

Simulation of flows with different shaped cylinders using CFD

Anjan Samanta

*Department of Basic Science and Humanities,
University of Engineering and Management, Kolkata, India*

We are going to discuss how turbulent flows are being affected in presence of a suitable bluff body which is considered as a cylinder. Simulation has been performed using FEATool Multiphysics to analyse the flow numerically. Subsequently two post simulation analysis have been done namely pressure difference and drag coefficient. Lastly one simulation has been performed for a turbulent flow past square cylinder.

1 Introduction

In fluid dynamics, a bluff body is defined as that body whose surface is not aligned with the streamlines, when placed in the flow. For this reason flow separation occurs at the leading edge of the body and it experiences huge pressure drag; and minimal friction drag. Due to the presence of bluff body in flow we are having two phenomenon - boundary layer and wake region. Because of flow separation a disturbed region can be observed behind the bluff body- known as wake region. These phenomenon can be analysed experimentally; but with the help of software we can predict these stuffs almost accurately. That's what CFD is all about - Simulation of fluid flow and numerical analysis of the properties related to fluid. I find this field very interesting and we can explore a lot of things here.

2 Review of Literature

2.1 Navier Stokes equation

The description of incompressible fluid flow can be represented in terms of set of partial differential equations; known as Navier Stokes equation. They are as follows:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\nu \frac{\partial u_i}{\partial x_j})$$

where u is the velocity in the streamwise direction, p is the pressure, ρ is the fluid density and ν is the kinematic viscosity of the flow.

2.2 Reynolds Number

The **Reynolds number (Re)** helps predict flow patterns in different fluid flow situations. At low Reynolds numbers, flows tend to be dominated by laminar flow, while at high Reynolds numbers flows

tend to be turbulent. It is defined as:

$$Re = \frac{\rho u L}{\mu} = \frac{u L}{\nu}$$

where L is a characteristic linear dimension (m) and μ is the dynamic viscosity of the fluid ($Pa \cdot s$ or Ns/m^2 or $kg/(m \cdot s)$)

2.3 Drag Coefficient

Drag coefficient (c_d) is a dimensionless quantity that represents how much a body is being dragged in a turbulent flow. It is defined as:

$$c_d = \frac{2F_d}{\rho u^2 A}$$

where:

F_d is the drag force, which is the force applied on the body due to pressure along the flow velocity direction.

A is the reference area, which is usually the 2D projection of the frontal area.

2.4 CFD - a Brief Introduction

CFD (Computational Fluid Dynamics) is a numerical and simulation technique by which one can analyse different aspects of fluid flow which are difficult or expensive to derive from experimental setup. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. In present day we have some very advanced softwares like ANSYS FLUENT, openFOAM, FEATool multiphysics to analyse flow specially turbulent flow. But it is very much required to validate the softwares; this can be done by performing experiment. In addition, previously performed analysis of a particular problem can be used for comparison.

