Effects of Turbulence Model and Numerical Time Steps on Von Karman Flow Behavior and Drag Accuracy of Circular Cylinder

E Amalia1) *, M A Moelyadi1), M Ihsan1)
1) Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung, Indonesia

*ema@ftmd.itb.ac.id

Abstract. The flow of air passing around a circular cylinder on the Reynolds number of 250,000 is to show Von Karman Vortex Street Phenomenon. This phenomenon was captured well by using a right turbulence model. In this study, some turbulence models available in software ANSYS Fluent 16.0 was tested to simulate Von Karman vortex street phenomenon, namely k-epsilon, SST k-omega and Reynolds Stress, Detached Eddy Simulation (DES), and Large Eddy Simulation (LES). In addition, it was examined the effect of time step size on the accuracy of CFD simulation. The simulations are carried out by using two-dimensional and three-dimensional models and then compared with experimental data. For two-dimensional model, Von Karman Vortex Street phenomenon was captured successfully by using the SST k-omega turbulence model. As for the three-dimensional model, Von Karman Vortex Street phenomenon was captured by using Reynolds Stress Turbulence Model. The time step size value affects the smoothness quality of curves of drag coefficient over time, as well as affecting the running time of the simulation. The smaller time step size, the better inherent drag coefficient curves produced. Smaller time step size also gives faster computation time.

1. Introduction

Flow around a circular cylinder is one of basic engineering problems and is extensively studied with the applications including heat exchanger, chimney, and offshore supporting feet [1]. From scientific perspective, flow around a circular cylinder is having some important phenomenon of fluid dynamics, such as transition to turbulent, separation, and vortex shedding. Vortex shedding gives important contributions to practical things such as lift and drag coefficients of the circular cylinder. Von Karman vortex street is one example of vortex shedding.

Moreover, the flow around a tube bundle is challenging from Computational Fluid Dynamics (CFD) point of view [2]. Its applications in nuclear power plant cooling system and heat exchanger makes study of flow around circular cylinder to be important to carry out. Both flows around a single singular cylinder and around a bundle of it are challenging to analyzed by CFD. These flows contain laminar and turbulent condition of flow. Analyzing such kind of flows by using CFD demands a right choice of technique and model of flow.

In this study, analyses of flow around a circular cylinder are carried out by using ANSYS Fluent 16.0, a software of CFD. The objectives of this study are: (1) to know which turbulence model that gives the best solution compared with experimental result; (2) to determine the time step size to
provide efficient and accurate computational results. Time step size is also varied to know its effect to computational result. In addition, some conditions of flow are analyzed.

2. Method
Methodologies employed in this study are literature study and numerical study. The steps are as shown in figure 1. Step 1 is literature study of experimental data, and step 2 until 5 are numerical study.

![Diagram of Methodology]

**Figure 1.** Methodology of this study

Purpose of Step 1 in figure 1 of determining Reynolds number is to assure validation of numerical calculation result. For flow around a circular cylinder, there are some ranges of Reynolds number with different experimental phenomenon as shown in figure 2. In this study, phenomenon chosen to be simulated is turbulent Von Karman Vortex Street, which happened in Range of Reynolds number between 300 and 300,000 or number 4 in Figure 2. Reynolds number chosen in this study is 250,000. Figure 3 shows turbulent Von Karman Vortex Street happened in cloud. This phenomenon is challenging in knowing which turbulence model in ANSYS Fluent 16.0 that best capturing the phenomenon of turbulent Von Karman Vortex Street. Also, it is interesting to know the effect of time step size variation to curve of lift coefficient of the circular cylinder in this phenomenon.

The step 2 (see figure 1) of this study is construction of test specimen of the circular cylinder by using software called ANSYS ICEM. There are two types of test specimen, which are two-dimensional and three-dimensional circular cylinder. The reason of why using two types of specimen is to know whether the same turbulence model that best in capturing the turbulent Von Karman Vortex Street in two-dimensional and three-dimensional condition of CFD model. Moreover, as shown in figure 4, one of turbulence model is LES (Large Eddy Simulation). This type of turbulence model is only usable in the condition of three-dimensional test specimen in ANSYS Fluent 16.0. Therefore, this study investigated both of two-dimensional and three-dimensional for the sake of complicity to check performance of some of turbulence model available in ANSYS Fluent 16.0 to case of flow around a
circular cylinder. Diameter of cylinder is 72.6 mm for two-dimensional case and 100 mm for three-dimensional case. Different diameter is used because of different reference of determining suitable mesh for the two cases, which is Reference [3] for two-dimensional case, and reference [4] for three-dimensional case.

