International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 10, October-2015

Numerical Investigation of Flow Around Two Cylinders using Large-Eddy Simulation

Baraal Qays Mahdi
M. Sc. Student,
Mechanical Engineering,
Department of Mechanical Engineering.
Universiti Kebangsaan Malaysia,
Malaysia

Dr. Mohammad Rasidi Rasani
Lecturer,
Department of Mechanical Engineering,
Universiti Kebangsaan Malaysia,
Malaysia

***Abstract* - The main objectives of this research is to develop, document and study numerically the flow around one and two circular cylinders, particularly in the near-wake region, under various flow Reynolds number and various gaps between the two side by side cylinders. Both the time-averaged and instantaneous flow fields are studied. 3-D unsteady viscous flow around one and two circular cylinders in a side by side arrangement is chosen to study the characteristics of the flow in both laminar and turbulent regimes. The method applied is based on the finite volume method on a Structure Hexahedral grid. The implemented computational method is firstly validated through simulation of laminar and turbulent flows around a single and two circular cylinder at Re of 100 and 10000. Then, the flow around two circular cylinders in a side by side arrangement is analyzed for different gaps between the cylinders and also with 5 different Reynolds number. The flow visualization parameters like drag coefficients and others are comprehensively presented and compared for different cases in order to reveal the effect of the Reynolds number and gap spacing between the cylinders on the behavior of the flow. Since the flow behind the cylinders generates vortices which contains mechanical energy and can be converted to electrical energy, so piezoelectric sensors can be installed to convert this energy into electricity. This project provides the details of absolute pressure behind the cylinders at different points, based on the CFD results any point with highest pressure can be chosen as a position where piezoelectric sensor can be installed and maximum energy of the flow can be converted to get electrical energy.**

Key words : - Side-by-side cylinders, vortex shedding, flow induced forces, piezoelectric effect, absolute Pressure

INTRODUCTION

The topic of this thesis is the numerical investigation of flow past a circular cylinder to observe the difference in the results of Mean Drag Coefficient for one and Two Cylinder using Laminar and Large Eddy Simulation (LES) Turbulence Model for Validation and then analyzing the cases with different Reynolds number and different T/D Ratio between the two cylinders. The Flow past a group of circular cylinders in different arrangements plays an important role in various engineering applications and it has been the subject of many researches for decades. Furthermore, study of flow past two circular cylinders provides a broad perspective toward understanding the complex behavior of different arrangements. With respect to cross-flow, the configuration of two circular cylinders is divided to tandem, side-by-side and staggered arrangements.

Energy harvesting eels (eel) is a new device used piezoelectric polymers to transform mechanical energy flow, which is available in the oceans and rivers, into electrical energy. Eel generators make use of the regular trail of traveling vortices behind a bluff body to strain the piezoelectric elements.

PROBLEM STATEMENT

There has been various researches done on piezoelectric up to date but between those researches none of them had stated the right place to install the piezoelectric behind the two cylinders. The mechanical energy (kinetic) which is energy due to an object's motion (flow energy), as an example for the flow energy the Vortex Induced Vibration (VIV) arises from the interaction of a moving fluid with an elastic structure. 'VIV' happen in several engineering states, such as bridges, smokestacks, transmission lines, plane control surfaces, off-shore constructions, thermo wells, engines, heat exchangers, drilling and production risers in oil production, airiness and spar hulls, pipelines, as well as other hydrodynamic and hydro acoustic uses. For all reasons above this project is attempting to find the exact location behind the two cylinder for installation of piezoelectric in order to get the maximum performance, the turbulent flows provide a unique opportunity to produce substantial pressure fluctuations which in turn may be utilized by converting mechanical energy of ambient to electrical energy by using piezoelectric.

SCOPE OF THE STUDY

The scopes of this project is to perform CFD Analysis for flow over one and two cylinder with different Reynolds number and different gap between the two cylinders and get the values of pressure behind the cylinders. The study will be limited to the study of absolute pressure behind the cylinders and the mean Drag Co-efficient for the Cylinders is used to validate with experimental data. This project studies the effect of flow of fluid over one and two cylinder with different Reynolds number and with different gap between the two circular cylinders to find the position where the absolute pressure is maximum behind the cylinders to install the piezoelectric sensor at that position.

OBJECTIVES OF THE STUDY

This study is conducted to investigate numerically the flow of water over one and two cylinder by using" Large eddy simulation" (LES) Turbulence Model. The objectives of present study are: 1. Investigation of the distribution of velocity and pressure around two cylinders and compare with one cylinder. 2. To examine the effects of 5 different gaps between the two side by side cylinders on the flow fields. 3. To analyze the effects of 5 different Reynolds Number on the flow fields over one cylinder and for five different gaps between the two cylinders.

METHODODLGY AND CFD SET-UP

Computational Domain

The computational domain and the boundary conditions for the simulation of the flow are shown in (Figure 1). The cylinder is simulated with a diameter (D) of 0.04 m and a depth of 1.6 and Length of 0.8 m as per the Literature. The computational domain consists of an upstream of 10 times the diameter of cylinder to downstream of 10 times the diameter of the cylinder, The Inlet is kept at 10 times of the Diameter and Outlet at 30 times the Diameter for One Cylinder and for Two Cylinders L/D Ratio is selected as 1.5, 2, 2.5, 3 and 4. Ansys Design modeler, the CAD modeling tool of Ansys is used to create the computational domain as per the dimensions shown in Figure 1, here D is the diameter; T is the gap between the two cylinders.

