The Analysis about flow Around a Single Circular Cylinder

Heng Cai

School of science, Southwest Petroleum University, Chengdu 610500, China

1405286409@qq.com

Abstract. The analysis for the flow around a fixed cylinder is very meaningful. Because this kind of research has extensive engineering application background, such as platform support in Marine engineering pile and the vibration of submarine pipeline under the action of wave, etc. Due to the limitation of theory and experiment conditions in earlier studies, the flow around a cylinder mainly focused on the determination of the force applied on the cylinder. With the rapid development of the computational fluid dynamics and modern experiment technology, numerical simulation and experimental measurement method is widely used in these aspects. This paper will use the CFD module from the ABAQUS6.12-3 for numerical simulation. The simulation regards the fixed single rigid cylinder in uniform flow under the conditions a certain Reynolds as investigation object. This paper will give the relevant parameter variation diagram. Meanwhile it will describe the corresponding physical phenomenon. Finally, the conclusion of the analysis will be given in this article.

Keywords: ABAQUS, cylinder, CFD.

1. Basic theory

According to the similarity theories, dimensionless physical parameters can be used as the basis of a physical phenomenon analogy between different systems. The major dimensionless parameters should be grasped before detailing the test methods and analysis model. These parameters are the basis for formulating and referring the model. These parameters not only reflect some universal regulation, but also to reflect the real situation between the physical model and production practices. Reynolds number is the most important and fundamental parameter in many dimensionless parameters which describes the fluid. It reflects the ratio between inertia force and viscous force:

$$\mathrm{Re} = \frac{\rho V L}{\mu}$$

2. Modeling parameters

2.1 Physical model

ABAQUS10.0 increased the CFD solver. Version 6.10 is limited to simulate the incompressible Newtonian fluid. The function is still relatively limited compared with the professional CFD software. This paper will use the ABAQUS6.12-3 version to make the numerical analysis for the flow field jacket. The paper regards the jacket flow field as a single rigid cylindrical flow problem. The problem is originally a two-dimensional plane problem. To simplify the problem as a layer of solid network model in that the ABAQUS CFD solver can only solve the three-dimensional problem. Simulation sets the thickness of the flow field to 0.001 m. The whole flow field is uniform laminar flow field .The Reynolds number Re=100. The simplified model size as shown in figure 2-1:

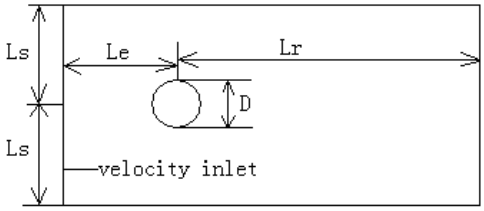

Figure.2-1 the simplified model size

Diagram: D = 0.1 m; Le = 4 d; Lr = 12 d; Ls = 4 d.

The entity model is established in ABAQUS software as shown in figure 2-2:

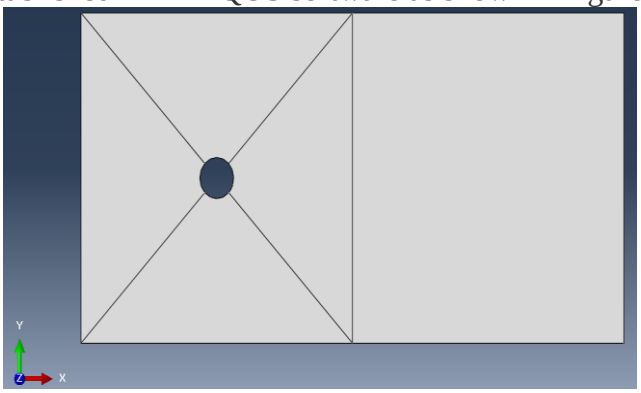

Figure.2-2 the entity model is established in ABAQUS software

2.2 Material properties and boundary conditions

Jacket sees as a cylindrical. The material is steel. The parameters are: density 7800Kg/m^3, the elastic modulus 2e11Pa, Poisson ratio of 0.3. The paper regards the flow field as laminar flow model in the simulation. There is big difference between physical parameters of the sea in different temperature, so this article uses the standard state parameters as the initial parameters. Its parameters are as follows: the density of 1000 Kg/m3, dynamic viscosity of 0.1 Pa. S. The boundary conditions are set as shown in figure 2-3:

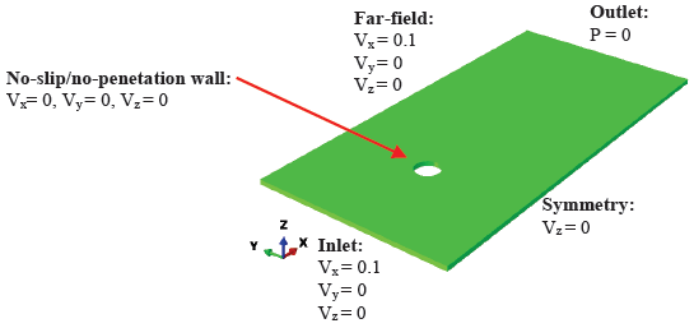

Figure.2-3 the boundary conditions of the Single cylinder flow

2.3 Mesh generation

Discretizing the area of the calculation should be made at first before using CFD method to calculate the flow field. The quality of the grid will directly affect the results of numerical calculation. It is necessary to predict the flow field on the basic physics before meshing. The grid can be relatively sparse for the far field or the area of uniform flow, it is necessary to arrange a relatively dense grid in the near field or under the condition of the complicated flow,