anjan.samanta@uem.edu.in

American Journal of Applied Mathematics and Computing

US ISSN: 2689-9957

©2019 Society for Makers, Artist, Researchers and Technologists

website: <http://thesmartsociety.org/ajamc/>

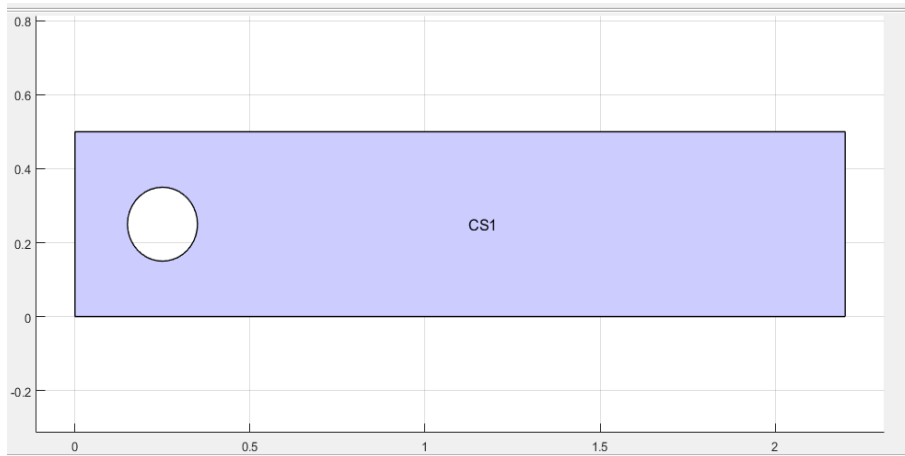


Figure 1: Computational Domain

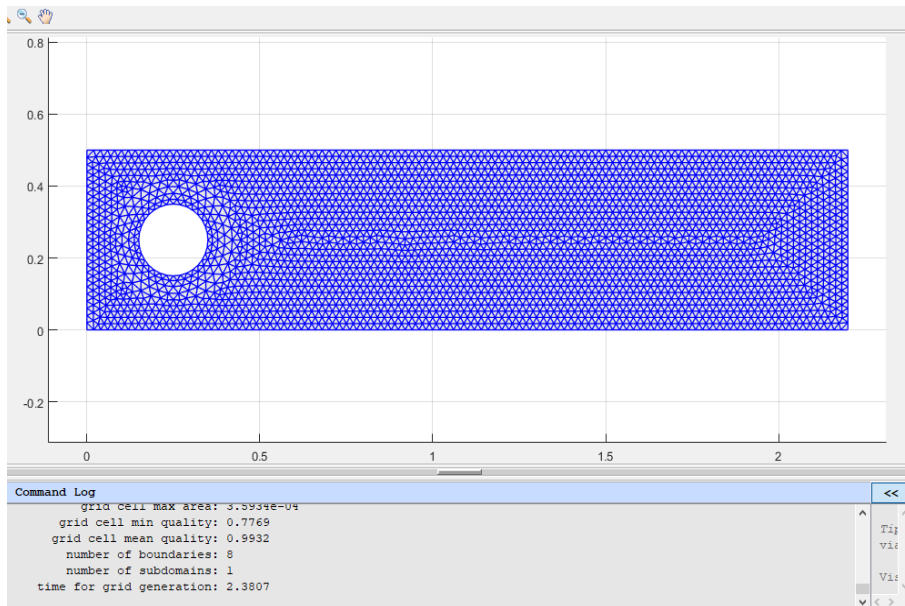


Figure 2: Grid generation

CFD has a wide range of application in research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering, fluid flows and heat transfer, and engine and combustion analysis.

3 Simulation of flow past cylinder

3.1 Numerical Details

3.1.1 Computational Domain

The computational region we have taken for flow simulation is the following diagram:

a rectangular tunnel with a cylinder is being taken for simulation. Dimension of rectangle is 2.2×0.5 . Circle of center (0.25, 0.25) and radius 0.1 in the diagram represents the cylinder (bluff body).

3.1.2 Grid generation

A grid is being generated in the computational domain of size 0.02 unit as follows:

3.1.3 Fluid density and viscosity

Fluid density is taken as 1 unit and viscosity as 0.002 unit.

3.1.4 Boundary conditions

- The inlet velocity profile (Boundary 4) for the flow is described to be parabolic i.e.

$$u_0 = 4 u_{max}y(h - y)/h^2$$

Where u_{max} = Maximum velocity at the center line and h is the width of the channel/region. Here $h = 0.5$ and $u_{max} = 0.35$.

- At outlet boundary 2 the pressure outflow is taken to be zero.
- At cylinder surface and top - bottom boundaries 1 and 3, we are assuming No-slip condition.

3.2 Simulation

Simulation is being done in **FEAtool Multiphysics**, a matlab software:

```
Grid Statistics:
Grid Generation Done.
number of grid points: 3183
number of grid cells: 6064
grid cell min area: 1.0351e-04
grid cell mean area: 1.7625e-04
grid cell max area: 3.5934e-04
grid cell min quality: 0.7769
grid cell mean quality: 0.9932
number of boundaries: 8
number of subdomains: 1
time for grid generation: 0.9591
```

it	def-u	def-v	def-p	rel-u	rel-v	rel-p
0	0.0071	0.00014	0.02			
1	0.0071	0.00014	0.02	1	1	1
2	0.0035	0.0036	0.0028	0.15	0.49	0.26
3	0.0014	0.00064	0.0011	0.058	0.14	0.057
4	0.00047	0.0002	0.00034	0.017	0.052	0.024
5	0.00014	4.9e-05	9.2e-05	0.0055	0.017	0.006
6	3.9e-05	1.4e-05	2.3e-05	0.0016	0.0047	0.0014
7	1e-05	4.4e-06	6.1e-06	0.00044	0.0013	0.00033
8	2.5e-06	1.2e-06	1.6e-06	0.00012	0.00032	8.7e-05
9	6.4e-07	3e-07	3.9e-07	3.1e-05	8.2e-05	2.3e-05
10	1.7e-07	7.7e-08	9.6e-08	8.6e-06	2.2e-05	6.2e-06
11	4.7e-08	2e-08	2.5e-08	2.5e-06	6.2e-06	1.7e-06
12	1.4e-08	5.5e-09	7e-09	7.4e-07	1.9e-06	4.9e-07
13	4.2e-09	1.6e-09	2.1e-09	2.2e-07	5.6e-07	1.5e-07

```
-----
Solver Statistics (solvestat)
-----
Number of dependent variables
-----
u : 3183
v : 3183
p : 3183
-----
Simulation timings
-----
t_asm(A) : 2.9s \ 39%
t_asm(f) : 0.0s \ 0%
t_sparse : 0.6s \ 8%
t_bdr : 2.0s \ 26%
t_lc/mv : 0.0s \ 0%
t_solve : 1.3s \ 17%
t_tot : 7.3
-----
```

