Influence of free surface and bottom on flow around a cylinder

Siqin Chen*, Frank Lim
China University of Petroleum, Beijing, China

*Corresponding author e-mail: 2018210902@student.cup.edu.cn

Abstract. Doing research on vortex-induced vibration (VIV), not only should we do some numerical simulation but also need to experiment. The existence of the free surface and bottom will influence the drag. We analyzed the drag under different velocity of water under different flow fields, then compared the results. The results show that the water depth will influence vortex shedding thus increase the drag. With a decrease of velocity, the effect of drag will be reduced. Nevertheless, it’s not always decreasing, when velocity lower than 0.8m/s, the effect would be steady.

1. Introduction
As a collective phenomenon in ocean engineering, vortex-induced vibration (VIV) is always regarded as an important research content. When the vortex-shedding frequency is close or equal to the natural frequency of the body, continuous periodic vibration of the structure could make it susceptible to fatigue failure, hence offshore structural members must be intended to prevent VIV [1-3]. After designed for those structures, some experiments should be made to check the effects of those structures on prevention of VIV. But usually, the experimental conditions are limited by so many things. What we want is to eliminate those limits and found the results without restriction.

By using software to do some simulation, can get the drag coefficients under different conditions. Compare the results and found the connection between them is exactly what we need to do.

In this paper, VOF model and the k-ε model are selected for multiphase model and viscous model respectively. The flow around a cylinder in finite and infinite flow fields is numerically studied. The influence of the free surface and bottom surface on the drag coefficient is discussed.

2. Model

2.1. The physical model
Fig.1 shows the situation in finite flow fields. The size of the flow field domain is 15m x 4m, with 2m of water depth and the rest part is air.
Fig. 2 shows the situation in the control group, 4m water depth is chose, and other conditions are same.

Every parameter is shown in the two figures, to avoid the water inrush effect at the inlet of the flow field, the distance between inlet and center of the circular cylinder is more than 10D. According to previous research [4,5], when the distance between free surface and center of the circular cylinder is more than 4D, the existence of the free surface will not affect the flow around the circular cylinder. So, 4m water depth is chosen to be the control group. The significance to do that is to ensure other conditions will not affect the results.

2.2. The simulation models
To be comparable, every condition is the same expect water depth. The whole computational domain adopts a structured grid, and the local grid around the cylinder is encrypted. This kind of method to draw the mesh is used by two different working conditions.

Fig.3~4 shows the method to draw the mesh with different flow fields.
The inlet of the flow field is defined as uniform incoming flow with a velocity of $U$. The outlet is deemed to be at infinity and is set to be a pressure outlet. The air had no velocity. The bottom is fixed as the wall. There was no relative slip between fluid and circular cylinder. Multiphase model chose VOF model and viscous model chose $k$-$\varepsilon$ model.

3. Results and discussion
Fig.5~6 shows the velocity diagram with different water depth and same velocity.

Figure 3. Mesh of water depth 2m

Figure 4. Mesh of water depth 4m

Figure 5. Velocity diagram of water depth 2m.
Figure 6. Velocity diagram of water depth 4m

As can be seen, with the same velocity, vortex formation in 2m depth is different from that in 4m depth. Overall, when the water depth is 2m, the vortex-shedding is more violent because the exist of free surface and bottom will facilitate vortex shedding and some vortex will shed before they grow up. Because of the influence on vortex shedding, the exist of free surface and bottom will influence the drag and drag coefficient.

Results of drag coefficient with 2m water depth have been shown in Table 1

| Velocity m/s | 0.5 | 0.7 | 0.8 | 0.9 | 1.2 | 1.4 | 1.6 |
|--------------|-----|-----|-----|-----|-----|-----|-----|
| Cd           | 0.61| 0.61| 0.61| 0.63| 0.65| 0.68| 0.72|

According to the results in the table. When the velocity is higher than 0.8m/s, the higher the velocity is, the bigger the drag coefficient is. But when the velocity is lower than or equal to 0.8m/s, the change of velocity will not affect the drag coefficient.

The results of 4m water depth show in Table 2.

| Velocity m/s | 0.5 | 0.7 | 0.8 | 0.9 | 1.2 | 1.4 | 1.6 |
|--------------|-----|-----|-----|-----|-----|-----|-----|
| Cd           | 0.50| 0.50| 0.50| 0.50| 0.50| 0.50| 0.50|

Be different with the situation in 2m water depth, the drag coefficient will not affect by velocity. So, in infinite flow fields, the drag coefficient will always be the same when the velocity changes.