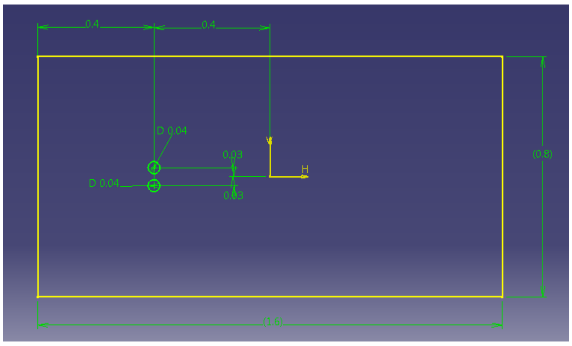


Figure 1: Sketch for Two side by side cylinder.

Meshing

A structured hexahedral mesh is employed in these simulations. Structured mesh is generated using ANSYS ICEM CFD, and then imported into ANSYS FLUENT. Figure below displays the mesh generation in the computational domain. Figure below shows the grid near the cylinder surface of the domain. Near to the cylinder wall and in the wake, very fine mesh is required to resolve boundary layer separation and the vortex street. The closer view of near the cylinder surface is shown in Figure below.

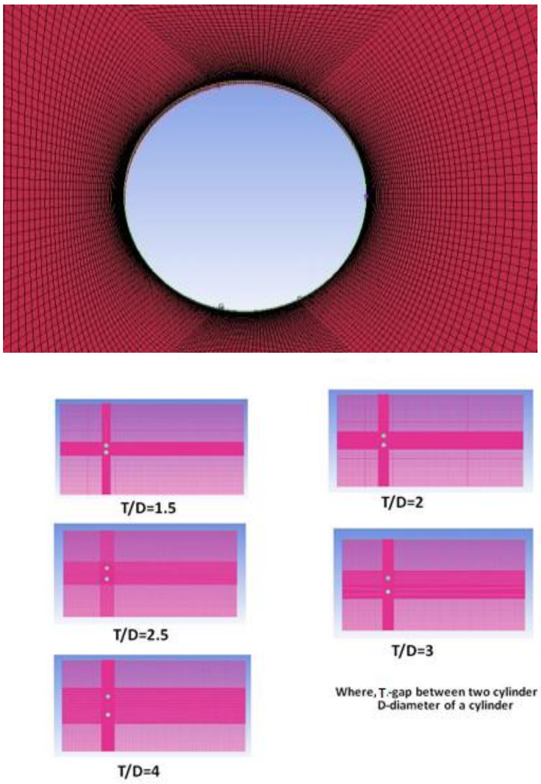


Figure 2: (a) Meshing near the Cylinder and (b) Meshing for the five cases.

Mesh Quality

The quality of the mesh is determined by the determinant 3x3x3 matrix in ICEM CFD. Value 0 indicates low quality elements and 1 indicates good quality elements. In our case we got a value 0.8 which is good and acceptable. Another quality matrix is aspect ratio which ranges from 0 to 100, where 0 indicates low aspect ratio i.e. good quality elements 100 indicates high aspect ratio i.e. bad quality elements. It is advisable to keep less than 50. In our case we had an aspect ratio of around 40 which is acceptable.

Problem setup & Boundary conditions

Uniform velocities of 0.002505 m/s, 0.097695 m/s, 0.2505 m/s, 0.37575 m/s and 0.5511 m/s are introduced at the inlet correspond to the Reynolds number of 100, 3900, 10000, 15000 and 22000 respectively. The outlet boundary is defined with an average static pressure of 0 Pa. The rest of the boundaries like side and bottom wall are defined as Symmetry, and the cylinder wall is no slip wall boundary condition where velocity increases from zero at the wall surface to the free stream velocity away from the surface.

Simulation set-up

The numerical solution of the flow is obtained using a commercial CFD code: ANSYS FLUENT 14.5. In this 3D-code, discretization is done based on a finite volume approach. In this section the various CFD code settings and options are summarized. An unsteady and pressure based solver are used. Boundary conditions and different discretization schemes are used depending on the turbulence model. Pressure based solver is used, pressure coupling is done using SIMPLE Scheme, flow is defined as laminar or turbulent based on Reynolds number, convergence criteria is kept to be 1e-04. Pressure is given as standard and momentum is defined as second order scheme. A fluid is considered with a density of 1000 kg/m3, viscosity of 0.001002 kg/m.s.

Turbulence Modeling

Large eddy simulation (LES) has been classified as a space filtering method in CFD. The large eddy simulation directly calculates the large scale turbulent structures which are responsible for the transfer of energy and momentum in a flow while modelling the smaller scale of dissipative and more isotropic structures. In order to distinguish between the large scales and small scales, a filter function is used in LES. A filter function dictates which eddies are large by introducing a length scale, usually denoted as Δ in LES, the characteristic filter cut-off width of the simulation (Deardorff, J. W. 1970) All eddies larger than Δ are resolved directly, while those smaller than Δ are approximated.

Grid Independency Test

Grid independency is a test in which we observe that no change in result is observed as we go on refining the mesh. In this work a grid independency test is carried for three sizes of mesh. 170x170x8, 190x190x8 and 210x210x8. These are the number of elements in the x, y and z direction. The value of mean Drag Co-efficient is taken for comparison of the results.

