ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 8, August 2017

Numerical Analysis of Drag And Flow Field of Multiple Bluff Bodies

[1] Suja T P
[1] Assistant Professor, Department of Civil Engineering, Manipal Institute of Technology, Manipal.

***Abstract:*— In and around the offshore industry, there are drastic developments in the construction of structures as well as in the analysis. In the design and analysis of most of the offshore structures, effect of other obstructions in the field is considered as a minor factor. But there is some influence on the force acting on the structures due to the presence of other neibouring structures. The concept of bluff bodies which is characterized by large amount of flow separation in the field of offshore structures. This work mainly focuses on the flow past bluff bodies. A 2-Dimensional steady state is considered for the simulation in the computational fluid dynamics. The flow domain considered is rectangular with appropriate boundary conditions. The fluid is assumed to be incompressible. k- ε turbulent model is used in the present work. Two dimensional Navier- Stokes equation is solved. Flow around single circular bluff body is simulated and the corresponding drag coefficient variation with respect to Reynolds number is studied. It is extended to two, three and four bluff bodies in the fluid flow. For the multiple bluff bodies, gap between them are changed to get the effect of gap on flow field. Reynolds number range chosen is 104 to107.**

***Index Terms*—bluff bodies, drag and flow field, k- ε turbulent model**

INTRODUCTION

Based up on the type of drag experienced on a body when it is in a fluid flow, the body can be stated as streamlined and bluff bodies. For a bluff body, the pressure drag experienced on the body will dominate the viscous drag. On the other hand, if the viscous drag dominates than the pressure drag, the body is treated as streamlined body. Generally bluff bodies look like a cube, cylinder, or an airfoil at large angle of attack and the streamlined bodies are like fish or airfoil with very small angle of attack. Because of their shape, bluff bodies are characterized by flow separation at the boundary layer. For the case of a general airfoil body, at small angle of attack, the boundary layers on the top and bottom surface experience only mild pressure gradients, and they remain attached along almost the entire chord length. The wake is very small, and the drag is dominated by the viscous friction inside the boundary layers. However, as the angle of attack increases, the pressure gradients on the airfoil increase in magnitude. The adverse pressure gradient on the top rear portion of the airfoil may become sufficiently strong to produce a separated flow. This separation will increase the size of the wake, and the pressure losses in the wake due to eddy formation. Therefore, the pressure drag increases. At a higher angle of attack, a large fraction of the flow over the top surface of the airfoil may be separated, and the airfoil is said to be stalled. At this stage, the pressure drag is much greater than the viscous drag. Moreover, the flow separation is highly depended on the Reynolds number, as at large Reynolds number, the drag is dominated by the pressure loss in the wake region. In most of the experimental and numerical studies, cylindrical bodies are taken as the general bluff body shape.

It is very much important in the flow field analysis and in the fluid-structure interaction studies to use the bluff body shapes. Moreover, to study the flow induced vibrations the bluff bodies are very much used in researches. The propensity of bluff bodies to vibrate in fluid is very significant.

Application of bluff body

Flow over a bluff body is a common occurrence associated with fluid flowing over obstacles or with the movement of a natural or artificial body. There are many applications for the bluff bodies in the offshore field especially in the flow past marine risers, legs of semisubmersibles etc. At much lower Reynolds numbers, the flow over a bluff body is highly viscous, and the force exerted on the body is mainly attributed to skin friction. However, when the Reynolds number exceeds a critical value, vortex shedding occurs in the wake, resulting in a significant pressure drop on the rear surface of the body.

Importance of Reynolds number

In offshore hydrodynamics context, the flow is highly turbulent and the Reynolds number is very high. So, the flow instabilities are very high. Taking the case of the offshore platforms, such as the TLP (Tension leg platforms) or semisubmersibles, the flow around legs are disturbed highly. This will influence the forces (especially the drag force) acting on the platform legs. Moreover, analysis of flow field in a proper way can lead to a cost-effective design of the offshore platforms. Taking the platform as a multiple bluff body and carrying the simulation and analysis of the flow around the legs leads to get the change in the force coefficients and thereby it can reach the above-mentioned goal. This can also be extended to the study of flow interference of moored ships and offshore platforms.

