NUMERICAL SIMULATION OF FLOW AROUND SIX IN-LINE SQUARE CYLINDERS

Submitted by

Monalisha
Department of Civil and Environment Engineering
Birla Institute of Technology, Mesra, Ranchi

29/06/2020

ABSTRACT

The project studies the flow around the six in-line square cylinders. I have used OpenFOAM-v7 for simulation and paraview for post-processing. The flow is as incompressible, turbulent, unsteady and the fluid is water. The flow has been simulated for different Reynolds Numbers and different S/d ratio, where S is the space between the two consecutive cylinders and d is the dimension of the cylinders. I have used k-Epsilon and K-Omega on same set of case to show the effect of different turbulence models on the results.

1. INTRODUCTION

The importance of the simulation of the flow over the six in-line square cylinders is that it helps in determining the aerodynamic performance of train and motor vehicles, structural design of bridges, buildings. The drag force and the lift force are important parameters to be considered while designing piers, trains etc. It further helps in optimizing the shape of the piers, train etc. to improve the life span and efficiency of the structure. OpenFOAM an open software has been used from creating geometry to the simulation of the case. Paraview has been used to obtain the result in the desirable format. A profound research and analysis have already been done in this domain of CFD.

2. PROBLEM SETUP

2.1 Geometry:

The figure (1) indicates the problem setup of the project. 6 in-line cylinders have been used because the addition of more cylinders is not going to affect the flow profile (C. M. Sewatet.al ,2012 [3]).The dimension of the computational domain has been so that the effect of the outer boundary will be negligible on the flow around the cylinders as per (C. M. Sewatet.al ,2012 [3]), flow around six in-line square cylinders). The length and breadth of the cylinder is 0.01 m and the height of the cylinder is 0.1 m.

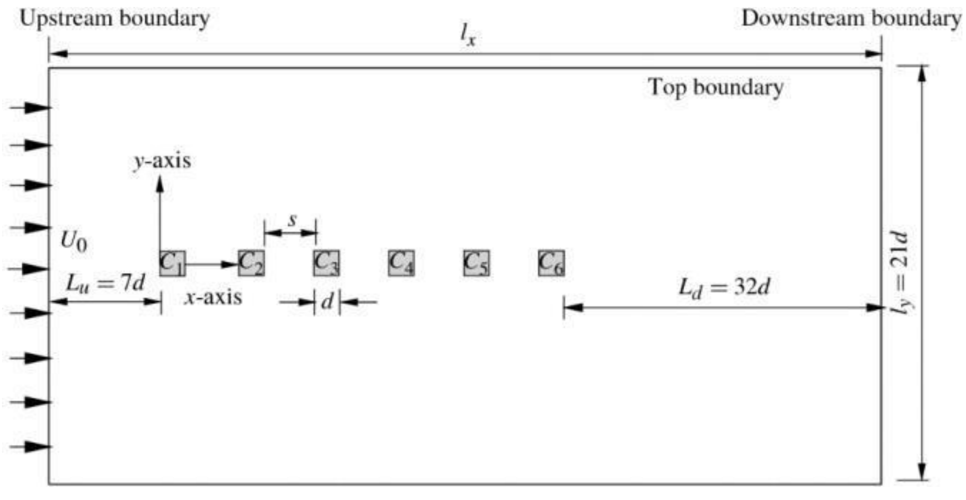


Fig. 01: computation domain
(C. M. Sewatet.al ,2012 [3])

The following cases have been studied in this project.

Case A:

I have used K- Epsilon on the following case setups:

S. No.	Cases	Reynolds Number	S/d	S (m)
1	Case 1	100	0.5	0.005
2	Case 2	100	4	0.04
3	Case 3	2400	4	0.04

Table 01

Case B:

I have used K- Omega on the following case setups:

S. No.	Cases	Reynolds Number	S/d	S (m)
1	Case 4	100	0.5	0.005
2	Case 5	100	4	0.04
3	Case 6	2400	4	0.04

Table 02

2.2 Fluid properties:

The fluid is water. The flow is considered as incompressible, turbulent, unsteady flow. The kinematic viscosity is $0.8*10^{-6}$ m^2/s , The density of water is 1000 kg/m^3.