Table 1: Grid Independency Test for a single cylinder at Re = 100

Grid type	Number of cells	Mean drag co-efficient
Finer	210x210x8	1.29
Fine	190x190x8	1.27
Standard	170x170x8	1.01

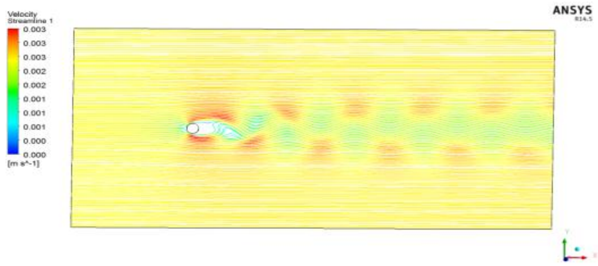
From the above table it can be seen that no significant change is found between the results of 190x190x8 mesh and finer 210x210x8 mesh hence to save the computational time and maintain the accuracy, 190x190x8 number of elements are chosen for all further cases.

RESULTS AND DISCUSSIONS

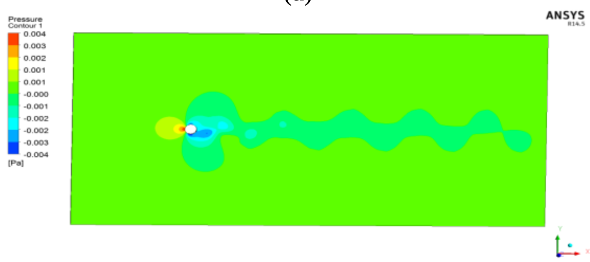
The simulations are conducted for a T/D ratio of 1.5, 2, 2.5, 3 and 4 with five Reynolds numbers of 100, 3900, 10000, 15000 and 22000. The choice of the Reynolds numbers of simulation depends largely on the experimental data available. Reynolds number 100 and 10000 is chosen for validation purpose since experimental data is available.

1. *Validation Case for A Single Cylinder with Re = 100*

In this section, we will discuss the flow over single cylinder with Re = 100. For this purpose, we will use laminar model.



(a)



(b)

Figure 3: Flow characteristics for a Single Cylinder at Re = 100.

The time domain data of the drag coefficient is converted into frequency domain by using the Fast Fourier Transform (FFT). Good agreement with the experimental result is observed. The results for pressure, velocity streamlines and Lift and drag co-efficient is shown in the figures.

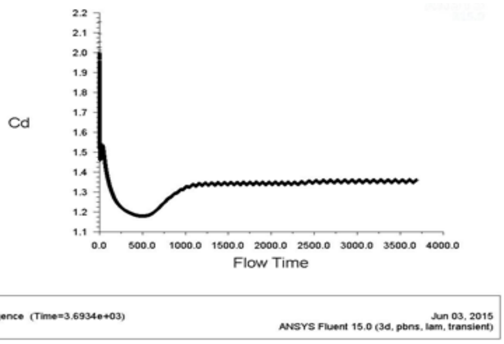


Figure 4: Time histories of drag forces for single cylinder at Re = 100

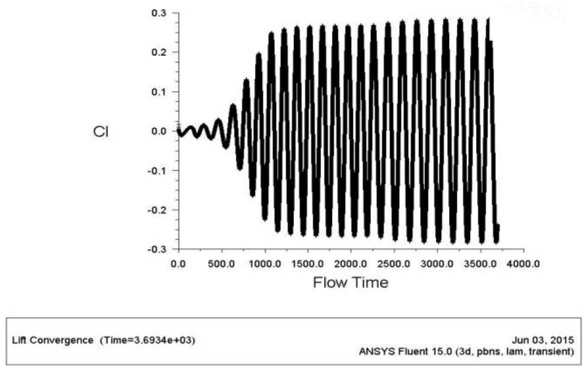


Figure 5: Time histories of drag forces for single cylinder at Re = 100

Five lines are created to get the values of Absolute pressure on those lines. The lines are created in such a way that first line is at a distance of 0.05 m from the cylinder and all other lines are 0.05 m away from each other.

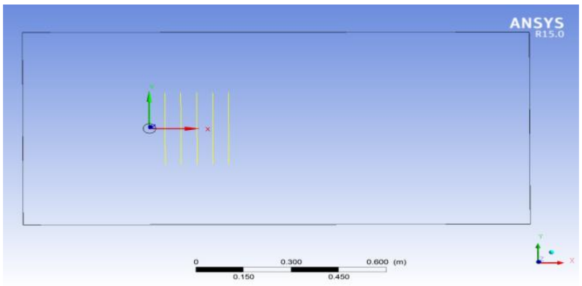


Figure 6: Five Lines created behind the single cylinder.

The length of the line is taken as 0.30 m, in which 0.15 m is above the central axis and 0.15 is below the central axis. All the two cylinders are symmetry about this axis. The position of the x and y axis is shown in figure 5 above. The values of absolute pressure are shown in figure 6 below.