where

- Surface plot represents the velocity field;
- Contour plot represents the pressure;
- Arrow plot represents the direction of fluid flow i.e. velocity field.

3.3 Post Simulation Analysis

From the simulation of flow field we can conclude that a recirculation zone has been formed behind the cylinder and the flow gradually becomes steady laminar downstream.

3.3.1 Pressure difference

Using FEAtool we can calculate the pressure difference between any two points (preferably front and rear of cylinder) which would help us to validate the simulation.

- Pressure at the point (0.15, 0.4) = 0.20525
- Pressure at the point (0.4, 0.2) = 0.02383

Hence pressure difference = $0.20525 - 0.02383 = 0.18142$

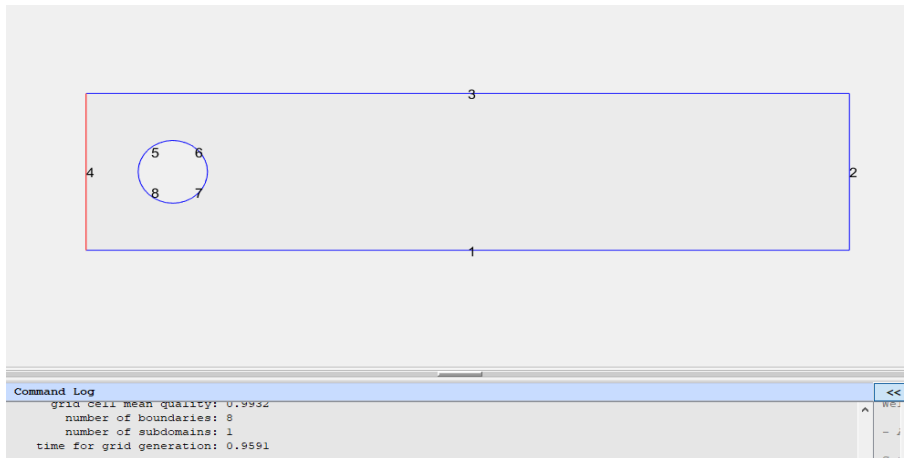


Figure 3: Boundary Conditions

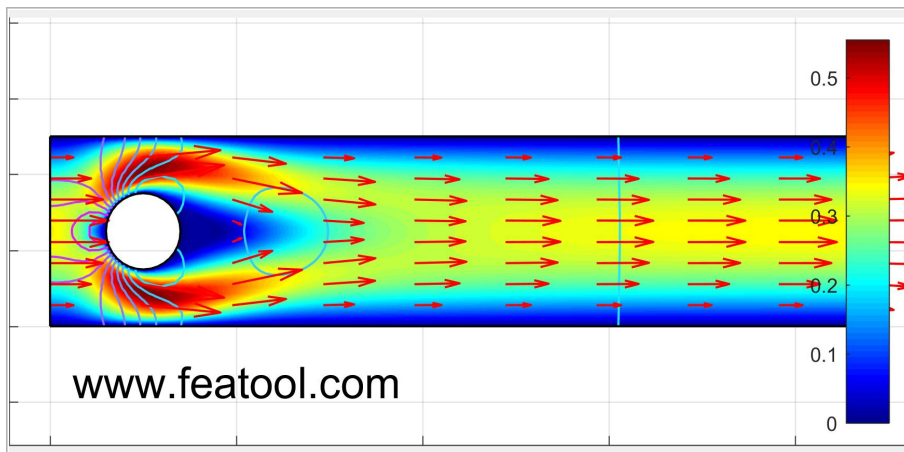
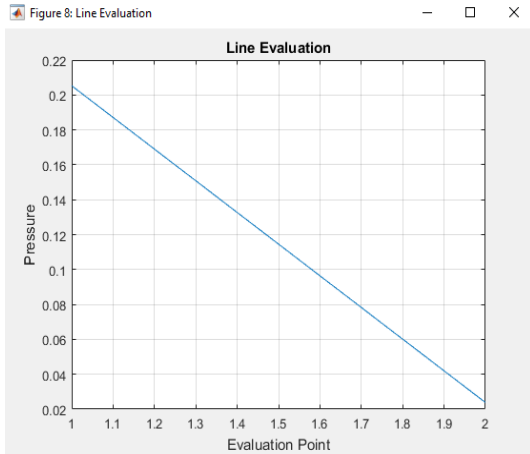