The division method of hexahedral mesh will be used in the paper. Mesh cylinder is relatively concentrated in the cylinder and wake, the rest of the grids gradual transition to the sparse. Meshing condition as shown in figure 2-4:

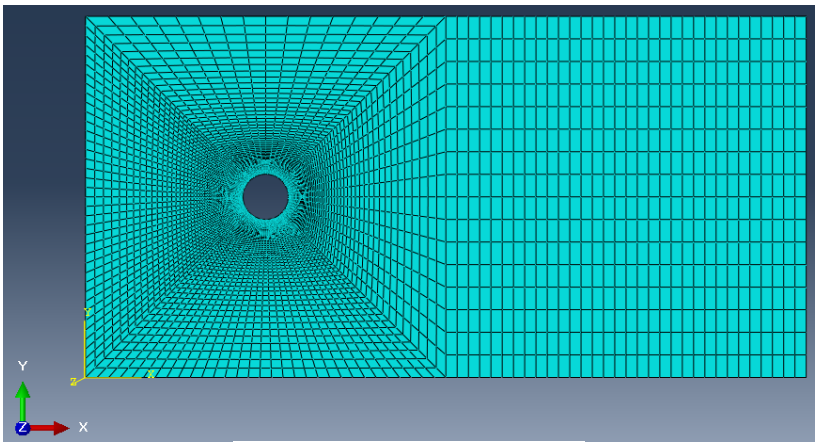

Figure.2-4 mesh generation

3. Analysis of the calculated results

The high single cylinder flow around the basic theory of fluid mechanics can be able to lead to conclusions: when the object continues in the state of uniform motion in a straight line, the object of resistance is zero. Meanwhile, fluid exist tangential velocity on the surface of the object .However, the offshore jacket platform is located in real fluid. At this time, the resistance of cylinder is not zero when the jacket is simplified as cylinder. At the same time, the velocity distribution, pressure distribution in the flow field are related to the Reynolds number. When the Reynolds number is close to 1, the speed of the whole flow field, pressure distribution is very close to the condition of ideal fluid. When the Reynolds number continues to increase, viscous fluid area will appear, the so-called boundary layer region. In addition, for single cylinder flow around the famous karman vortex street often happens. It is a strange flow field change rule.

According to the actual situation of this example, the corresponding flow field model was established. It is concluded that the velocity of flow field around a single cylinder distribution, respectively, as shown in figure 3-1 ~ 3-3:

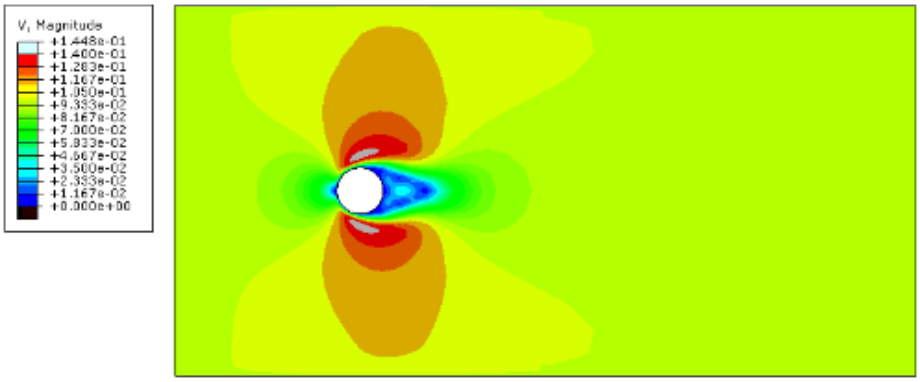

Figure. 3-1 single cylinder speed cloud - vortex shedding (t = 0 s)

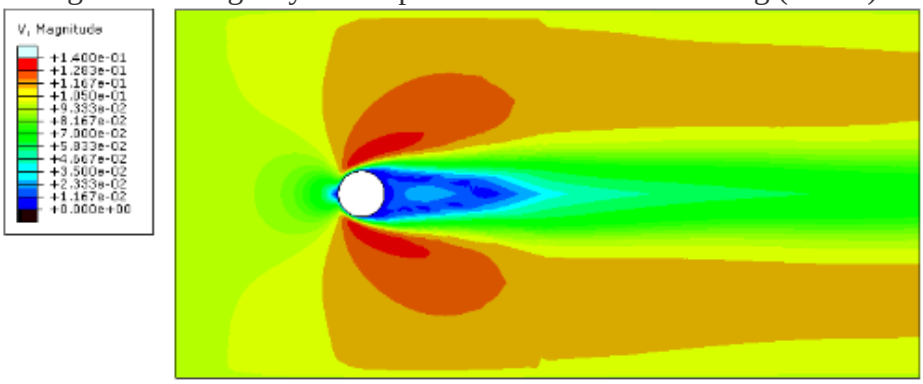

Figure. 3-2 single cylinder speed cloud - vortex shedding (t = 1s)