2.3 Meshing and wall types:

Hexahedral block meshing has been done in the whole domain. The mesh is refined in the region surrounding the six in-line cylinders. Along z-direction, there is only one block and empty condition has been provided on patch to make it 2- dimensional. No slip wall condition has been provided on all the walls.

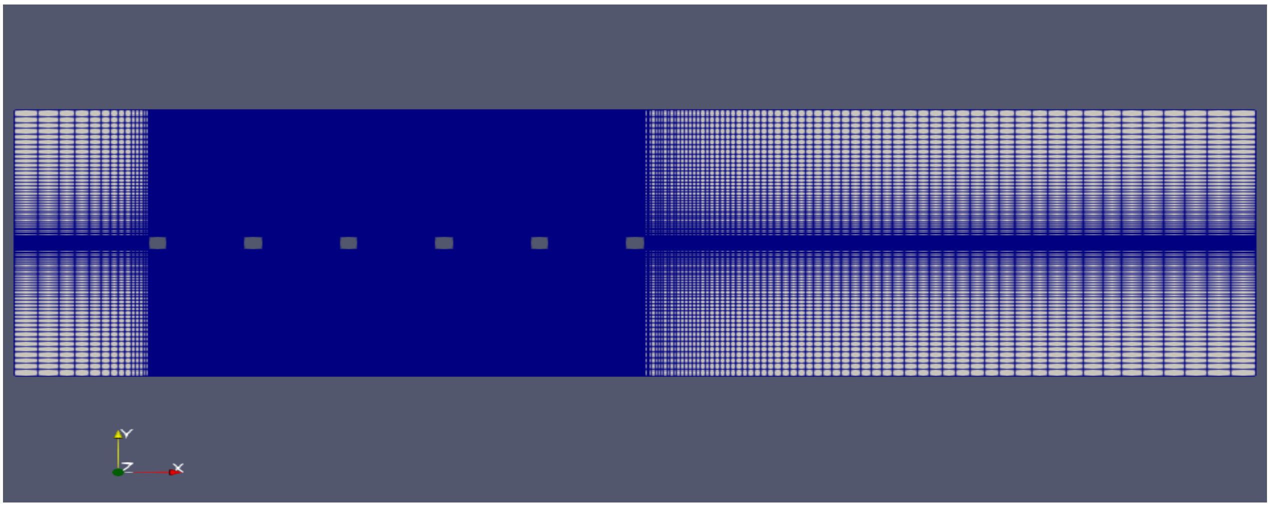


Fig. 02 : Computation meshing

2.4 Solver:

PimpleFoam solver has been used for the simulation of all the three cases. PimpleFoam is a transient solver for incompressible flow of the turbulent flow of the Newtonian fluid on a moving mesh. PIMPLE algorithm is the combination of the PISO and SIMPLE algorithm.

2.5 Turbulent Model:

The two-turbulence model have been used: K- epsilon and k-omega. K-epsilon turbulence model solves turbulent kinetic energy and turbulent dissipation rate transport equations. k-omega turbulent model solves turbulent kinetic energy and specific dissipation rate transport equations.

2.6 Initial and Boundary Conditions:

Outlet boundary condition

S. No.	Parameters	Outlet	Walls
1	velocity	zeroGradient	No slip
2	pressure	0 pa	zeroGradient
3	k	zeroGradient	kqRWallFunction
4	Є (valid for case 1,2, and 3)	zeroGradient	epsilonWallFunction
5	nut	zeroGradient	nutkWallFunction
6	ω (valid for Case 4, 5, and 6)	zeroGradient	omegaWallFunction

