Numerical Simulation of Flow Around a Cylinder based on Fluent

Donghai Chen¹a*, Ying Pan¹ and Yibo Liu¹b
¹Changjiang institute of survey, planning, design and research, Wuhan, Hubei 430081, China
²Zhixing College of Hubei University, Wuhan, Hubei 430081, China
³Key Laboratory of Metallurgical Equipment and Control Technology of Ministry of Education, Wuhan University of Science and Technology, Wuhan, Hubei 430081, China

*chendonghai@cjwsjy.com.cn; b840877291@qq.com

Abstract. In this paper, the fluid flow around a cylinder is taken as an example to study the different characteristics of fluid flow. Gambit is used to build a 2D model of the watershed and its cylinders, and divide the mesh and define the boundaries. The flow model was constructed with Fluent, the steady-state flow in this flow state is simulated and calculated. And draw the velocity vector of the steady flow of the fluid, use the velocity vector to clearly show the double vortex behind the cylinder. Finally, through Carmen vortex street flow simulation and other parameters, the single vortex frequency is calculated and compared with the single vortex frequency calculated by Fluent to verify the correctness of Fluent's calculation.

Keywords: Fluid Flow, 2D Model, Cylinders

1. Introduction
In daily life, cylindrical shapes are very common, such as supporting piles for platforms, water pipes, and wooden beams of houses [1]. The flow around a cylinder has been the subject of many theoretical analyses, experimental studies and numerical simulations. Research on the flow around a cylinder is of great significance in engineering practice, such as wind resistance of cable stayed cables, wind resistance research of high-rise buildings, heat exchange of offshore platforms and tube bundles, etc. [2]. When we want to carry out numerical simulation analysis of flow around a cylinder, we need to understand two related background knowledge.

1.1. Split Stream
Separated flows exist widely in the flow of various objects: such as aviation aircraft, buildings, fans, etc. Due to viscous forces, the surface of all solid objects passing through the fluid will form a boundary layer around them. The boundary layer can be layered or turbulent. A reasonable assessment
of whether the boundary layer is laminar or turbulent can be made by calculating the Reynolds number of local flow conditions. When the boundary layer travels far enough relative to the unfavorable pressure gradient, the speed of the boundary layer relative to the object drops to almost zero, at which time flow separation occurs, a separated flow is generated, the fluid fluid is separated from the surface of the object, and the vortex is taken form.

1.2. Fluid Separation and Carmen Vortex
For solids passing through a fluid, the fluid flow and vortex formation conditions are related to the Reynolds number. According to the von Carmen vortex street theory [3], for a fluid flowing through a cylinder, the frequency \( f \) of each single vortex of the vortex street is directly proportional to the velocity of the flow around \( v \) and inversely proportional to the diameter \( d \) of the cylinder, that is, \( f = Sr (v/d) \). Sr is Strouhal number, which is mainly related to Reynolds number. When the Reynolds number is 40 to 150, Sr is approximately 0.2; when the Reynolds number is greater than 3.5x106, Sr is about 0.27 [4].

2. Building the Model

2.1. Flow Field Size Setting, Reynolds Number Calculation, Incoming Flow Speed Selection

2.1.1. Flow Field Size Selection. The flow field in this paper is rectangular. The diameter of the cylinder is \( d = L = 1m \). The height of the flow field is 10m and the length is 15m. The center of the cylinder is 5m from the left of the rectangle and 5m from the lower edge of the rectangle.

Where the Reynolds number formula is

\[
Re = \frac{\rho U_e L}{\mu}
\]

Where Re represents the Reynolds number, \( \rho \) free flow density (Kg/m3), \( U_e \) free flow velocity (m/s), \( L \) characteristic length (m), and \( \mu \) flow force viscosity (Pa s or N s/m²).

2.1.2. Analysis of Reynolds Parameters Around Cylindrical Flow. The fluid flow field analysis was performed using Fluent software. The commonly used fluid media parameters are shown in Table 1. In this paper, the fluid medium is selected as air, and the characteristic length \( L \) is taken as the diameter \( d \) of the cylinder.

| Table 1. Fluent common fluid density and dynamic viscosity parameters |
|------------------|------------------|
| Density \( \rho \) (Kg/m³) | Dynamic viscosity \( \mu \) (Pa s) |
| Air              | 1.23            | 1.79E-5          |
| Water            | 998.2           | 1.003E-5         |

According to the applicable requirements of this article, the range of Reynolds number of the flow field to be numerically simulated is divided into two types: Re values are 5 ~ 40 and 40 ~ 150 respectively. Let Re = 30, bring the parameters of air \( \rho = 1.23 \text{ Kg/m}^3 \), \( \mu = 1.79E-5 \) into formula (2-1), and get

\[
V = 0.000436 \text{m/s}
\]