Figure 4: Simulation Obtained

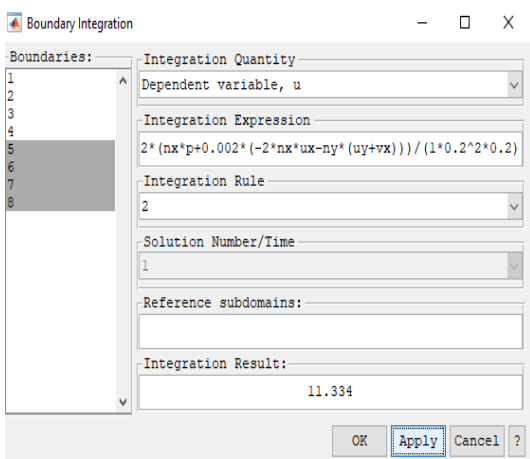


3.3.2 Drag coefficient

Drag coefficient can be determined by the following formula: $c_d = \frac{2F_d}{\rho u^2 A}$ where F_d is the drag force defined as follows:

$$F_d = \int_S \left(\mu \frac{\partial \mathbf{u}_\tau(t)}{\partial n} n_y - p n_x \right) dS$$

we can evaluate the above integral and find the drag coefficient. Here $C_d = 11.334$



4 Flow past a square cylinder

Square cylinder means rectangular or almost rectangular sharp edged cylinder such as beams, fences or buildings. Let us assume the same experimental set up like the previous one and perform the simulation of flow past square cylinders. The diagram obtained from FEATool given in Fig 6.

5 Simulation when we have two square cylinders in the flow

The following simulation we are getting for presence of two square cylinders taking all the initial conditions same as previous. We know Reynolds

number,

$$Re = \frac{\rho U_{mean} D}{\mu}$$

where D represents the pipe diameter. For the previous case $Re = 100$. Let us now increase the Reynolds number and try to see how it affects the fluid flow.

We assume $D = 1$, $U_{mean} = 0.5$ so that $U_{max} \approx 0.75$, $\mu = 0.001$ and $\rho = 1$. Then $Re = 500$ and simulation in FEATool gives us the following result. Clearly the wake region has been increased behind the 1st cylinder because of increment in Reynolds number, pipe diameter and decrement in viscosity.

6 Corner Modification - Proposed work

We have seen how fluid behaves for circular and square cylinder. Because of the presence of corner, the flow structure is affected enough for the case of square cylinder. So it will be interesting to see how corners are influencing the fluid flow. If we gradually convert the square to circle; i.e. corner radius from zero to half of side length ($r/D = 0.5$) and perform simulation that would be worth to investigate.

In paper [3] research has been done on corner modification for corner ratio, $r/D = 0, 0.167, 0.247, 0.333$ and compared it with the properties of flow past circular cylinder. We get conclusion from the paper that the drag coefficient c_d is inversely proportional to the corner ratio values, except for $r = 0.33$. I am proposing my work as follows: We will consider corner ratio values from 0.25 to 0.5 and we will see if we are getting same kind of anomaly or not; if so we will try to understand why this kind of deviation is happening. The reason could be also associated with Reynolds number; hence we will perform our simulations for different Reynolds numbers.

References

- [1] HP Kavya, Banjara Kotresha, and K Naik. "CFD Analysis of 2-D unsteady flow past a square cylinder at an angle of incidence". In: *International Journal of Advanced Research in Mechanical and Production Engineering and Development* 1.2 (2014), pp. 117–125.
- [2] Veeralkumar Thakur, Tarun Yadav, and B Rajiv. "Drag optimization of bluff bodies using CFD for aerodynamic applications". In: *Int J Comput Eng Res* 7.4 (2017), pp. 25–32.
- [3] Rujun Liu. "Flow Around Bluff Bodies with Corner Modifications on Cross-sections". In: (2019).

