Indian Journal of Fundamental and Applied Life Sciences ISSN: 2231– 6345 (Online)
An Open Access, Online International Journal Available at www.cibtech.org/sp.ed/jls/2015/02/jls.htm
2015 Vol. 5 (S2), pp. 210-216/Amjadimanesh and Ghalatian
Research Article

NUMERICAL STUDY OF WIND FORCE ON VERTICAL PIPES

***Amjadimanesh A. and Ghalatian M.**
Hampa Energy Engineering & Design Company (HEDCO), Shiraz, Iran
**Author for Correspondence*

ABSTRACT
Wind plays an important role on forces and stresses of pipes and equipments which are connected to them. As most of petrochemical industrials, include high elevation equipments, their riser pipes are exposed to wind. The drag force of wind may change the value of stresses and forces. Therefore, precise calculation of wind force on these pipes is much more critical. ASCE#7 2005 standard suggests values to take wind force into consideration. On the other hand, continuity and momentum equations are solved by computational fluid dynamic to determine drag force due to wind on pipes. In order to have better accuracy, five different wind speeds are chosen to study and their results are compared withASCE#7 standard. Although standard includes some safety factors to be on the safe side, numerical simulation would help have a better estimation of drag force on pipe due to wind.

Keywords: *Wind, Numerical Simulation, Riser Pipes, Drags Force*

INTRODUCTION
Understanding the drag characteristics of objects in fluid flow is essential for engineering design aspects, such as to reduce the drag on piping, automobiles, aircrafts, buildings, chimneys, towers.
Drag force is the net force exerted by a fluid on body (here a pipe) in direction of flow due to the combined effects of wall shear and pressure forces (Cangel and Climbala, 2006). There are many papers with subject of drag on pipes. Some of previous works are as follows:
Masami and Takaya (2012) studied numerically the flow past a circular cylinder. A wide range of reynolds number has been considered in order to study influence of reynolds number on the numerical results.The results demonstrated that CFD approach is in a good agreement with experimental data. Therefore, it will be possible for a mechanical engineer to analyze the problem of fluid flow around a circular cylinder with CFD.
Bruschi and Coworkers (2003) and in an experimental work, calculated drag coefficient of a cylinder from data obtained by performing tests in a water tunnel at three different flow velocities. Two methods of analysis were used to calculate drag measurements on the cylinder. In theory, it is explained that for a smooth cylinder under laminar flow, the coefficient of drag is independent of the velocity within the velocity range under analysis. Brower (2006) studied drag force acting on a cylinder in a steady flow field, by placing a cylindrical rod into a flow field and measuring the velocity of the fluid both before and after the cylinder. This drag force was found to increase exponentially as the wind speed increased. This increase was due to the fluid interacting with the surface of the obstructing object and creating shear force due to the viscosity of the fluid. With a lower viscosity fluid, or a smoother surface, the drag force could be decreased. Butt and Egbers (2013) considered flow over circular rough cylinders. Investigations were performed in a subsonic wind tunnel to observe the effect of hexagonal patterns on the flow of air at Reynolds numbers ranging from 3.14E+04 to 2.77E+05. The investigations revealed that a patterned cylinder with patterns pressed outwards (can be referred as hexagonal bumps) has a drag coefficient equal to 65% of the smooth one. A circular cylinder produces large drag due to pressure difference between upstream and downstream. The difference in pressure is caused by the periodic separation of flow over surface of the cylinder.
Mallick and Kumar (2014) studied the drag force on cylindrical bodies with varying cylinder diameters and air velocity. The drag coefficient of a cylinder was calculated from data obtained by performing tests in an air flow with varying flow velocities and diameters as 12.5mm, 15 mm, 20 mm, and 25 mm. They concluded the drag force increases with increase in diameter of the cylinder. Also, for a cylinder of particular diameter, drag force has been found to increase with increase in air velocity.

In this paper, a 3-dimentional vertical pipe is simulated numerically by a commercial CFD (Computational Fluid Dynamics) code (FLUENT) to obtain the fluid flow around it and calculate the drag force and coefficient. In order to have an integral visualization and better accuracy, five different speed of wind are chosen.
Then, the results are compared with data which are available on ASCE#7 standard. This would help us make a sense about the accuracy of standard.