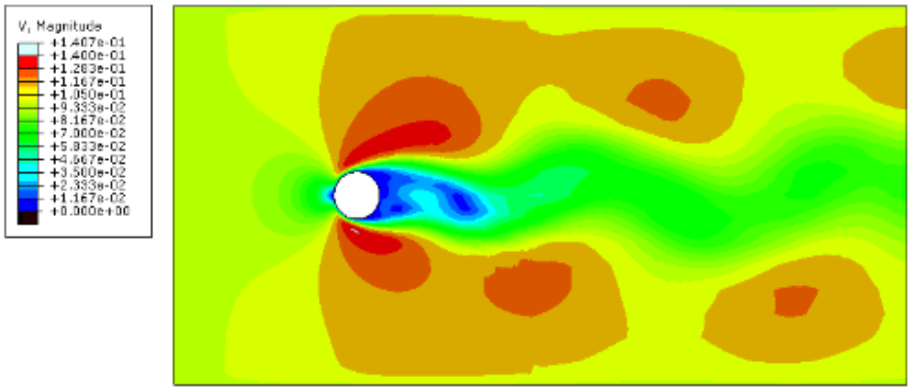

Figure. 3-3 single cylinder speed cloud - vortex shedding (t = 2s)

From figure 3-1 ~ 3-3, it can be seen that for the laminar flow field model of Reynolds number is 400,there is a big difference between the velocity distribution of flow field in each time, including the flow field of the boundary layer separation ,vortex shedding and karman vortex street effect. Overall, the change rule of the flow field around a single cylinder is relatively simple. Figure 3-4shows the flow field at a particular moment, vortex shedding flow chart.

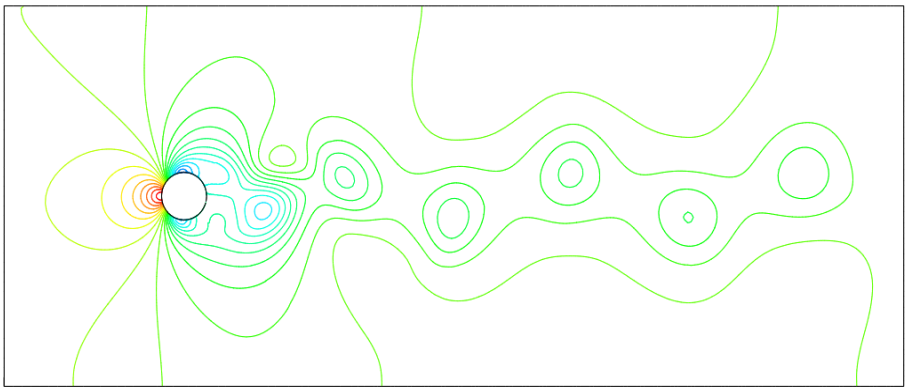

Figure. 3-4 vortex shedding flow chart

4. Conclusion

This paper obtain a satisfactory results by using hexahedral mesh division method. For a model of single flow around a cylinder, in certain circumstances Re number, alternating vortices will appear on the back of the cylinder, and gradually spread to the rear. At the same time, the vortex will impact cylinder lift and drag coefficient. Meanwhile, there has been a more significant effect called Karman Vortex Street, which also includes the boundary layer separation and vortex shedding, etc. It not only laid the foundation for subsequent correlation analysis, also demonstrates the ABAQUS CFD software can also make flow field numerical simulation.

Reference

[1] Y.Q.Huang, J Deng, A.L. Ren. Research on lift and drag in unsteady viscous flow around circular cylinders. Journal of Zhe Jiang University (Engineering Science), Vol.37 (2003) No.5, p.596-601.

[2] L.Q.Zhao, B.Chen. Two-dimensional FEM model of Vortex-Induced Vibration of a Circular Cylinder. Ocean Technology, Vol.25 (2006) No. 4, p.117-121.

[3] J.R.Meneghini, F.Saltara. Numerical simulation of flow interference between two circular cylinders in tandem and side-by-side arrangements. Journal of Fluids and Structures, 2001, 15:327-350.

[4] G.X.Wu, Z.Z.Hu. Numerical simulation of viscous flow around unrestrained cylinders. Journal of Fluids and Structures, 2006, 22:371-390.

[5] R.M.C.SO, X.Q.Wang. Vortex-induced vibrations of two side-by-side Euler-Bernoulli beams. Journal of Sound and Vibritions, 2003, 259(3):677-700.