Progressive transition of the flow field around the single cylinder from laminar to turbulent is driven by the value of Reynolds number. Moreover, the drag coefficient is closely related to these transitions. Many more researches have been done on the effect of Reynolds number on the force acting on the cylinder and on the nature of flow past a single cylinder. For a Reynolds number, less than 400, the flow is fully laminar. Shear layer transitions occurs at a Reynolds number of 10^5 to 10^6 and at a Re > 10^6 the flow becomes fully turbulent. The effect of turbulence can be evaluated using turbulence intensity I_u and turbulence length scale L_s.

Flow field around different bluff bodies

Different shaped bluff bodies are used in many of the contests in offshore industry. Especially when considering the platform hull portion, combinations of circular and rectangular/ square cross sections are used. It is so very important to know about the flow field around different bluff bodies. When compared with a smooth circular cylinder section, the square and rectangular cross sections are subjected to a high degree of flow separation because of their sharp edges. But in general, the flow around square cylinder is like the flow around circular cylinder and the wake created around a square cylinder is wider than that created around circular cylinder. When more than two bluff bodies are placed in a fluid flow, the flow pattern of one body is highly disturbed by the other and vice versa.

This work focuses to simulate the flow field past single, dual and multiple bluff bodies using computational fluid mechanics. Also, to compare the effect of gap between the bluff bodies on drag coefficient.

Theoretical formulation

The basis of the numerical study using CFD is the Navier-Stokes (NS) equation. For the present work, NS equation used is for two-dimensional incompressible fluid flows. The fluid is assumed to be incompressible. The analysis is fully in a steady state solver and two dimensional. Only k-ε turbulent model is used in the present work.[4]

Methodology

Flow around single circular bluff body is simulated and the corresponding drag coefficient variation with respect to Reynolds number is studied. It is extended to two, three and four bluff bodies in the fluid flow. For the multiple bluff bodies, gap between them are changed to get the effect of gap on flow field. Reynolds number range chosen is 10^4 to10^7.

Results and analysis

The flow field and drag of single, dual, triple as well as for four bluff bodies are analyzed for different Reynolds number and for different gap between the bodies. Circular cross sections are selected for the numerical study. The parameter of primary interest is the drag coefficient and change in drag coefficient according to the change in Reynolds number and with respect to change in gap was taken into account. The drag coefficient is given by the equation

$$C_d = \frac{F_d}{\frac{1}{2}\rho A V^2} \qquad \text{Eqn 1}$$

Where, F_d is the drag force, ρ is the density of fluid, A is the characteristic dimension and V is the free stream velocity.

A two-dimensional mesh was generated for the solution. Using it, an unstructured mesh of triangular element (three noded) in a rectangular domain was created. The boundary conditions applied for the mesh in all case are inlet boundary (at the upstream), outflow (at the downstream), symmetry (at the side), and wall boundary (for the bluff body).

II. FLOW PAST SINGLE CIRCULAR BLUFF BODY

For the present analysis, an unstructured triangular mesh is generated with a rectangular domain of size 77.3mx

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 8, August 2017

46.2 m with the bluff body diameter equals to 2m. For a varying Reynolds number ranging from 10^4 to 10^5 is taken [1] for the analysis. In the turbulent region, a standard k-ε model is used. The analysis is carried out for 2 dimensional, steady state incompressible fluid. Variation of drag coefficient with respect to the Reynolds number is plotted. Validation of the results is done with the numerical study performed by Gandhi et al (2004).

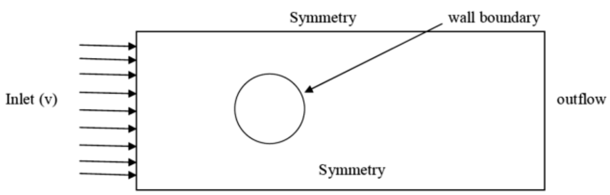

Fig .1 General sketch of domain and boundary conditions adopted

Table (i)

number	Reynolds number (Re)	Obtained C_d
1	10^4	0.850
2	$2.5x10^4$	0.741
3	$5x10^4$	0.670
4	$7.5x10^4$	0.658
5	10^5	0.653

From the table (i), it is observed that the C_d value varies inversely with the Reynolds number. It also implicates that there is not much separation in the flow field near the bluff body for low Reynolds number.

