Advances in Engineering Research (AER), volume 105
3rd Annual International Conference on Mechanics and Mechanical Engineering (MME 2016)

The Research on Flow Past a Cylinder

Da-Wei SUN[1,a,*] and Bing-Xi SUN[1]
[1]College of Physics and Electronic Information,
Inner Mongolia University for the Nationalities, Tongliao,
Inner Mongolia, 028000, China
[a]weidasun@163.com
*Corresponding author

Keywords: Cylindrical disturbed flow, Reynolds number, Lift forces.

Abstract. The flow around a circular cylinder is one of the classical problems in CFD, in engineering will often encounter this kind of problem, such as river flows through Piled Wharves, piers, offshore drilling platform pillar, and all kinds of large-scale industrial equipment and facilities how better resistance to the interference of external factors such as. Thus, for the cylindrical fluid research and exploration has become the focus of attention of many scholars, because it is the understanding that other columnar flow around a bluff body based and in engineering technology in the most widely used, so for the study of the flow around a circular cylinder, with important significance. So the introduction of multi physics numerical simulation software COMSOL. In the simulation of incompressible fluid around the cylinder and the cylinder is placed in the center of the pipeline, the Reynolds number of approximately 100. The results show that produced a vortex, formation of Karman vortex street.

Introduction

Due to the instability of the flow in the flow of a bluff body, such as the car it is difficult to calculate. This body by the wake vortex of disorder, of various sizes, the human body produces a large resistance. In contrast, aircraft turbulence and streamlined body of the fish next to only produce a weak flow disturbances. When you a slender body in a slow flow, since the swirl organized, which exception occurs. Both sides von KáRMán vortex appears a predictable frequency and vortex shedding involves alternately. Everyday example of this phenomenon include: in the air stream of telephone lines and car radio antenna vibration. From an engineering point of view, it is important to predict the vibration frequency at different fluid velocities, so as to avoid vibration and vortex solid structures undesired resonances between off. To help reduce this effect, the plant's engineers put a spiral on the upper part of the high chimney; structure prohibited element changes the shape of the vortex from a different location for the constructive interference.

Model Definition

The conservation equations of mass and momentum are

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial U_i}{\partial t} + \rho \frac{\partial}{\partial x_j}(U_j U_i) = \rho \nu \frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{\partial P}{\partial x_i} \tag{2}$$

Where t is time; P is pressure; the kinematic viscosity is defined as $\nu = 10^{-3} m^2 / s$; the fluid density is $1.0 kg / m^3$.

The drag and lift forces are

$$F_D = \int_s (\rho v \frac{\partial v_t}{\partial n} n_y - P n_x) dS \tag{3}$$

$$F_L = -\int_s (\rho v \frac{\partial v_t}{\partial n} n_x - P n_y) dS \tag{4}$$

$$C_D = \frac{2F_D}{\rho L U_{mean}^2} \tag{5}$$

$$C_L = \frac{2F_L}{\rho L U_{mean}^2} \tag{6}$$

2D Cases

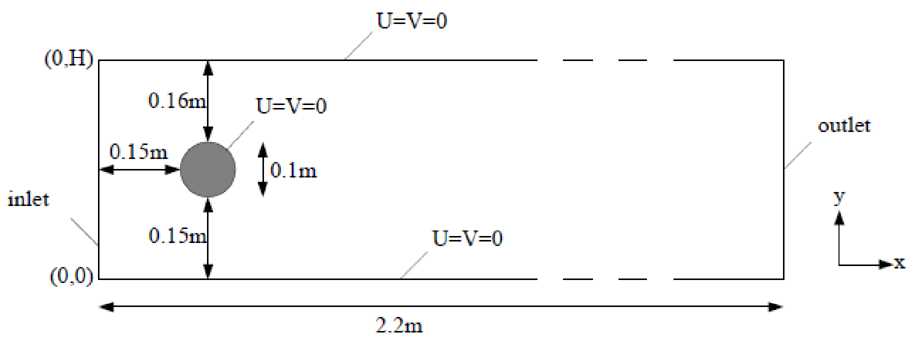

Figure 1. Test cases with boundary conditions

The inflow condition is

$$U(0,y,t) = 4U_m y(H-y)/H^2, V = 0$$

Where $H = 0.41m; U_m = 1.5m/s$; $\mathrm{Re} = 100$.

Conclusions

The multi physics field software COMSOL for single cylinder are numerically simulated, respectively of the flow around a circular cylinder of vorticity, contours of pressure; pressure coefficient, lift coefficient and time function diagram; velocity vector diagram, and through analysis to the following node on the results:(1) at Reynolds number Re = 100 to simulate incompressible fluid through a cylindrical, from found force exerted by the fluid on the cylinder is high frequency changes, and there are periodic, cylindrical upstream surface and dorsal stream surface pressure difference exists, with temporal fluctuations.(2) COMSOL software can be used to simulate such problems, reflecting the flow characteristics.