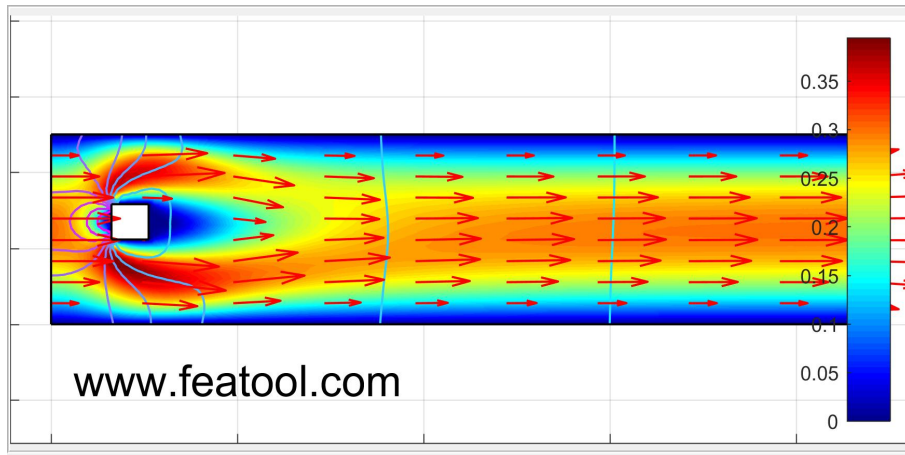


Figure 5: Flow past a square cylinder

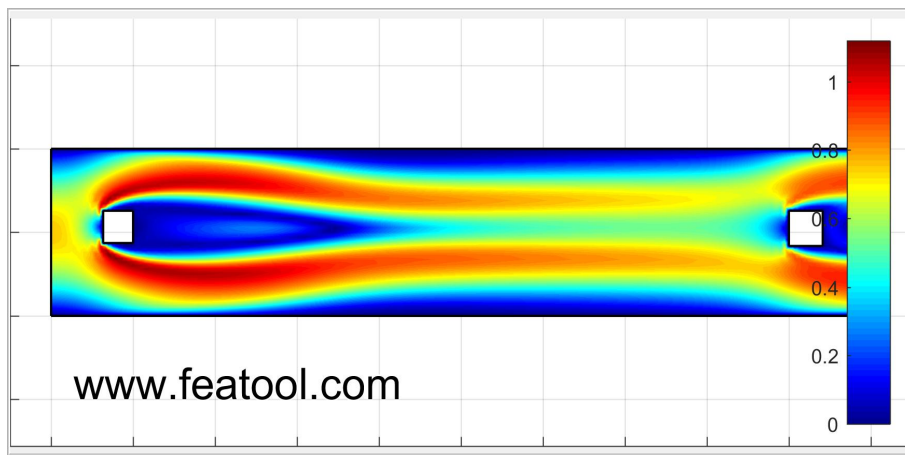


Figure 6: Flow past two square cylinders under certain conditions

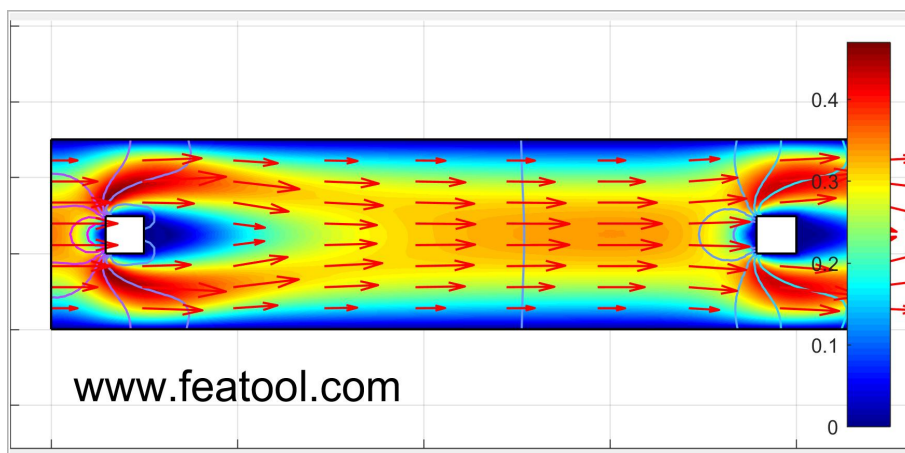


Figure 7: Flow past two square cylinders