III. FLOW PAST TWO CIRCULAR CYLINDERS

To carry out the computations of flow past two circular cylinders; both tandem as well as side-by-side arrangement is considered. Upstream boundary is taken to be at distance of 21.3 times the radius away from the center of upstream circular body. A constant gap of 3times the diameter (D) is taken for both tandem and side-by-side arrangement. An unstructured triangular mesh is generated with proper boundary condition. A fine meshing is adopted near the cylinder surface to get the variation of flow field more accurately. Reynolds number value adopted is 100 to 200. [3]

When two cylinders are placed in tandem arrangement, the flow scenario around the bodies changes drastically when compared with that of single cylinder placed in fluid flow in the same conditions. The presence of the upstream cylinder changes the velocity, pressure as well as the force on the downstream cylinder. Hence it is observed that there exists a reduction in the drag coefficient.

Table (ii) Value of C_d for circular cylinder placed in tandem arrangement

No:	Reynolds number	C_d (upstream cylinder)	C_d (downstream cylinder)
1	100	1.37	1.09
2	200	0.83	0.685

When the cylinders are placed in a side by side arrangement, the value of C_d obtained at a Re of 100 for the upper cylinder is 1.383 and that for the lower cylinder is 1.387. As the Re increase to a value of 200, the corresponding value of C_d changed to 0.563 and 0.566 respectively.

Table(iii) Variation of C_d for cylinders in side by side arrangement

No:	Reynolds number	C_d (upper cylinder)	C_d (lower cylinder)
1	100	1.383	1.387
2	200	0.563	0.566

IV .FLOW PAST THREE CYLINDERS

A rectangular domain of dimension 300m x 200m is generated for the study. Unstructured mesh of three noded triangular elements are used. Diameter of the circular bluff body is taken as 14.14m. The bluff bodies are placed in the form of equilateral triangle [5]. The analysis is done for different gap distance ranging from D, 2D, up to 6D, where D is the diameter of the cylinder and for a Reynolds number ranging from 10^7 to $4x10^7$. The boundary conditions applied are the inlet, outflow, symmetry and wall boundaries for the upstream, outlet, sides of domain and for the cylindrical body respectively.

IFERP
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 8, August 2017

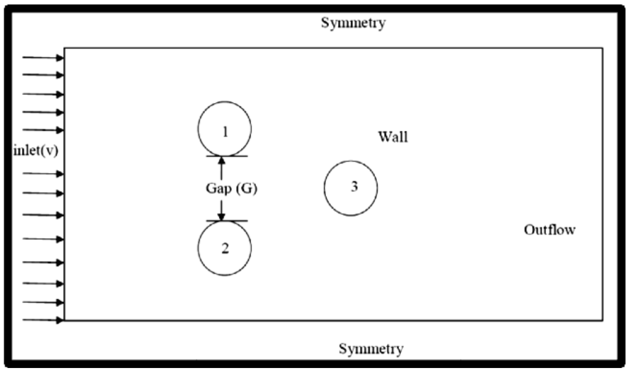

Fig (ii)

Table (iv) Variation of drag coefficient with respect to change in gap for three circular cylinders. ($Re=10^7$, velocity =0.95m/s)

No:	Gap (G)	C_d		
		Cylinder 1	Cylinder 2	Cylinder 3
1	D	0.315	0.3154	0.3051
2	2D	0.252	0.252	0.254
3	3D	0.309	0.308	0.303
4	4D	0.275	0.275	0.272
5	5D	0.317	0.314	0.308
6	6D	0.321	0.324	0.318

From the table (iv), it is noticed that there is not much variation for the drag coefficient for cylinder 1 and 2. But the value is decreased for cylinder 3. The value of drag coefficient decreases till gap attains a value of 3D after which it increases slowly. This happens at a gap distance between 3D and 4D.