This contribute to have more accurate stress analysis of piping around towers which are critical because of exposing to wind, high temperature, and low allowable load of nozzles.

Numerical Scheme and Governing Equations

The flow around vertical pipe is laminar (in all speeds which are taken into consideration Reynolds number is less than 5×10^5).

The flow is assumed to be steady and incompressible (in this air condition, the Mach number of the flow is much lower than 0.3 (White, 2010).

Governing equations include conservation of mass and momentum equations (eq. 1, 2). No slip boundary condition is utilized on the walls.

The atmospheric conditions are set as the boundary conditions at the outlet of the flow field and velocity inlet boundary condition is applied in inlet of the field. Air properties has been considered in 21°C temperature.

The equations of Conservation; mass, momentum are as followed (respectively) (Versteeg and Malalasekera, 2007).

$$\nabla.(\rho\vec{v}) = 0 \tag{1}$$

$$\nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) \tag{2}$$

Drag force is calculated by shear stress which is applied on the external wall of the pipe. Drag force is divided to two part. First, skin drag which is due to friction between pipe and fluid (for the sake of viscosity of fluid). Second, pressure drag which is because of the pressure difference between upstream and downstream flow around the pipe (eq. 3). Drag coefficient also obeys this division (eq. 4).

$$F_D = F_{D\,skin} + F_{D\,pressure} \tag{3}$$

$$C_D = C_{D\,skin} + C_{D\,pressure} \tag{4}$$

Drag coefficient is determined by the following equation (eq. 5):

$$CD = \frac{F_D}{\frac{1}{2}\rho V^2 A} \tag{5}$$

Where CD denotes the drag coefficient, ρ the density of the fluid, V the speed of the fluid and A is the cross-sectional area opposing the flow. F_D also is the drag force (Cengel and Cimbala, 2006).

Geometry
Dimensions of the pipe considered in the Figure 1.

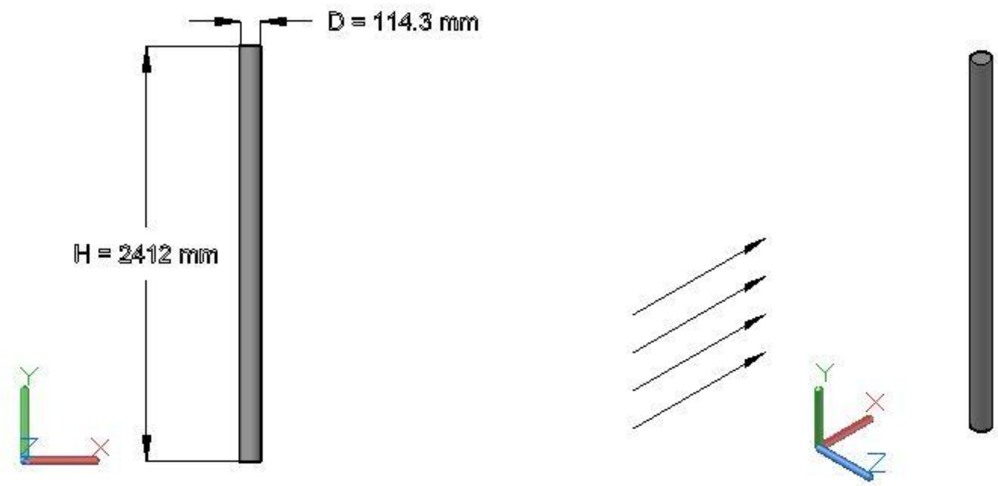
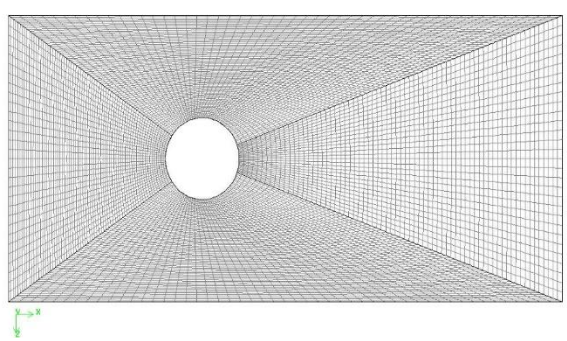