Table 03

Inlet condition

1. Pressure: zeroGradient for all the cases.

2. Velocity

S. No.	Cases	Values (m/s)
1.	Case 1, 2,4 and 5	0.008
2.	Case 3 and 6	0.191328

Table 04

3. Turbulent kinetic Energy

S. No.	Cases	Values (m^2/s^2)
1	Case 1, 2,4 and 5	2.4e-7
2	Case 3 and 6	1.368e-4

Table 05

4. Turbulent dissipation rate

S. No.	Cases	Values (m^2/s^3)
1	Case 1 and 2	2.8e-8
2	Case 3	3.756e-4

Table 06

3. Specific dissipation rate

S. No.	Cases	Values (1/s)
1	Case 4 and 5	0.3
2	Case 6	9.139

Table 07

4. Turbulence kinematic viscosity

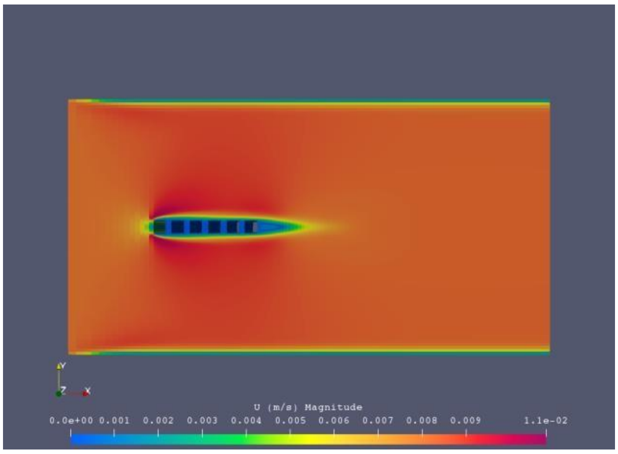
S. No.	Cases	Values (m^2/s)
1	Case 1, 2,4 and 5	1.85e-7
2	Case 3 and 6	4.48e-6

Table 08

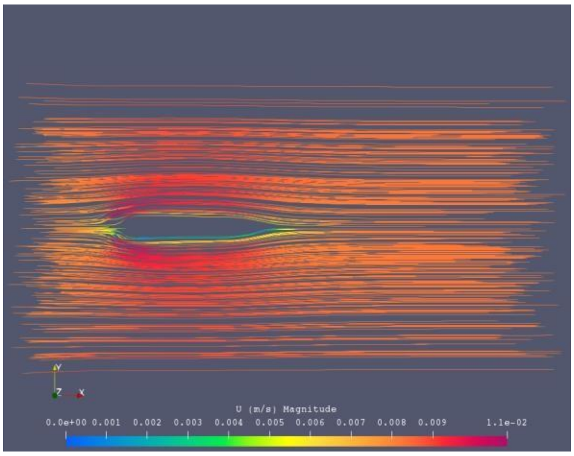
5. RESULTS

Case 01: R_e = 100; S/d = 0.04

The obtained profile does not show any vortices after 3000 iteration for the given conditions. The flow of fluid within the blocks is negligible. The different velocity layers can be seen in the below figures.



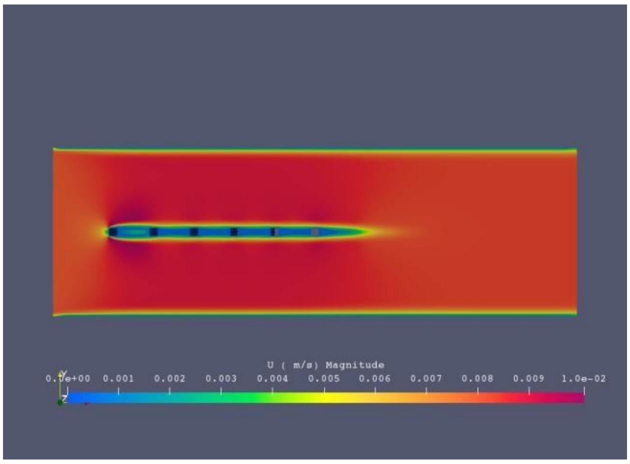
Complete velocity profile for case 01



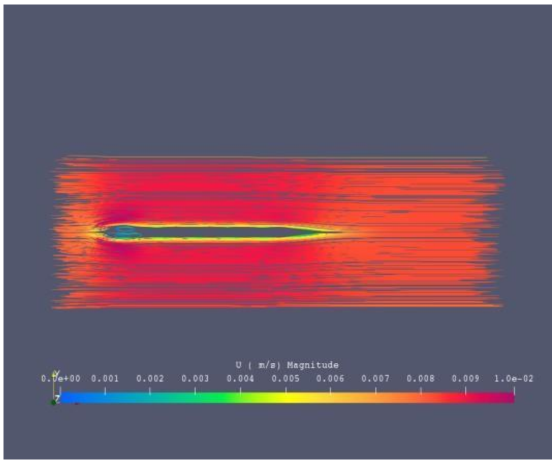
Streamline profile for Case01

Case 02: R_e = 100; S/d = 4

Velocity profile is almost similar to that of case 01. Unlike case 01, Fluid flow can be seen between 2 consecutive blocks after 3000 iterations. In streamline profile, vorticities can be seen.