Table (v) Variation of drag coefficient with respect to change in gap between cylinders for three circular cylinders. (Re=2x107, velocity =1.5m/s)

No:	Gap (G)	C_d		
		Cylinder 1	Cylinder 2	Cylinder 3
1	D	0.682	0.684	0.649
2	2D	0.582	0.582	0.578
3	3D	0.523	0.522	0.513
4	4D	0.681	0.681	0.676
5	5D	0.759	0.7588	0.756
6	6D	0.468	0.467	0.482

From the close analysis of the tables (iv), (v) and (vi), it could be concluded that the value of c_d is optimum for the three cylinders when the gap between them is 3D**.**

Table (vi) Variation of drag coefficient with respect to change in gap between cylinders for three circular cylinders. ($Re=4x10^7$, velocity =2.721 m/s)

No:	Gap (G)	C_d		
		Cylinder 1	Cylinder 2	Cylinder 3
1	D	1.184	1.183	1.112
2	2D	1.098	1.093	1.099
3	3D	1.272	1.275	1.245
4	4D	1.190	1.189	1.211
5	5D	1.144	1.147	1.135
6	6D	1.249	1.248	1.239

V FLOW PAST FOUR CYLINDERS

The four cylinders are analysed in an approprite domain with the same boundary conditions which is used for the analysis of three cylinders. In the particular domain the gap distance as well as the Reynold's number are varried and the parameter drag coefficient is obtained.

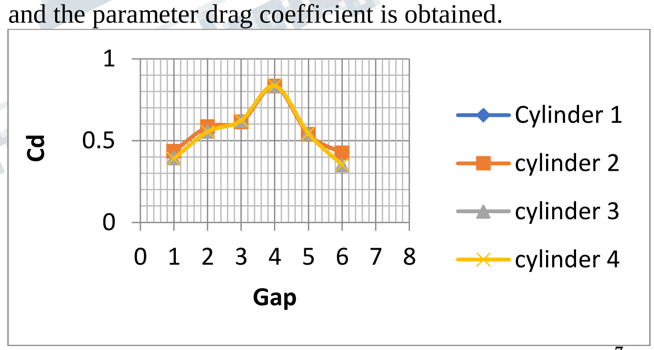

Fig (iii) Cd v/s gap for four circular cylinders ($Re = 10^7$)

The variation of drag coefficient with respect to the gap as well as Reynolds numbers inferes that the optimum gap for the four cylinders should be 3D to 4D for all range of Reynolds number selected

VI .CONCLUSION

Flow field past single, dual, three and four circular bluffbodies in fluid flow are simulated successfully. It is generally observed that the value of drag coefficient

IFERP
connecting engineers... developing research
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 8, August 2017

varies with Reynold's number and gap distance in all cases and all arrangements. Variation of C_d of downstream cylinder for three bodies in fluid flow is prominent for a gap of 3D. It gives 2% reduction in C_d for the third cylinder for all the Reynolds numbers considered. C_d for upstream cylinder is lesser than that of downstream cylinder by a value of 1%. The design and analysis of triangular or four legged TLP s or semisubmersibles, the variation of C_d could be taken in to account.

REFERENCES

[1] B A Younis, P Teigen, V P Przulj; 'Estimating the hydrodynamic forces on mini TLP with computational fluid dynamics and design- code techniques'; Ocean engineering 28 (2001) 585-602

[2] B.K Gandhi et al, 'Effect of bluff body shape on vortex flow meter performance' Indian jrnl of engineering and material science (2004), volm 11, pp 378-384P.

[3] J.R. Meneghini et al, 'Numerical simulation of flow interference between two circular cylinders in tandem and side by side arrangements' jrnl of fluids and structures (2001)

[4] M Zhao et al, 'Hydrodynamic forces on dual cylinders of different diameters in steady currents, Journal of fluids and structures (2007)

[5] S. Chandrasekaran , A.K. Jain; 'Influence of hydrodynamic coefficients in the response behavior of triangular TLPs in regular waves' ; Ocean Engineering 31 (2004) 2319–2342