Figure 1: Geometry

Grid Generation

The grid system is shown in Figure 2. Figure 2(a) shows the overall scheme of grid system. Figure 2(b) demonstrates grid system in top view. Hexahedral mesh (structure grid) is used in for flow field to achieve better accuracy. The quality of mesh generation is checked of aspect ratio and skewness. The aspect ratio of 100 percent of grids are in the range of 1 and 5 and the skewness of about 90 percent of grids are lower than 0.5 (High quality grid generation).
In order to capture the details of the flow field, around the walls (pipe) and high velocity and pressure gradient regions the mesh number is increased.



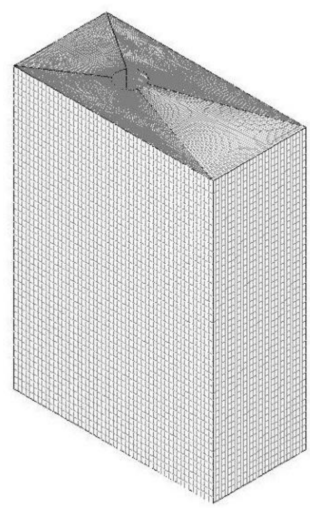


Figure 2(a): Grid system **Figure 2(b): Scheme of grid in top view**

Grid Study

It must be shown that the numerical results are independent of grid size. For this means, mesh number is increased by a factor of two, and variation of total drag coefficient is computed (results have obtained in velocity 40m/s). Figure 3 shows the variation of total drag coefficient for a set of mesh numbers. As shown, the results between 560000 and 1120000 mesh number are close to each other.
As shown in the Table. 1, variation of results by increasing the mesh number from 560000 to 1120000 is very small and the percent change is 0.27%. Therefore regarding to computational cost, the 560000 mesh

number is appropriate and sufficient (it is noted that grid study results for other velocities are very similar to these results, so they have been eliminated to avoid repetition).

Table 1: Grid Study

Mesh number	CD	Error percentage (%)
140000	1	-
280000	1.06	5.66
560000	1.1	3.64
1120000	1.103	0.27

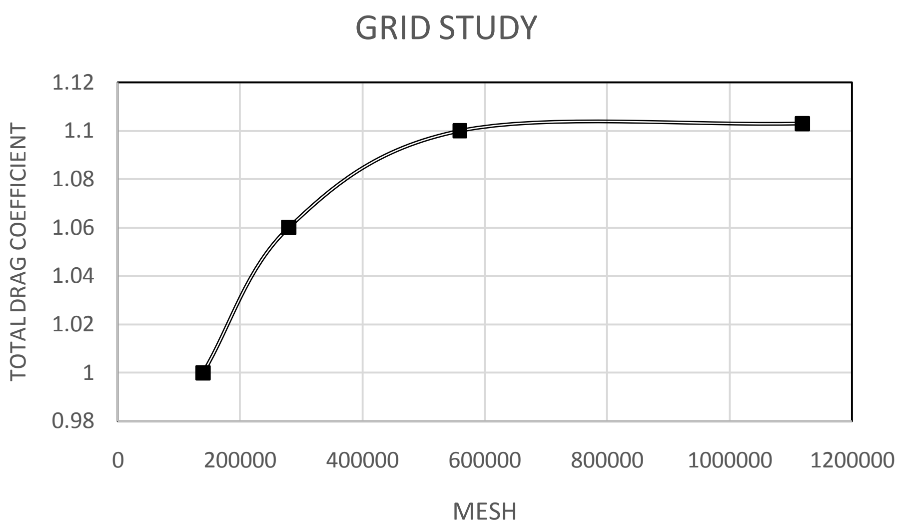


Figure 3: Grid study

Validation

Experimental data reported in reference (Cengel and Cimbala, 2006) are compared with numerical results of different speeds to validate numerical simulation. Table 2, presents drag coefficient of experimental work and drag coefficient which calculated by numerical simulation. According to table, numerical and experimental results are in quite good agreement and the maximum error occurs is about 15% (Amjadimanesh and Ghalatian, 2014). Regarding to the fact that in these speeds the Reynolds number is so close to critical Reynolds number, the flow field is in transient region which causes kind of instability between numerical simulation which has been done by laminar flow assumption, and experimental data. Therefore, having a little high error percentage in these speeds, should not cause any concern about the validity of numerical simulation.