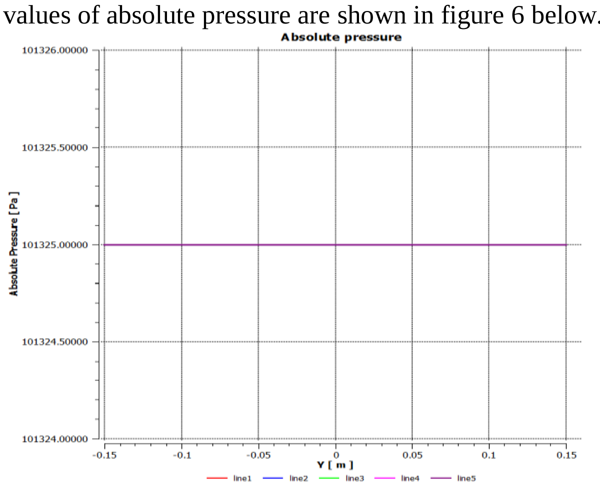
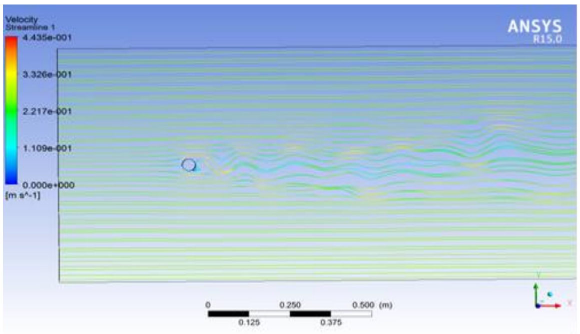


Figure 7: Absolute Pressure plot on five lines for Re = 100

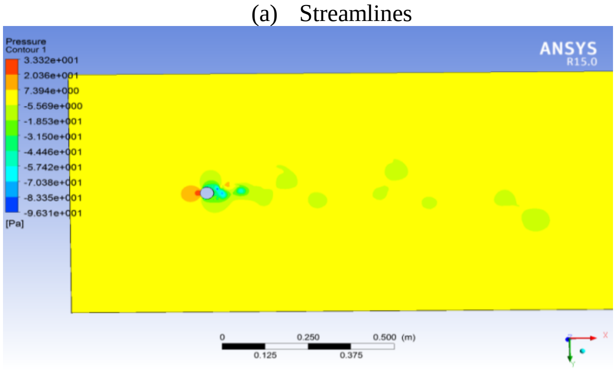
The simulation results of flow past a single cylinder is presented for mean drag and lift coefficient and are compared with the data in literature in Table 1. From Table 1, we can see that at Re = 100 the results of the present work agree closely with other results. For single cylinder with Reynolds number of 100 the value of maximum gauge pressure behind the cylinders is -0.002 pa and absolute pressure is 101324.998 Pa.

2. VALIDATION CASE FOR A SINGLE CYLINDER WITH RE = 10000

In this section, we will discuss the case with Re = 10000. For this purpose, we will use LES Turbulence Model. Good agreement with the experimental results is observed.



(a) Streamlines



(b) Pressure contour

Figure 8: Flow characteristics for a Single Cylinder at Re = 1000.

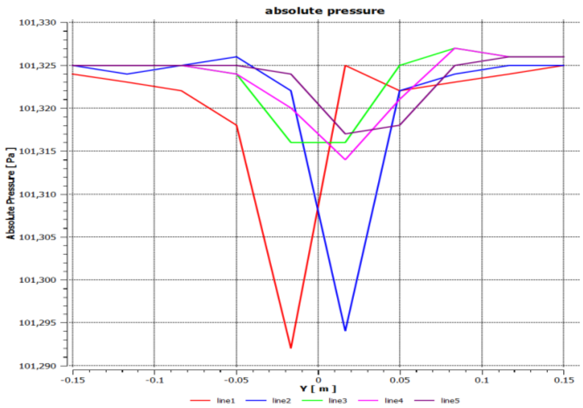


Figure 9: Absolute Pressure plot on five lines for Re = 10000

As can be seen from the streamlines the separation area is reduced in this case as compared to laminar flow, this is due to the tendency of the turbulent flow to remain attached to the wall. There is no gap behind the cylinder is observed here as it was in previous case, here the flow reverses due to higher velocity and gets attached to the cylinder which is not observed in Reynolds number of 100. From figure 9 we can observe that the maximum absolute pressure is on line 3 which is at a distance of 0.15 m from the cylinder in x direction. The maximum absolute pressure is 101327 Pa. The (x, y) coordinates for maximum absolute pressure is (0.15, 0.07) m respectively.

Table 1: Comparison of Mean drag Coefficient or Present results with Literature for single cylinder, Re=100

Number Of Cylinders	Re	Present	Dehkordi Behzad Ghadiri	Gopalkrishnan Experimental	Maneghini	Ding Et Al	Singha Et Al	Dong, DNS Method
One	100	1.27	1.39	-----	1.37	1.35	1.431	
One	10000	1.32	-------	1.186	------	----	----	1.143

Table 2: The table shows the Mean Drag Co efficient for Two Cylinder at T/D = 2, Re= 10000

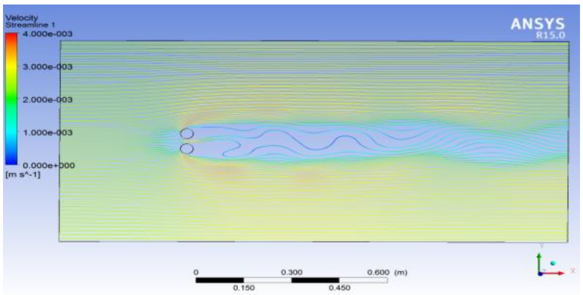
Number of Cylinders	Cylinder	Cd (Present)	Cd (SarvghadNavid)
Two Cylinder	Top	1.41	1.37
	bottom	1.36	1.37

