Numerical simulation of the Magnus effect and its application

Liwei Song*
School of science, University of Science and Technology Liaoning, Anshan, China

*Corresponding author e-mail: ur135@qq.com

Abstract. Numerical simulations based on computational fluid dynamics are used to study the Magnus effect. The cross force can be obtained through solving the governing equations of the flow around rotating cylinder. Through analyzing the characteristics and structure of the flow field, the Magnus effect can be enhanced by using endplates to change the flow pattern. Numerical results prove the effectiveness and practicality of the improvement.

1. Introduction
Together with theoretical analysis and scientific experiment, scientific computing has been regarded as one of the major methods in modern scientific researches. For the fluid mechanical problems, due to the nonlinear characteristics of the flow equations, difficulties exist in theoretical analysis. Through scientific computing, the flow field information and fluid dynamic parameters can be obtained by solving numerically the flow governing equations.

For a rotating object, when the direction of the rotating angular velocity is not the same as that of the translational velocity, the cross force is generated due to the boundary layer displacement thickness distortion on both sides of the rotating body. The Magnus effect has been used in airborne wind energy generation systems, flying machines, ship propulsion and stabilization, which utilize the Magnus effect to create driving force with a rotating cylinder. The flow around a rotating cylinder [1-4] is the basic problem of the Magnus effect.

2. Numerical Simulation
2.1. Mathematical Model
Fluid dynamics are based on the three fundamental physical principles: Mass is conserved, Newton's second law and Energy is conserved. These physical principles are applied to a model of the flow. The Navier-Stokes equations are the governing equations of fluid flow, which include mathematical statements of the particular physical principles, namely the continuity, momentum, and energy equations. For incompressible flow, ignoring the influence of the temperature and body forces, the Navier-Stokes equations [5] are

\[ \nabla \cdot [ \rho \mathbf{v} ] = 0 \] (1)

\[ \frac{\partial}{\partial t} ( \rho \mathbf{v} ) + \nabla \cdot ( \rho \mathbf{v} \mathbf{v} ) = - \nabla p + \nabla \cdot \mathbf{T} \] (2)

Where \( t \) is the time, \( \mathbf{v} \) is the velocity vector, \( \rho \) is the density, \( p \) is the pressure, \( \mathbf{T} \) is the viscous stress tensor, \( \nabla = \partial_x + \partial_y + \partial_z \) is a vector operator.
2.2. Turbulence equations
The Reynolds-averaged Navier-Stokes (RANS) equations are time-averaged equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. To obtain the Reynolds-Averaged Navier-Stokes equations, each solution variable $a$ is decomposed into its mean value $\bar{a}$ and its fluctuating component $a'$, these equations can be written as

$$\nabla \cdot \left[ \rho \bar{\mathbf{v}} \right] = 0$$

$$\partial_t (\rho \bar{\mathbf{v}}) + \nabla \cdot (\rho \bar{\mathbf{v}} \mathbf{v}) = -\nabla p + \nabla \cdot (\bar{\mathbf{u}} + \mathbf{\tau}_{RANS})$$

where $\mathbf{\tau}_{RANS}$ is the Reynolds stress tensor. The Boussinesq eddy-viscosity approximation is adopted to calculate $\mathbf{\tau}_{RANS}$ using the $k$-$\varepsilon$ two-equation turbulence model. The turbulent eddy viscosity $\mu_T$ is calculated as [6].

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon}$$

where $C_\mu$ is a model coefficient, $k$ is the turbulent kinetic energy, $\varepsilon$ is the turbulent dissipation rate.

2.3. Solving Methods
In the numerical simulation of flow problem, geometry modelling and mesh generation are pre-processing, and then the governing equations and turbulence model equations are discreted on the grid cell $V$ using finite volume method. The above equation becomes

$$\frac{d}{dt} \int_V \rho \phi dV + \int_{\partial V} \rho \mathbf{v} \cdot \mathbf{n} dS = \int_V \mathbf{F} \cdot dS + \int_V Q^d dV$$

where $\phi$ represents the transport of a scalar property. Equation(6) has four distinct term: the transient term, the convective flux, the diffusive flux and the source term. The linear algebraic equations can be obtained by discretig the integral equation [5].

3. Results
3.1. Flow around rotating cylinder in 2D
Regardless of the spanwise flow, a three-dimensional problem is reduced to a two-dimensional one and thus the calculation is much simplified. The cylinder diameter $D = 0.12$ m, the rotating angular velocity $\omega = 30$ rps, the center of the cylinder remains stationary, the inflow condition on the left side is $v_\infty = 7$ ms$^{-1}$, the Reynolds number $Re = \rho v_\infty D / \mu = 5.78 \times 10^4$.