Table 2: Validation

Velocity (m/s)	CD total	CD theatrical	Error percentage (%)
35	1.05	0.95	10
40	1.1	0.95	15
45	0.99	0.95	4
50	1.1	0.95	15
55	1.091	0.95	14

RESULTS AND DISCUSSION

Results

Flow field characteristics of the flow over the pipe have been shown in Figure 4. Figure 4 demonstrates the velocity contour around the pipe in 40 m/s inlet velocity. Fluid enters to flow field with uniform velocity. Velocity is zero in front of the pipe. The flow passes around the pipe and the maximum velocity occurs in 90 degree of the pipe and is about 63.2 m/s (according to Bernoulli equation, the lesser in pressure, the more in velocity). After that, separation occurs and the flow returns back. In the Figure 5 maximum pressure is about 1190 gauge pressure. Maximum pressure occurs in front of the pipe where the velocity is near zero and minimum pressure occurs in 90 degree. There is also a low pressure region in downstream of the field. In Figure 6 the velocity vectors of the flow have been shown.
In the back of the pipe, there are many wakes with low velocity which rotate in counter direction of each other alternatively.

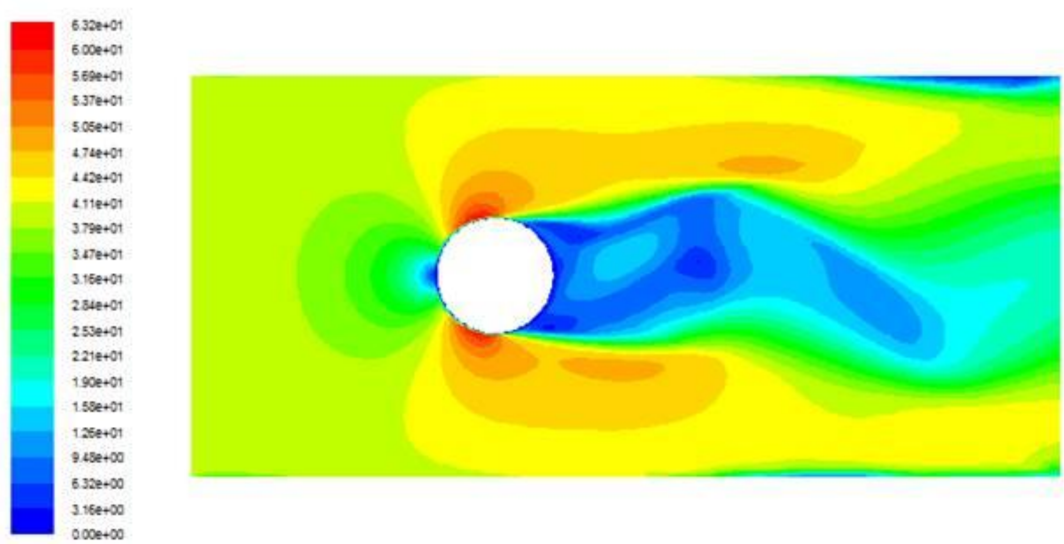


Figure 4: Velocity contours

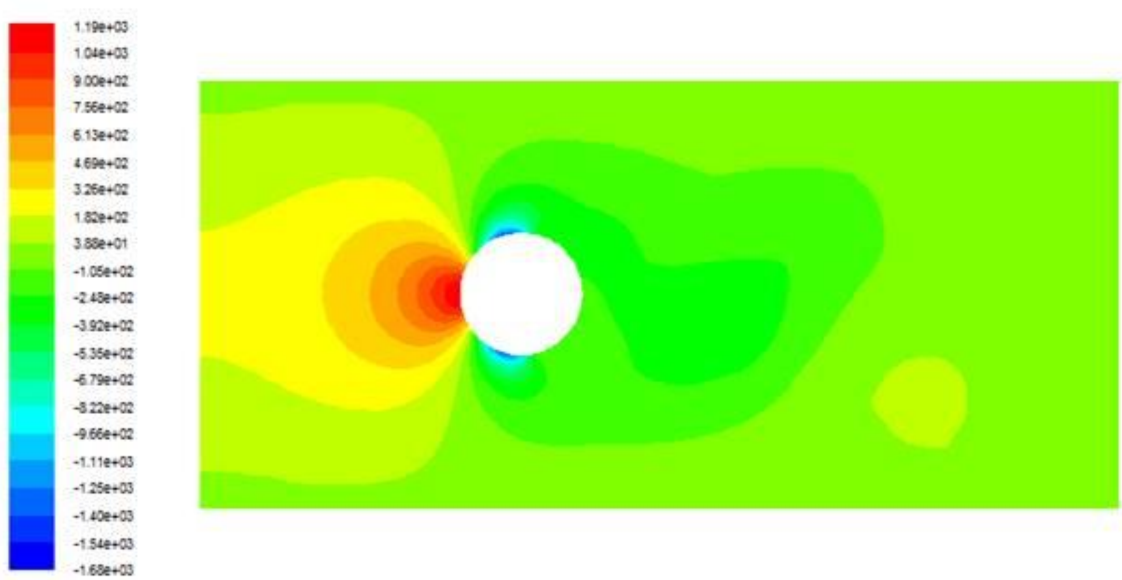


Figure 5: Pressure contours

Research Article

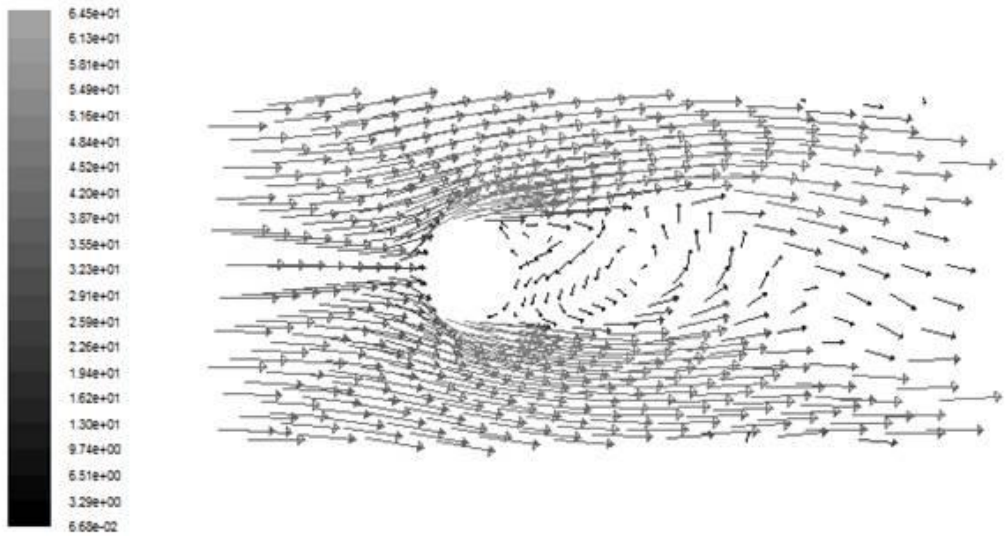


Figure 6: Velocity vectors

Figure 7 demonstrates the drag forces vs wind speeds and numerical result have been compared with data calculated by Caesar II and base on the ASCE#7 standard (Kharaiti, 2005) which is used in most of stress analysis of pipes. According to Figure the drag force of numerical simulation and standard increase by the increase in wind speed.But the rates of increase are different. The slope of the numerical result is higher which leads to higher drag force in higher wind speeds. Drag forces of numerical study are about 1.5 times higher than standard values.