3. RESULTS FOR 25 DIFFERENT CASES CONSIDERED FOR TWO SIDE BY SIDE CYLINDERS

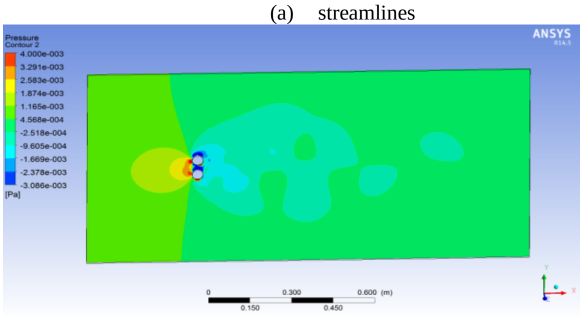
Analysis results of the twenty five Cases are shown in this section, and comparison of the results for pressure forces behind the cylinder is presented.

Discussion for Two Side by Side Cylinder with T/D = 1.5 and Re = 100

The flow behavior around two side by side cylinders in laminar regime is analyzed. The results are shown below.



(a) streamlines



(b) Pressure Distribution

Figure 10: Flow characteristics for a Side by Side Cylinder at Re = 100.

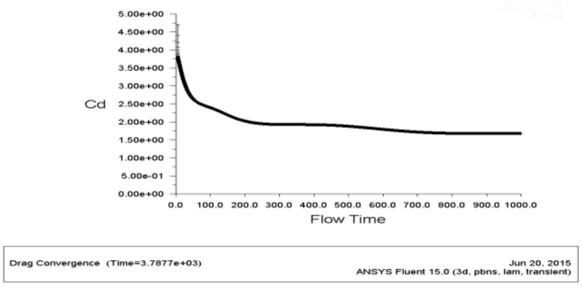


Figure 11: Cd for Top cylinder

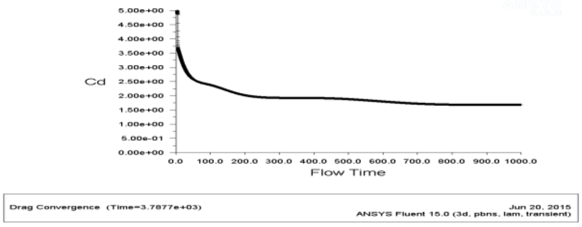


Figure 12: Cd for Bottom cylinder

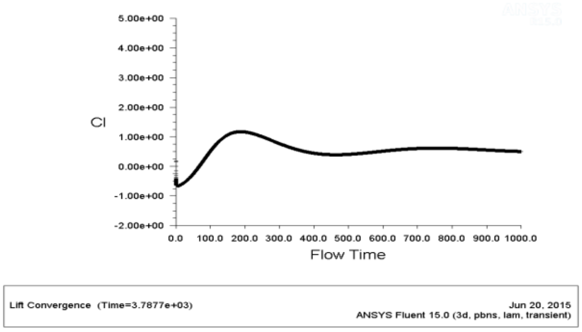


Figure 13: Cl for top cylinder.

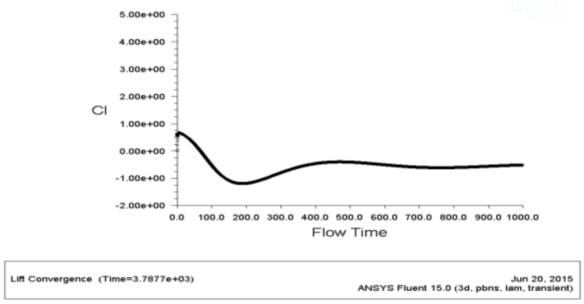


Figure 14: Cl for bottom cylinder.

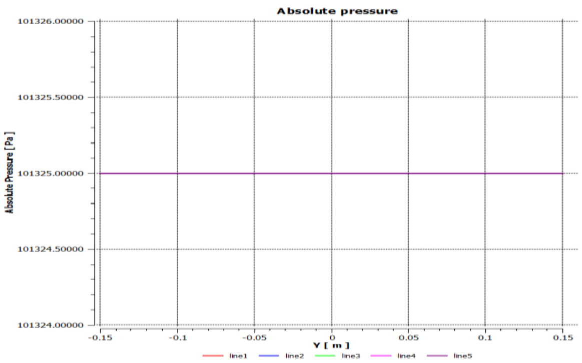


Figure 15: Absolute Pressure plot on five lines for two side by side cylinder at Re = 100

Due to small gap between the two cylinders a small portion of flow passes through this region. From figure 15 we can observe that the value of absolute pressure is same for all the lines since not much vortices and flow is observed behind the cylinders, hence not much change in the value of pressure. The value of the absolute pressure is 101325 Pa. The absolute pressure is same for all the x and y coordinates on the lines i.e. the co-ordinates are 0.05 to 0.25 m in x direction and -0.15 to 0.15 in the ‘y’ direction. Very small change of the scale of 0.002 Pa is observed which cannot be observed in absolute pressure graph.
Similarly 25 cases are done and there observations is shown below.

