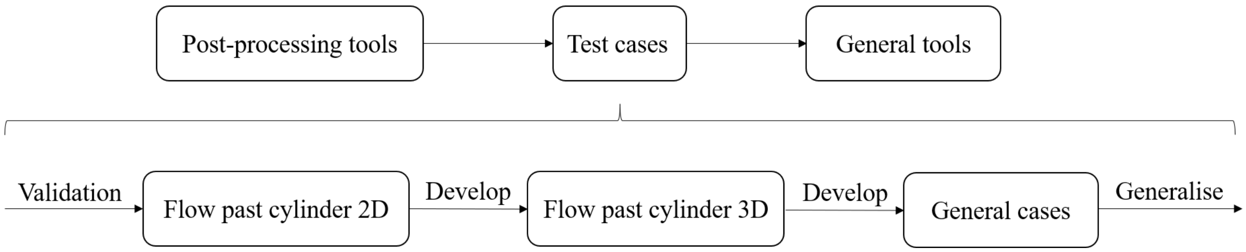

Figure 8: Work flow to develop post-processing tools.

Prototype code logics are as follows:
(1) Read in the position and shape parameters of the cylinder (coordinates, diameter and length etc.).
(2) Run the CFD simulation and get the quantity fields by vtk tools.
(3) Draw probes/surfaces on the cylinder and the wake (As the methodology in the introduction section above).
(4) Output the data in the specified position (probes/surfaces) and plot graghs in files.

3 Future plan (gannt chart)

The Gannt chart below shows the milestones in a 13-week time frame.

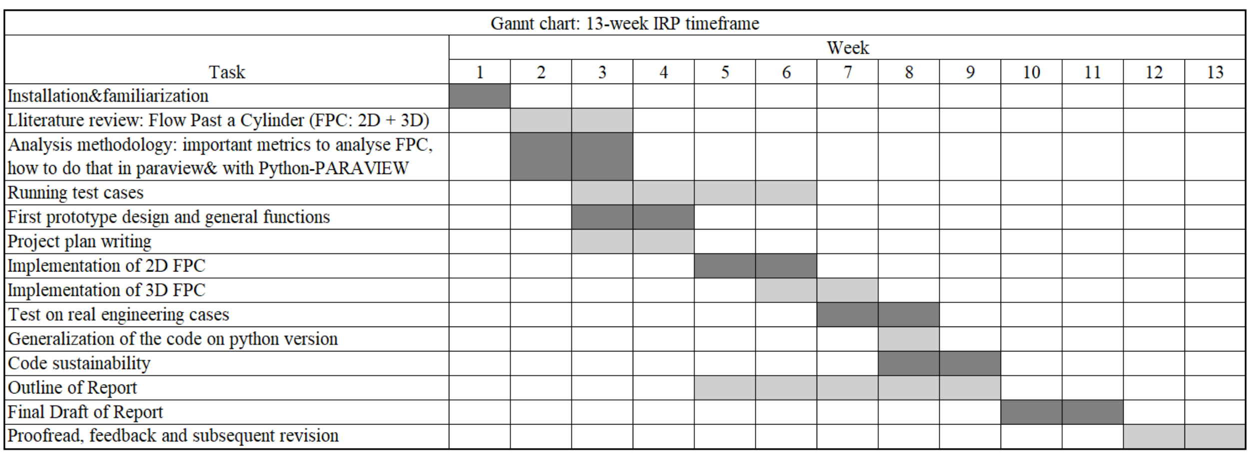

Figure 9: Gannt chart: 13-week IRP timeframe.

References

[1] Anagnostopoulos, P., Iliadis, G., and Richardson, S. Numerical study of the blockage effects on viscous flow past a circular cylinder. *International Journal for Numerical Methods in Fluids 22*, 11 (1996), 1061–1074.

[2] Beaudan, P., and Moin, P. Numerical experiments on the flow past a circular cylinder at sub-critical reynolds number. Tech. rep., Stanford Univ CA Thermosciences Div, 1994.

[3] Braza, M., Perrin, R., and Hoarau, Y. Turbulence properties in the cylinder wake at high reynolds numbers. *Journal of fluids and Structures 22*, 6-7 (2006), 757–771.

[4] Franke, J., and Frank, W. Large eddy simulation of the flow past a circular cylinder at red= 3900. *Journal of wind engineering and industrial aerodynamics 90*, 10 (2002), 1191–1206.

[5] Rajani B.N, A Kandasamy, S. M. Les of flow past circular cylinder at re = 3900. *Journal of Applied Fluid Mechanics 9*, 3 (2016), 1421–435.

[6] Sato, M., and Kobayashi, T. A fundamental study of the flow past a circular cylinder using abaqus/cfd. In *2012 SIMULIA Community Conference* (2012).

[7] Zdravkovich, M. M. *Flow around circular cylinders: Volume 2: Applications*, vol. 2. Oxford university press, 1997.