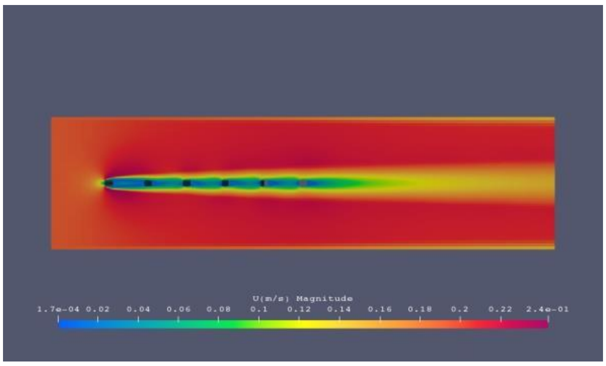
Complete velocity profile for case 02



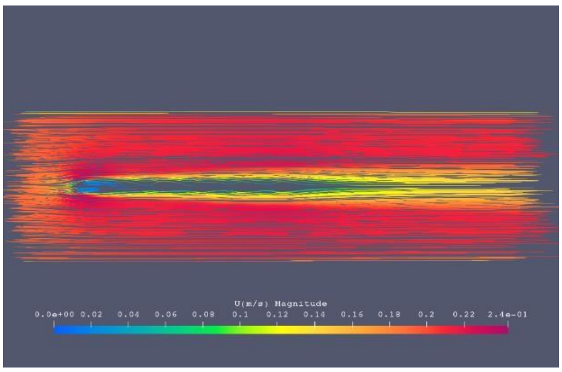
Streamline profile for case 02

Case 03: R_e = 2400, S/d = 4

Due to high Reynolds number, the flow around the cylinders is different from above 2 cases. The streamline also looks like wave around the cylinders after 1200 iterations.



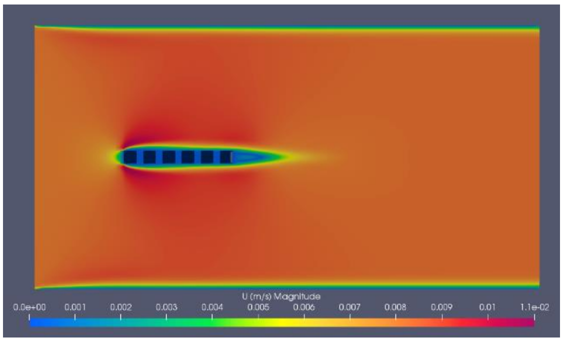
Complete velocity profile for case 03



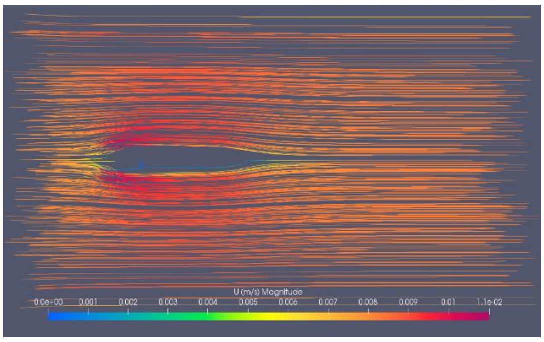
streamline profile for case 03

Case 04: R_e = 100; S/d = 0.04

The obtained profile for case 04 is obtained after 2000 iterations. K-ω is used for this case. Small vortices are obtained in the case 04. The spacing between the consecutive cylinders are small that the flow seems to flow without any interruption.