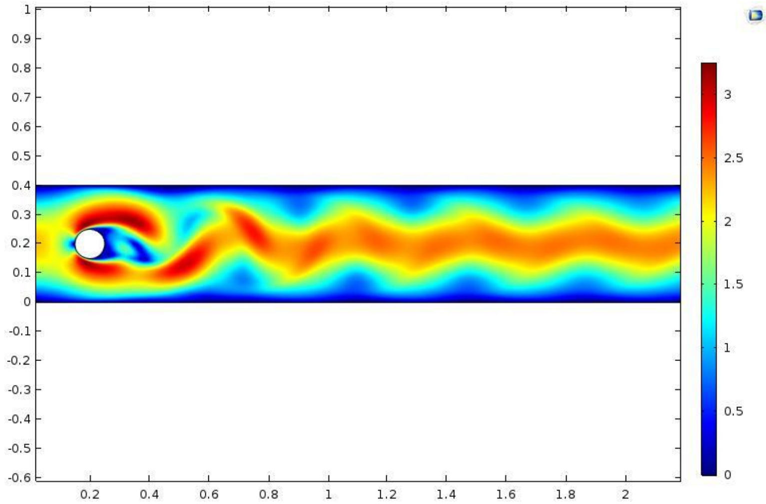

Figure 1. Time=7s Velocity magnitude (m/s)

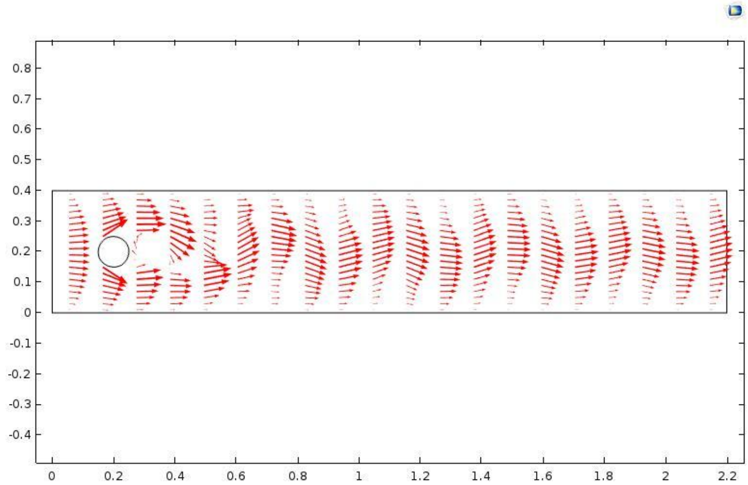

Figure 2. Time=7s Velocity field

Acknowledgements

This work was financially supported by the Doctoral Scientific Research Foundation of Inner Mongolia University for Nationality NO. BS325.

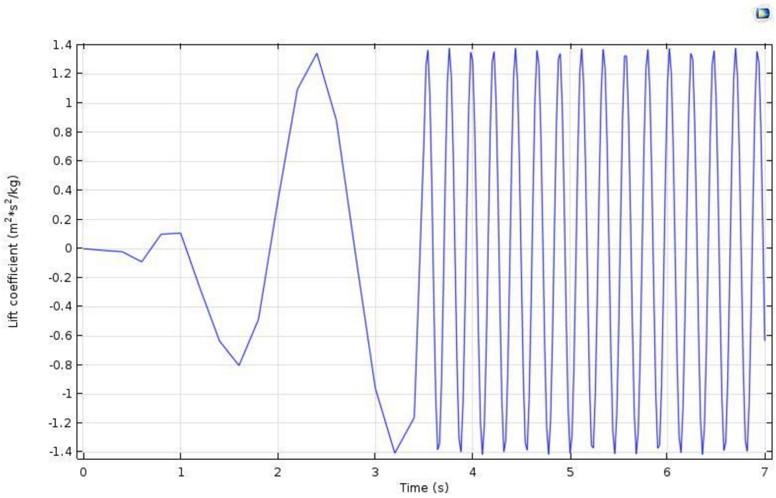

Figure 3. Lift coefficient

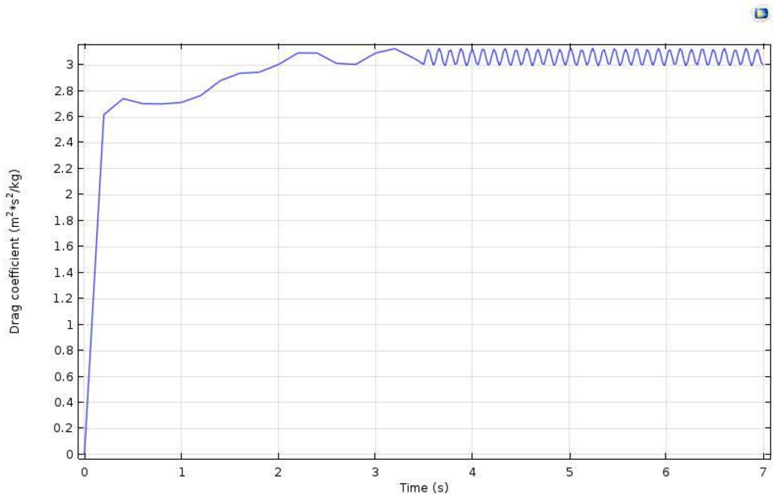

Figure 4. Drag coefficient

References

1. M. Schäfer and S. Turek, "Benchmark Computations of Laminar Flow Around Cylinder," E.H. Hirschel ed., Flow Simulation with High-Performance Computers II, Volume 52 of Notes on Numerical Fluid Mechanics, Vieweg, pp. 547–566, 1996.