Figure 1. (a) shows the velocity field contours and streamlines, where $\theta$ is the central angle. More streamlines in the front of the cylinder rounded the cylinder from the top due to the rotation of the cylinder. The flow velocity increases on the top of the cylinder because the streamlines are much more concentrated. The pressure field contours are shown in Figure 1. (b), an obvious negative pressure zone arose in the top of the cylinder. Pressure difference between the top and the bottom of the cylinder is the main cause of the cross force.

![Figure 1](image_url)

**Figure 1.** Flow past cylinder 2D: (a) velocity contour and streamline, (b) pressure contour.
3.2. Flow around rotating cylinder

The flow around a rotating circular cylinder in three-dimension is numerical simulated. The cylinder diameter \( D = 0.12 \) m, the length of the cylinder \( l = 0.2 \) m, the center of the cylinder located at the origin, the cylindrical symmetry axis coincides with the z-axis. The flow field is calculated for different inflow velocity and rotating angular velocity. Table 1. shows the cross force coefficients for different computational conditions. The cross force coefficient increased with the rotating angular velocity and decreased with the inflow velocity.

| inflow velocity /ms\(^{-1}\) | rotating angular velocity /rps |
|-------------------------------|-------------------------------|
| 20                            | 30                            | 40                            |
| 5                             | 1.03                          | 1.25                          | 1.3                          |
| 7                             | 0.91                          | 1.02                          | 1.18                         |
| 9                             | 0.86                          | 0.92                          | 1.02                         |

Figure 2. (a) shows the streamlines and the pressure field contours of the flow field on the condition that \( v_\infty = 7 \) ms\(^{-1}\) and \( \omega = 30 \) rps. The streamlines in the front of the cylinder rounded to the end of the cylinder, which is cause by the pressure difference between the front and the end of the cylinder. The Magnus effect is weakened near the ends of the cylinder.

Figure 2. (b) shows the pressure field contours, where the maximum of pressure located at 183° and the minimum located at 88°. The pressure difference between the top and the bottom are weakened near the ends of the cylinder, which cause the decreasing of the cross force.

4. Numerical simulation applications

The Magnus effect can be increased through increasing the length of cylinder or rotating angular velocity, but these are not efficient in term of structure and energy efficiency. Based on the results in 2.3, the spanwise flow, weakening the Magnus effect, can be reduced by adding endplates at the ends of the cylinder.

The height of the endplate \( h = 0.01 \) m, the inflow velocity \( v_\infty = 7 \) ms\(^{-1}\) and rotating angular velocity \( \omega = 30 \) rps. The streamlines are shown in Figure 3. (a). It can be seen that the spanwise flow is restrained effectively near the ends of the cylinder by the endplates. More streamlines in the front of the cylinder rounded the cylinder from the top.

Figure 3. (b) shows the pressure field contours. The maximum of pressure located at 189° and the minimum located at 74°. This illustrates more fluid flow past the cylinder form the top. The cross force coefficient is increased nearly by 1.8 times further, from 1.02 to 2.87. Appending endplates equates
increasing the effective length of the cylinder, improving the Magnus effect and enhancing the cross force of the rotating cylinder.

\[\text{(a) } \quad \text{(b) } \]

**Figure 3.** Flow past cylinder appending endplates: (a) streamlines, (b) pressure contours.

5. Conclusion

The Magnus effect was studied by numerical simulation. Through the flow field simulation of the rotating cylinder, found the flow structural characteristics and the influence factor of the cross force. Based on the analysis of the flow field, the Magnus effect can be improved by appending endplates. It is proved through numerical simulation that appending endplates is effective.

References

[1] S Mittal, B Kumar. Flow past a rotating cylinder, Journal of Fluid Mechanics. 476 (2003) 303-334.
[2] I K M Lam. Vortex shedding flow behind a slowly rotating circular cylinder, Journal of Fluids & Structures. 25 (2009) 245-262.
[3] Jan O Pralits, Luca Brandt, Flavio Giannetti. Instability and sensitivity of the flow around a rotating circular cylinder, Journal of Fluid Mechanics. 650 (2010) 513-536.
[4] Jan O Pralits, Flavio Giannetti, Luca Brandt. Three-dimensional instability of the flow around a rotating circular cylinder, Journal of Fluid Mechanics. 730 (2013) 5-18.
[5] Moukalled F., Mangani L., Darwish M. The finite volume method in computational fluid dynamics an advanced introduction with OpenFOAM and Matlab, Springer, 2016, pp. 49, 57, 85.
[6] David C. Wilcox. Turbulence modeling for CFD, third ed., DCW Industries, 2006, pp. 128-130.