Similarly, let Re = 300, get fluid velocity \( V = 0.00435 \).
To sum up, the size of the flow field and the fluid velocity under different Reynolds numbers are shown in Table 2:

| Reynolds number | 30   | 300  |
|-----------------|------|------|
| Flow field size (m) | 15*10 | 15*10 |
| Cylinder diameter (m)  | 1    | 1    |
| Incoming speed (m/s)   | 0.000436 | 0.00435 |

Calculate time step:

\[ \Delta t = \frac{\pi D}{100} = \frac{3.14 \times 1}{100} = 0.0314 \]

(3)

2.2. GAMBIT Flow Field Modeling and Meshing

2.2.1. Create Flow Field Domain. According to the size selected earlier, create a cylindrical surface face1 (circular, radius 0.5m), a rectangular flow scene face2 (rectangular, 15m * 10m), the center of the circle is 5m from the left of the rectangle, and 5m from the bottom of the rectangle. Through GAMBIT Boolean operation function, face1 is subtracted from face2, and the remaining face2 is the required flow field;

2.2.2. Meshing and Boundary Settings. The shape of the face mesh is Tri [5]. The mesh result is shown in Figure 1

![Figure 1 Mesh division and flow field setting](image)

2.2.3 Stable Twin Vortex Flow Simulation. Import the msh file completed in GAMBIT into fluent, and set the parameters to simulate the stable double vortex flow.

After extracting the calculated results, the steady-state flow velocity vector diagram is shown in Figure 2.
3. Carmen Vortex Flow Simulation

3.1 Simulation of Steady-State Flow Velocity
Import the msh file completed in GAMBIT into fluent, and set the parameters to simulate the stable double vortex flow.

After the parameters are set, the steady-state flow velocity cloud diagram can be extracted after calculation, as shown in Figure 3.

3.2 Transient Flow Simulation, View Vortex Street
Import the msh file completed in GAMBIT into fluent, and set the parameters to simulate the stable double vortex flow.

The steps for setting transient simulation parameters in this article are as follows:
1) General: Select the grid unit as mm, select transient in the time column of the solver, and then check to check the quality of the grid;
2) Model: Select laminar flow Viscous-laminar;
3) Material: Select fluid as Air;
4) Boundary conditions: InletSet the boundary condition velocity-inlet, the value corresponds to the flow velocity 0.00435, outf is set to outflow, pout is set to pressure-outlet, and the value is 0;
5) Solution initialization: Select the speed entry to initialize and set the speed to 0.00435;
6) Run calculation;
7) Calculation and extraction results: transient flow velocity cloud diagram, as shown in Figure 4.
3.3 Determine single vortex motion frequency
Monitor the velocity vector of the transient point and determine the frequency of the single vortex motion by FFT.

Get the following interface through the fft function in the plot, and save the FFT file file-name.

Export the frequency curve data, and finally get the frequency value, f=0.000999.
Validation: From the formula $f = Sr(U_\infty / d)$, the Reynolds number is at $Re = 300$, and the approximate parameter value of $Sr$ is 0.21, then $V = 0.00435 m / s$, $d = L = 1 m$, Get $f = 0.0009135 Hz$.

4. Conclusion
At this article, and wherein the cylindrical basin with Gambit 2D model, after meshing, the boundaries defined by Fluent construct flow model, flow simulation in the steady state flow. Draw the velocity vector of the steady flow of the fluid, and use the velocity vector to clearly show the double vortex behind the cylinder. Finally, the calculation results show that Fluent software is effective for the numerical simulation of the flow field around a multi-cylinder.

References
[1] Hu Weihua. Numerical simulation of flow around a multi-cylinder based on Fluent [J]. Science & Technology Review, 2010 (24): 77-80.
[2] Sun Qingli, Huang Peng. Calculation and Analysis of Flow Around a Cylinder Based on Fluent [J]. Anhui Architecture, 2014 (04): 70-71.
[3] Wang Zhendong. Von Carmen and Carmen Vortex [J]. Chinese Journal of Nature, 2010, 32 (4): 243-245.
[4] Zhan Hao, Li Wanping, Fang Qinhan, et al. Simulation calculation of flow around a cylinder under different Reynolds numbers [J]. Journal of Wuhan University of Technology, v.30; No.191 (12): 129-132.

[5] YUAN Xin, XU Lijun, YE Zhiquan, et al. Numerical simulation of separation flow of large axis of attack for horizontal axis wind turbine airfoil [J]. Journal of Solar Energy, 1997 (1): 35-40.