After got the drag coefficient under two distinct situations, use the drag coefficient in 2m water depth to divide the counterparts in 4m water depth to get the correction coefficient.

| Velocity m/s | 0.5 | 0.7 | 0.8 | 0.9 | 1.2 | 1.4 | 1.6 |
|--------------|-----|-----|-----|-----|-----|-----|-----|
| Correction coefficient | 1.22| 1.22| 1.22| 1.25| 1.31| 1.37| 1.45|
The correction coefficient with different velocity has been demonstrated in Table 3. By using this correction coefficient, a connection between finite flow fields and infinite flow fields has been found. That means after experiment, this coefficient can be used to correct the experimental results in order to eliminate the impact of site restrictions.

In order to expand the application scope of this result, an empirical formula is needed to apply in more situations. The velocity is the independent variable and the correction factor is the dependent variable, get an empirical formula by fitting.

![Figure 7](image1.png)

**Figure 7.** Relationship between velocity and correction coefficient ($v \leq 0.8\text{m/s}$)

![Figure 8](image2.png)

**Figure 8.** Relationship between velocity and correction coefficient ($v > 0.8\text{m/s}$)

Fig. 7 and Fig. 8 show the relationship between velocity and correction coefficient when the velocity is lower or equal to 0.8m/s and higher than 0.8m/s respectively. Those graphs directly show when the velocity is lower or equal to 0.8m/s, the correction coefficient will always be same. However, when the velocity is bigger than 0.8m/s, the change of velocity will affect the correction coefficient. As can be observed in Fig.8, with the increase of velocity, the correction coefficient will increase corresponding.

Using MATLAB to get the empirical formula.

\[
\text{Correction Coefficient} = \begin{cases} 
1.22 & v \leq 0.8\text{m/s} \\
0.2173x^2 - 0.2611x + 1.311 & v > 0.8\text{m/s}
\end{cases}
\]  \hspace{1cm} (1)
Eq. 1 show the empirical formula we got. This formula shows the connection between 2m water depth flow fields and infinite flow fields. Bring different velocity into this formula, can get corresponding correction coefficient. So, when experimental results need to be revised, using velocity under corresponding working conditions to get the correction coefficient, so that we can get the results in the infinite flow field.

Although this formula can only be used when water depth is 2m, but this method to connect the finite flow field and infinite flow field can always be effective.

4. Conclusion
In this paper, the main target is to find the connection between finite flow field and infinite flow field, and uses this relationship to eliminate the impact of site restrictions on experimental results. The numerical simulation method is adopted to study the influence of the existence of the free surface and the bottom. By comparing with the simulation results in the infinite flow field, the influence of different velocities on the drag coefficient when the free surface and the bottom are existing is discussed. Draw the following conclusions:

(1) The existence of the free surface and the bottom will affect the vortex shedding, that’s why the results in this situation are different from the results in the infinite flow field.
(2) In a finite flow field, the velocity’s change will affect the drag coefficient. In 2m water depth flow field, when the velocity is higher than 0.8m/s, with the increasing of velocity, the drag coefficient increase. However, when the velocity is lower than or equal to 0.8m/s, the change of velocity will not affect the drag coefficient.
(3) In an infinite flow field, things are different. The drag coefficient will not change when the velocity changes. It is constant at any velocity.
(4) A correction coefficient can be used to connect two different situations, and an empirical formula between velocity and correction coefficient can be found by fitting. After getting this formula, could get different correction coefficient under different velocity and then use this coefficient to correct the experiment results.
(5) Although this formula can only be used when the water depth is 2m, but the way to connect those two situations can be referenced.

Acknowledgments
Thanks to Frank Lim for his guidance on this research.

References
[1] Schewe G. On the force fluctuations acting on a circular cylinder in cross flow from subcritical up to transcritical Reynolds numbers. J Fluid Mech 1983;133. 265 – 85.
[2] Bearman, P. W. On vortex shedding from a circular cylinder in the critical Reynolds number regime. Journal of Fluid Mechanics, 1969, 37 (3), 577 - 585.
[3] Breuer, Michael. A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow. International Journal of Heat and Fluid Flow, 2000, 21.5: 648 - 654.
[4] YU Pangdeng (2012). Numerical Simulation of High Reynolds Number Flow around a Circular Cylinder near the Wall, OFFSHORE OIL, Vol 32, No2, pp 102 - 105.
[5] Zhang Li (2016). Research on the Effect of the Height of Free Surface on the Flow Around a Circular Cylinder, JOURNAL OF ENGINEERING THERMO PHYSICS, Vol 37, No 2, pp 328 - 331.