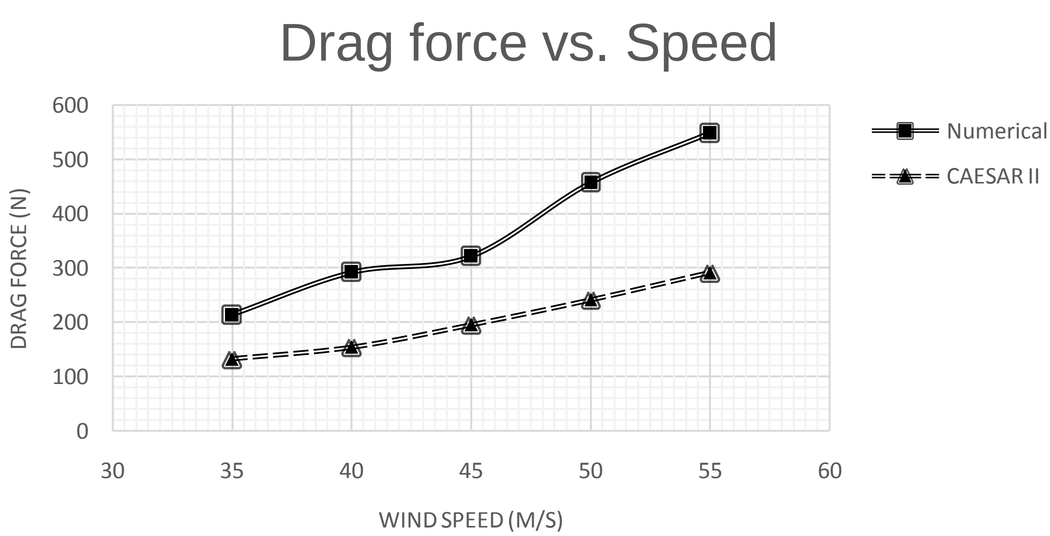


Figure 7: Drag force vs. speed

Conclusion

In this paper, effect of wind on drag force of pipe was studied numerically. Results are compared with ASCE#7 standard. It was shown that the drag force which is predicted by standard is about 1.5 times lower than real forces. Although most standards take a safety factor in to account to be on the safe side, here not only there is no safety factor, but also the values of drag forces is much lower than the real values. Therefore, it is maybe necessary for engineers to use CFD analysis besides standards to predict the forces with better accuracy and also lower risks. The importance of this issue is more critical in case of risers around towers with low allowable nozzle loads and moments and in high elevation which the pipe is more in exposure of the high speed winds.

ACKNOWLEDGEMENT
Special thanks to HEDCO Company which do not spare any financial and spiritual support to this paper.

Nomenclature

A	Cross sectional area	P	Pressure
C_D	Drag coefficient	ρ	Density
D	Diameter of pipe	$\bar{\bar{\tau}}$	Shear stress tensor
F_D	Drag force	$\vec{v}$	Velocity vector
H	Height of pipe		

REFERENCES

Amjadimanesh A and Ghalatian MA (2014). Numerical simulation of wind force on petrochemical industrial riser pipes. *Third Scientific Conference of Process Engineering*, Tehran.
Brower Matt (2006). *Drag Force on a Cylinder* (New York: White Publication).
Bruschi G, Tomoko N, Kevin T and Rick W (2003). A comparison of analytical methods drag coefficient of a cylinder. *Journal of Mechatronic and Electerical Engineering* **4**(2) 34-43.
Butt U and Egbers C (2013). Aerodynamic Characteristics of Flow over Circular cylinders with patterned surface. *International Journal of Materials, Mechanics and Manufacturing* (IACSIT Press) **1**(2) 121-125.
Cengel YA and Cimbala M (2006). *Fluid Mechanics Fundamentals and Applications,* first edition, SI Units (McGraw-Hill).
Kharaiti L and Abrol M (2005). ASCE 7 Minimum Design Loads for Buildings and Other Structure. Virginia: American Society of Civil Engineers.
Mallick M and Kumar A (2014). Study on Drag Coefficient for the Flow Past a Cylinder. *International Journal of Civil Engineering Research* (Research India Publications) **5** 301-306.
Sato M and Takaya K (2012). A fundamental study of the flow past a circular cylinder using Abaqus/CFD. *SIMULIA Community Conference.*
Versteeg HHK and Malalasekera W (2007). An introduction to computational fluid dynamics: The finite volume method. Pearson Education Australia.
White F (2010). *Fundamentals of Fluid Mechanics*, 7th edition (McGraw-Hill).