Discussion about the change in Reynolds number for T/D = 1.5

In this section we discuss about the effect of change of Reynolds number for the gap considered T/D = 1.5. From all the five Reynolds number simulation it is observed that as the Reynolds number increases the flow rotates near the cylinder and remains attached to the cylinders due to which the value of drag for cylinders reduces as the flow Reynolds number is increased. As the Reynolds number is increased the value of pressure behind the cylinder increases due to more energy in flow and generation of vortex. The maximum value of absolute pressure behind the cylinders is observed for a Re of 22000 on the line 4 behind the cylinder. From above five results we can infer that the case with Reynolds number of 22000 is giving higher pressure behind the cylinder.

Discussion about the change in Reynolds number for T/D = 2

In this section we discuss about the effect of change of Reynolds number for the gap considered as T/D = 2. Similar observations are seen as in case of T/D = 1.5. From all the five Reynolds number simulation it is observed that as the Reynolds number increases the flow rotates near the cylinder and remains attached to the cylinders due to which the value of drag for cylinders reduces as the flow Reynolds number is increased. For Reynolds number of 3900 and 15000 not much vortices are observed but the gap behind the cylinder is less compared to Reynolds number of 100. As the Reynolds number is increased the value of pressure behind the cylinder increases. The maximum value of absolute pressure behind the cylinders is observed for a Re of 22000 on the line 2 behind the cylinder. The maximum value of pressure is 101345 Pa. From above five results we can infer that the case with Reynolds number of 22000 is giving higher pressure behind the cylinders.

Discussion about the change in Reynolds number for T/D = 2.5

In this section we discuss about the effect of change of Reynolds number for the gap considered as T/D = 2.5. Similar observations are seen as in case of T/D = 1.5 an 2 From all the five Reynolds number simulation it is observed that as the Reynolds number increases the flow remains attached to the cylinders due to which the value of drag for cylinders reduces as the flow Reynolds number is increased. In this case the maximum vortex generation is observed only for Reynolds number of 10000 whereas for higher Reynolds number cases the gap behind the cylinder is narrower compared to Reynolds number of 100 but no much vortices are observed as in previous cases. This may be due to higher T/D ratio.

In this case also as the Reynolds number is increased the value of pressure behind the cylinder increases. The maximum value of absolute pressure behind the cylinders is observed for a Re of 22000 on the line 1 behind the cylinder. The maximum value of pressure is 101600 Pa. From above five results we can infer that the case with Reynolds number of 22000 is giving higher pressure behind the cylinders.

Discussion about the change in Reynolds numbers for T/D = 3

In this section we discuss about the effect of change of Reynolds number for the gap considered as T/D = 3. In this case it is observed that there is no significant change in the value of drag as the Reynolds number is increased, only in the case of re = 100 higher drag is observed other than that all cases are giving similar drag, because the T/D ratio is higher due to which there is no much effect of flow through the cylinders to each other. Since there is enough gap in between the cylinders for the flow to past smoothly. Vortex generation is observed for cases with Reynolds number 10000 and 22000, whereas in cases with re 100, 3900, 15000 the flow remains smooth. We observed the maximum value of pressure in this case is observed for Reynolds number of 3900 and 10000, whereas in previous cases we observed with the maximum pressure was only for Reynolds number 22000. And also the value of pressure is highest in all the cases considered till now. The value of Maximum pressure is 101410 Pa and is for Reynolds number 3900 and is on line number 1 which is 0.05m away from the cylinders in x direction, similar pressure of 101405 Pa is observed for Reynolds number of 10000.

Discussion about the change in Reynolds numbers for T/D = 4

In all the above five cases considered for T/D = 4 it is observed that the flow remains attached to the cylinders for higher value of Reynolds number compared to lower value of Reynolds number. In this case it is observed that there is no significant change in the value of drag as the Reynolds number is increased, only in the case of re = 100 higher drag is observed other than that all cases are giving similar drag, because the T/D ratio is higher due to which there is no much effect of flow through the cylinders to each other. For energy harvesting using the flow energy we need a point of higher pressure behind the cylinder, in this regard is observed that for cases with higher Reynolds number we get a higher value of pressure behind the cylinders compared to cases with lower Reynolds number. For all the five cases of T/D = 4 it is observed the for Reynolds number the absolute pressure is highest and the value is 101350 Pa, and the point of maximum pressure is 0.15 m away from the cylinder in x direction and 0.1 m below the central axis.

Discussion for results of all the cases with change in the gaps.

As mentioned earlier in this project five different gaps of T/D = 1.5, 2, 2.5, 3 and 4 were considered and each gap was simulated for five different Reynolds number. It is observed that for cases with lower T/D ratio of 1.5, 2 and 2.5 the value of drag decreases as the flow velocity increases but such trend is not observed for case with T/D ratio of 3 and 4 where the value of drag is high for only Re =100 case but for other all four cases with Reynolds number of 3900,1000,15000 and 22000 the value of drag remains more or less same. The value of pressure behind the cylinders is less in case of T/D ratio of 1.5 as can be seen in figure 16, and for T/D ratio of 3 the value of pressure is found to be highest for Reynolds number of 3900 and 10000 as seen in figure 16.