![Diagram](image)

**Figure 2.** Regime of fluid flow across a smooth tube [5]

![Image](image)

**Figure 3.** Turbulent Von Karman Vortex Happened in Cloud [6]
Step 3 in figure 1 is carried out by considering the phenomenon that will be captured. To capture Turbulence Von Karman Vortex Street, fine grid near wall and downstream the circular cylinder is needed. To determine size of numerical calculation domain, we refer to Ref.[3] and Ref.[4]. Calculation domain for two-dimensional case is as illustrated in figure 5, while figure 6 shows calculation domain for three-dimensional case. Three-dimensional case is having 5 mm in thickness. To construct a good mesh for capturing turbulence Von Karman Vortex Street, distance of the first mesh from wall of circular cylinder should be determined first by using equation 1. In equation 1, $\Delta S$ is distance of first mesh from the wall, $y^+$ is a constant value chosen based on turbulence model, $\rho$ is density, and $U$ is velocity of flow. By choosing $y^+$ as 2, $\rho$ as 1.225 kg/m$^3$, $\mu$ as 1.789e-5 kg/ms, and $U$ as 36.51 m/s, we get $\Delta S$ for two-dimensional case is 0.0012 and for three-dimensional case is 0.017 mm. Mesh used in this study is structured grid type for the sake of accuracy. Mesh for two-dimensional case is as shown in figure 7, and for three-dimensional case is as shown in figure 8. More detailed of setting in mesh construction can be found in Ref. [8] chapter 3.
\[ \Delta S = \frac{y'\mu}{\rho U} \]  \hspace{2cm} (1)

**Figure 6.** Calculation domain size for three-dimensional case

**Figure 7.** Mesh for two-dimensional case
Figure 8. Mesh for three-dimensional case

Step 4 in figure 1 by using ANSYS Fluent 16.0 consists of exporting mesh file from ANSYS ICEM and doing many setting of solver as can be found in detailed in Ref. [8]. The main setting is to set reference values and turbulence models. Table 1 shows reference values for two dimensional and three dimensional cases. Table 2 shows turbulence models that can be utilized in ANSYS Fluent 16.0 for two-dimensional and three-dimensional cases. For two-dimensional case, turbulence models to be used are SST k-omega, k-epsilon, Reynolds Stress, and Detached Eddy Simulation (DES). As for three-dimensional case, turbulence model to be utilized are SST k-omega, k-epsilon, Reynolds Stress, Detached Eddy Simulation (DES), and Large Eddy Simulation (LES). Equations utilized in ANSYS Fluent 16.0 are explained in detailed in Ref. [8] chapter 2.

Results from step 4 are processed further by using ANSYS CFD-Post in Step 5 (see figure 1) to be easier to analyze and validate. In other word, step 5 is conducted so that the result of Step 4 will be more user-friendly and easy to compare with experimental data as figure 2 and 3.

Table 1. Reference values used in numerical calculation

| Reference Values     | Two-Dimensional Case | Three-Dimensional Case |
|----------------------|----------------------|------------------------|
| Area (m²)            | 0.004142563          | 0.00314                |
| Density (kg/m³)      | 1.225                | 1.225                  |
| Depth (m)            | 0.072644             | -                      |
| Enthalpy (j/kg)      | 0                    | 0                      |
| Length (mm)          | 72.644               | 100                    |
| Pressure (pascal)    | 0                    | 101325                 |
| Temperature (k)      | 288.16               | 288.15                 |
| Velocity (m/s)       | 50.27                | 36.51                  |
| Viscosity (kg/m-s)   | 1.7894e-05           | 1.7894e-05             |
| Ratio of Specific Heats | 1.4                  | 1.4                    |
Table 2. Turbulence Models available in ANSYS Fluent 16.0

| Turbulence Models            | Two Dimensional Case | Three Dimensional Case |
|------------------------------|----------------------|------------------------|
| Inviscid                     | Available            | Available              |
| Laminar                      | Available            | Available              |
| Spalart-Allmaras (1 eqn)     | Available            | Available              |
| k-epsilon (2 eqn)            | Available            | Available              |
| k-omega (2 eqn)              | Available            | Available              |
| Transition k-kl-omega (3 eqn) | Available            | Available              |
| Transition SST (4 eqn)       | Available            | Available              |
| Reynolds Stress (5 eqn)      | Available            | Not available          |
| Reynolds Stress (7 eqn)      | Not available        | Available              |
| Scale-Adaptive Simulation (SAS) | Available            | Available              |
| Detached Eddy Simulation (DES) | Available            | Available              |
| Large Eddy Simulation (LES)  | Not available        | Available              |

3. Results and Discussion
For case of two-dimensional circular cylinder, turbulence models that employed are SST k-omega, k-epsilon, Reynolds Stress, and Detached Eddy Simulation (DES). In this paper, only result of turbulence model of SST k-omega will be presented. For two-dimensional case, SST k-omega gives the best capturing of turbulent Von Karman Vortex Street (number 4 in figure 2 and figure 3). Figure 9 shows velocity contour of flow around two-dimensional circular cylinder with SST k-omega turbulence model, and figure 10 shows value of $C_L$ of the two-dimensional cylinder with the same turbulence model. More completed result of other turbulence models could be seen in Ref. [8].