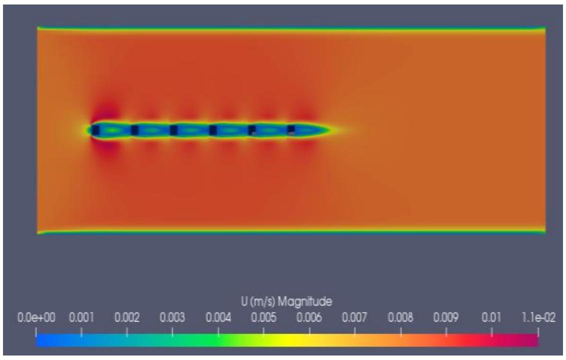
Complete velocity profile for case 04



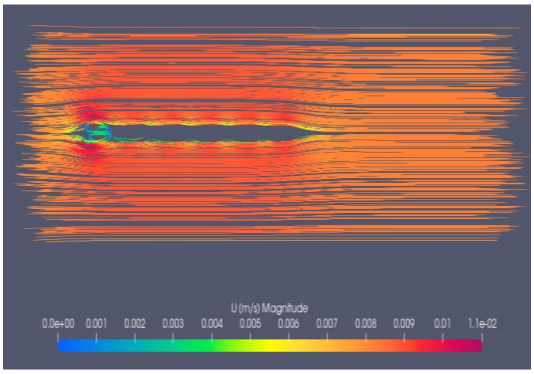
Streamline profile for Case 04

Case 05: R_e = 100; S/d = 4

The obtained velocity and streamline profile seem to be affected by the large spacing between the cylinders. Vortices have formed and it can be seen in front of inline cylinders. Theses profiles have obtained after 2000 iterations.



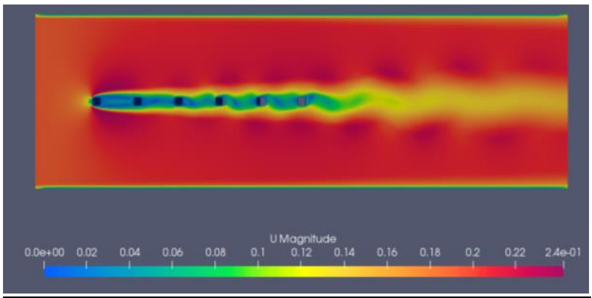
Complete velocity profile for case 05



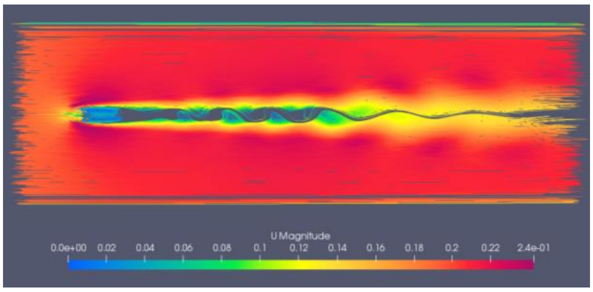
Streamline profile for Case 05

Case 06: R_e=2400, S/d = 4

The streamlines are wavy not as uniform as above cases. The effect of high Reynolds number and larger spacing can be clearly seen: Due to larger spacing, water has entered between the two consecutives cylinders and the tail of flow behind the last cylinders are very larger and prominent than the other cases due to high Reynolds number.



Complete velocity profile for case 06



Streamline profile for Case 06

6. CONCLUSION

In this work, a computational study of the experiments of Sewatkar et al.(2012) has been performed. The experiment looked at flow over six in-line square cylinders, and the computations attempted in this study has tried to reproduce the flow dynamics observed in the experiments. The effect of spacing between the square cylinders have also been looked at. The effect of spacing can clearly be seen from the figure: for case 1 and case 4 the spacing between the cylinders is very small that water seems to flow without any interruptions while for other 4 cases, waviness in the streamline can be seen; water has entered between the spacing of every cylinders. The effect of Reynolds number is also significant. For case 3 and case 6, the effect due to the presence of cylinders on the flow becomes prominent as we increase the Reynolds number. Two turbulence model have been used: K- epsilon and k-omega. The flow near the surface of cylinders are clearer in the cases using k-omega turbulence model. Vortices are captured in a better manner for case 4,5, and 6. So, I think it is better to use k-omega for this problem.

References:

1. https://www.cfd-online.com/Forums/
2. http://foam.sourceforge.net/docs/Guides-a4/OpenFOAMUserGuide-A4.pdf
3. C. M. Sewatet.al (2012), flow around six in-line square cylinders.
4. An introduction to computational Fluid Dynamics by H.Versteeg and W.Malalasekra.