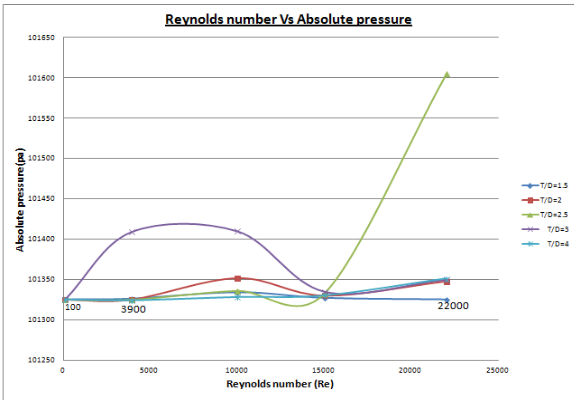


Figure 16: Comparison for different T/D and Re

Discussion about the x and y co-ordinates for different Reynolds number and different T/D ratio.

The below two figures shows the graph of Reynolds number vs. the x and y co-ordinates for all the five T/D ratio models. From figure 17 it is observed that as the Reynolds number is increased the 'x' co-ordinate value for maximum pressure is near to the cylinders, this is due to the fact that flow remains attached to the cylinders for higher Reynolds number flow. The 'x' co-ordinate for maximum pressure of 101600 Pa is 0.05 meter. For T/D ratio of 3 the 'x' co-ordinate is found to be near to the cylinders whereas cases with T/D = 1.5, 2 and 2.5 the 'x' co-ordinate is far from the cylinders. For T/D = 4 the 'x' co-ordinate is in between as seen in the figure.

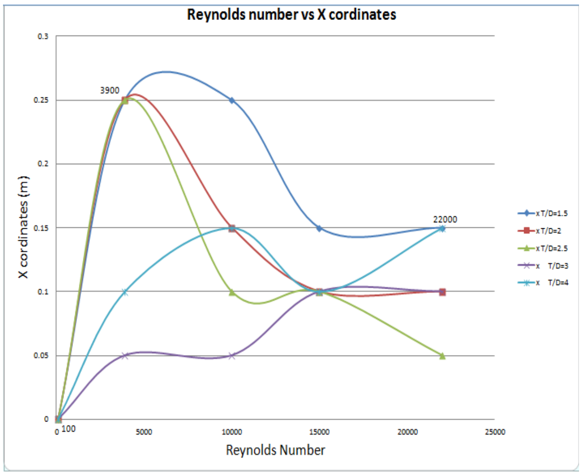


Figure 17: Reynolds number vs 'X' co-ordinate for Absolute pressure.

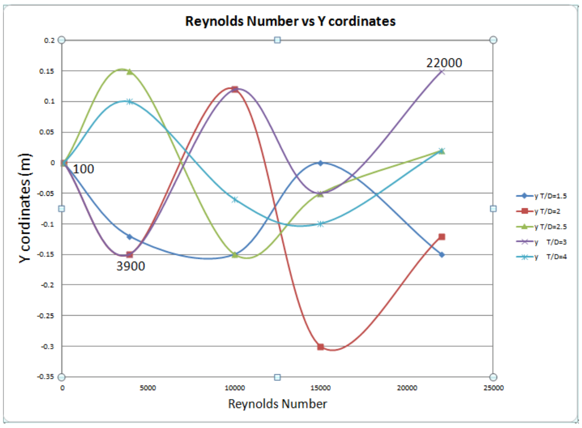


Figure 18: Reynolds numbers vs 'Y' co-ordinates for absolute pressure.

CONCLUSIONS

In this project, a scheme based on control-volume method has been used to simulate the flow over one and two side by side cylinders in laminar and turbulent flows. Following conclusions are drawn.

1. For single cylinder with Re 100 the drag co-efficient is found to be 1.27 by using a mesh of 190*190*8 in the x,y and z direction which closely matches with the experimental work of DEHKORDI Behzad Ghadiri and numerical work of Maneghini et al. Hence 190*190*8 is considered as grid independent mesh.
2. For two cylinder it is found that the mean drag co-efficient reduces as the gap between the two cylinders increases. Also, it is found that the drag is more for laminar cases for all the gaps compared to turbulent cases.
3. When we compare the value of pressure behind the one and two cylinder it is found that the value of pressure behind the one cylinder is less compared to the value of pressure behind the two cylinders with T/D of 1.5, 2, 2.5, 3 and 4. For single cylinder with Reynolds number of 100 the value of pressure behind the cylinders is 101324.998 Pa whereas in two cylinder with T/D =1.5 and Reynolds number 100 the value of pressure is 101324.999 Pa and for T/D = 2, 2.5, 3 and 4 the value is 101325.00135, 101325.0014, 101325.0075 and 101325.0002 Pa respectively. The changes are negligible and hence cannot be observed in the graphs shown in chapter 4. For single cylinder with Reynolds number 10000 the maximum absolute pressure behind the cylinder is 101327 Pa and for T/D = 1.5, 2, 2.5,3 and 4 the maximum absolute pressure value is -101334 Pa, 101330 Pa, 101337 Pa, 101410 Pa and 101330 Pa respectively. Hence we can conclude that the value of absolute pressure behind the two cylinders is more compared to the values of pressure behind a single cylinder for both laminar and turbulent cases.
4. For comparison of T/D ratio it is observed that for cases with lower T/D ratio of 1.5, 2 and 2.5 the value of drag decreases as the flow velocity increases but such trend is not observed for case with T/D ratio of 3 and 4 where the value of drag is high for only Re =100 case but for other all four cases with Reynolds number of 3900,1000,15000 and 22000 the value of drag remains more or less same.
5. For the pressure values behind the cylinders the lines were created and the values of absolute pressure is plotted. It is found that the pressure value increases as the Reynolds number of the flow increases, hence for 22000 Reynolds number the pressure behind the cylinders is found to be maximum for cases with T/D ratio 1.5,2 and 2.5.
6. From figure 17 it is observed that as the Reynolds number is increased the x co-ordinate for maximum pressure is near to the cylinder, this is due to flow being attached to the cylinders for higher Reynolds number flow. The 'x' co-ordinate maximum pressure for T/D ratio 3 is found to be most near to the cylinders whereas cases with T/D = 1.5,2 and 2.5 the 'x' co-ordinate is far from the cylinders. For T/D = 4 the 'x' co-ordinate is in between as seen in figure 4.168. For the 'y' co-ordinate there is no any particular pattern which is followed since fluctuations are observed for all the T/D ratio with maximum pressure usually found either above or below the cylinder and in some cases values are observed above and below the cylinders.
7. From all the models considered it is found that the model with T/D = 2.5 and Reynolds number 22000 the value of absolute pressure is 101600 Pa, which is highest in all the cases considered. This value is found on line 1 and the (x, y) co-ordinates are (0.05 m, 0.02 m) Next peek value is for T/D = 3 and for a Reynolds number of 3900 the value of pressure is 101410 Pa, this value is found on line 1 , and this peak value occurred at 0.15 m below the centre axis which is a symmetric axis between the two cylinders. The (x,y) co-ordinate for this value is (0.05 m, -0.15 m) respectively. Next peak value is observed for again T/D = 3 and Reynolds number of 10000, the value is 101405 Pa on line 1 and is 0.1 m below the central axis. For T/D = 4 the peak value 101350 Pa is observed on line 1, 2 and 3.
8. The above mentioned models with corresponding positions are suitable to install piezoelectric sensor to generate the piezoelectricity by harvesting the mechanical energy from the flow.