Figure 9. Velocity contour of flow around a two-dimensional circular cylinder for Reynolds number of 250,000 by using turbulence model of SST k-omega
Figure 10. Lift Coefficient of the two-dimensional circular cylinder for Reynolds number of 250,000 by using turbulence model of SST k-omega

After getting result that turbulence model of SST k-omega gives the best result for two-dimensional case, investigation of effect of time step size are conducted only for this turbulence model. Figure 11 and 12 gives example of results for time step size of 0.25 ms and 4 ms. It could be seen that smaller time step size gives smoother result of drag coefficient curve over time.

Figure 11. Drag coefficient value for two-dimensional circular cylinder with turbulence model of SST k-omega by using time step size of 0.25 ms
Figure 12. Drag coefficient value for two-dimensional circular cylinder with turbulence model of SST k-omega by using time step size of 4 ms

As for case of three-dimensional circular cylinder, turbulence models that employed are the same with them for two-dimensional case with addition of Large Eddy Simulation (LES). For three-dimensional case, turbulence model of Reynolds stress that gives the best result of capturing turbulent Von Karman Vortex Street. Figure 13 shows velocity contour result for three-dimensional circular cylinder by using turbulence model of Reynolds stress. It captured the same phenomenon as shown in figure 2 number 4 and figure 3. Figure 14 shows lift coefficient curve for the three-dimensional cylinder.

Figure 13. Velocity contour of three-dimensional circular cylinder for Reynolds number of 250,000 by using turbulence model of Reynolds Stress
After getting result that turbulence model of Reynolds stress gives the best result for three-dimensional case, investigation of effect of time step size are conducted only for this turbulence model. Figure 15 and 16 gives example of results for time step size of 0.25 ms and 0.5 ms. It could be seen that smaller time step size gives more correct result.

Figure 14. Lift Coefficient of three-dimensional circular cylinder for Reynolds number of 250,000 by using turbulence model of Reynolds Stress

Figure 15. Drag coefficient value for three-dimensional circular cylinder with turbulence model of Reynolds stress by using time step size of 0.25 ms
Figure 16. Drag coefficient value for three-dimensional circular cylinder with turbulence model of Reynolds stress by using time step size of 0.5 ms

4. Conclusions
From this study, it could be concluded that turbulence models used in a numerical calculation have crucial effect in getting a right result. For two-dimensional circular cylinder, turbulence models that gives results that the most agree with experiments is turbulence model of SST k-omega. As for the case of three-dimensional circular cylinder, turbulence models that gives the best result compared to experiments is Reynolds Stress. Moreover, time step size has very important effect to numerical calculation result. Smaller time step size gives faster and more accurate result although with more iteration number.

References
[1] Tutar, M., and Holdø, A.E., Computational Modelling of Flow around a Circular Cylinder in Sub-critical Flow Regime with Various Turbulence Models, Journal of Numerical Methods in Fluids, Vol. 35, pp. 763-784, 2001.
[2] Bouris, D., and Bergeles, G., Two Dimensional Time Dependent Simulation of The Subcritical Flow in a Staggered Tube Bundle Using a Subgrid Scale Model, Journal of Heat and Fluid Flow, Vol. 20, pp. 105-114.
[3] Kensrud, J.R., Determining Aerodynamics Properties of Sport Ball in Situ, Master Thesis, Washington State University, 2010.
[4] Nidhul, K., Sunil, A.S., Benphil, C.P., Effect of Domain Size and Grid Spacing on Flow Past A Circular Cylinder at Low Reynolds Number, International Journal of Engineering Research and Technology, 2014.
[5] Blevins, R.D., Flow Induced Vibration, 2nd Edition, Van Nostrand Reinhold Co., 1990.
[6] https://commons.m.wikimedia.org/wiki/File:Vortex-street-1-90deg.jpg
[7] Patel, Y., Numerical Investigation of Flow Past a Circular Cylinder and in a Staggered Tube Bundle Using Various Turbulence Models, Master Thesis, Lappeenranta University of Technology, 2010.
[8] Ihsan, M., Analysis of Turbulence Model and Effect of Time Step Size Value of Analysis of Flow around Circular Cylinder, Bachelor Thesis, Institute of Technology Bandung, (to be published on 2017).