Finally, it is important to mention that the implemented solver is sufficiently applicable for simulation of flow over complex geometries such as two circular cylinders. Also, the LES turbulence model has been used in this thesis which has given satisfactory results as seen in the validation cases.

ISSN: 2278-0181
Vol. 4 Issue 10, October-2015

Future Works

The following recommendations are provided which can be considered in future analysis.

1. In this project flow around two circular cylinders is considered, in future flow over Square shape, Elliptical shape, drop shape etc. type of geometry can be taken for flow behind these geometry and calculation of pressure, vortices behind the cylinders.
2. Flow can be sent at different angle of attack to observe the difference in drag, pressure.
3. CFD analysis by considering the piezoelectric effect can be performed and result validated with these pressures.
4. Large Eddy Simulation turbulence model is used in this project which is itself a complex and computationally sensitive models but direct numerical Simulation (DNS) turbulence model can be used which solves the equation directly without modelling the smaller eddies.

REFERENCES

[1] Abarbanel, S. S. 2010. Secondary frequencies in the wake of a circular cylinder with vortex shedding. Hampton, Va.: National Aeronautics and Space Administration, Langley Research Center.

[2] Anderson, D. John,. (1995). Computational fluid dynamics , First edition. Printed in Singapore by (a.d.john 1995)

[3] Aradag, S. 2009. Unsteady turbulent vortex structure downstream of a three dimensional cylinder. J. of Thermal Science and Technology 29(1): 91-98.

[4] Berselli, L. C., Iliescu, T. & Layton, W. J. 2005. *Mathematics of Large Eddy Simulation of Turbulent Flows*. Springer.

[5] Deng, j., An-lu, R & Wen-qu, C. 2005. Numerical simulation of flow-induced vibration on two circular cylinders in tandem arrangement. *Journal of Hydrodynamics*, Ser. B, 17(6): 660-666.

[6] Deardorff, J. W. 1970. A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Numbers. *Journal of Fluid Mechanics* 41(02): 453-480.

[7] Devaud, C., J. Weisinger, D. Johnson & E. Weckman 2009. Experimental and numerical characterization of the flowfield in the large-scale UW live fire research facility. International journal for numerical methods in fluids 60(5): 539-564.

[8] Dehkordi, B. G., H. S. Moghaddam & H. H. Jafari 2011. Numerical simulation of flow over two circular cylinders in tandem arrangement. *Journal of Hydrodynamics, Ser. B* 23(1): 114-126.

[9] Elsenaar, A. 2005. Reynolds number effects in transonic flow. Neuilly sur Seine, France: AGARD.

[10] Franke, J. & W. Frank 2002. Large eddy simulation of the flow past a circular cylinder at ReD=3900. *Journal of Wind Engineering and Industrial Aerodynamics* 90:1191–1206.

[11] Foden, G. 2010. Turbulence. New York: Alfred A. Knopf.

[12] Grinstein, F. F. 2007. Implicit large eddy simulation: computing turbulent fluid dynamics. Cambridge: Cambridge University Press.

[13] Guang-wang, L., R. An-lu & C. Wen-qu 2004. An ALE method for vortex-induced vibrations of an elastic circular cylinder [J]. *Acta Aerodynamica Sinica* 3: 008.

[14] John, V. & M. Roland 2007. Simulations of the turbulent channel flow at Reτ= 180 with projection-based finite element variational multiscale methods. International Journal for Numerical Methods in Fluids 55(5): 407-429.

[15] Kang, S. 2003. Characteristics of flow over two circular cylinders in a side-by-side arrangement at low Reynolds numbers. Physics of Fluids (1994-present) 15(9): 2486-2498.