LARGE-EDDY SIMULATION OF FLOW PAST A CIRCULAR CYLINDER USING OPENFOAM AND NEKTAR++

Hongyi Jiang

School of Engineering, The
University of Western Australia
Perth, Australia

Xiaoying Ju

School of Engineering, The
University of Western Australia
Perth, Australia

Yucen Lu

School of Engineering, The
University of Western Australia
Perth, Australia

ABSTRACT

Steady incoming flow past a circular cylinder has been a classical problem in fluid mechanics owing to its extensive practical applications in e.g. offshore engineering. In this study, large-eddy simulations are performed for flow past a circular cylinder at the Reynolds number (Re) of 3900. Particular focuses are on the comparisons of different numerical methods and computational domain patterns. The case $Re = 3900$ is computed with both OpenFOAM and Nektar++, which are based on the conventional finite volume method and the high-order spectral/hp element method, respectively. It is found that the computational cost for the Nektar++ model is only less than 10% of that for the OpenFOAM model. In addition, both circular and C-shaped domains are tested for the OpenFOAM and Nektar++ models. It is found that a circular domain is required for the OpenFOAM model to minimise the footprint of mesh non-orthogonality on the simulated flow, while the Nektar++ model does not have strict requirements for the orthogonality of the mesh. The present findings regarding the computational cost and the domain/mesh patterns are expected to be applicable to the numerical modelling of bluff-body flows in general. Based on Nektar++ and the circular domain, additional simulations are performed at $Re = 1000$ and 7000 . For the three Re values investigated, the Strouhal number, hydrodynamic forces and the streamwise and spanwise vorticity fields are examined and compared.

Keywords: Circular cylinder; Wake; Large-eddy simulation

1. INTRODUCTION

Steady incoming flow, of a Newtonian and incompressible fluid, past a smooth and slender circular cylinder (referred to as flow past a circular cylinder hereafter) has been a classical problem in fluid mechanics, owing to its extensive practical applications in the fields of civil, mechanical, hydraulic, offshore and wind engineering (e.g. risers, pipelines and cables used in offshore engineering). Owing to a rapid increase in the

computing power over the past two decades, it has become possible to simulate accurately the flow past a circular cylinder over a range of practical Reynolds numbers using large-eddy simulation (LES), where the Reynolds number $Re (= UD/\nu)$ is defined based on the velocity of the incoming flow (U), the diameter of the cylinder (D), and the kinematic viscosity of the fluid (ν).

A commonly used benchmarking case for flow past a circular cylinder is $Re = 3900$ (e.g. Beaudan and Moin, 1994; Breuer, 1998; Parnaudeau et al., 2008). Despite geometric simplicity of the body, complex flow behaviours exist, which include three-dimensionality and turbulence in the wake region and in the separating shear layers (see e.g. Williamson, 1996). Over the ~ 30 numerical studies in the literature involving the case $Re = 3900$, different numerical methods and computational domain patterns (and consequently mesh patterns) were employed. Commonly used numerical methods include (i) conventional finite difference method, finite volume method and finite element method, and (ii) high-order spectral/hp element method (Karniadakis and Sherwin, 2005). Commonly used domain patterns include circular, rectangular and C-shaped (i.e. circular for the upstream and rectangular for the downstream of the cylinder) domains. However, comparisons of different numerical methods and/or computational domain patterns were scarce.

In light of these knowledge gaps, the present study aims at evaluating the performance of different numerical methods and computational domain patterns on the numerical modelling of flow past a circular cylinder. To this end, the case of flow past a circular cylinder at $Re = 3900$ is simulated using two open-source codes, OpenFOAM version 5.0 (www.openfoam.org) and Nektar++ version 4.4.1 (Cantwell et al., 2015), which correspond to the finite volume method and the spectral/hp element method, respectively. In addition, both circular and C-shaped domains are tested for the OpenFOAM and Nektar++ models.

2. NUMERICAL MODEL

2.1. The OpenFOAM model

In the framework of OpenFOAM, the governing equations are the filtered continuity and incompressible Navier–Stokes equations:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j}, \quad (2)$$

where an overbar denotes a filtered quantity, $x_i = (x_1, x_2, x_3)$ are Cartesian coordinates, which is used interchangeably with (x, y, z) , u_i is the velocity component in the direction x_i , t is time, p is kinematic pressure (i.e. pressure divided by fluid density), ν is kinematic viscosity, and τ_{ij} is the subgrid scale stress tensor. The deviatoric part of τ_{ij} is:

$$\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = -2\nu_t \bar{S}_{ij}, \quad (3)$$

where

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right), \quad (4)$$

where ν_t is the turbulent eddy viscosity. The turbulent eddy viscosity is calculated using the wall-adapting local eddy-viscosity (WALE) model (Nicoud and Ducros, 1999):

$$\nu_t = \left(C_w V^{1/3} \right)^2 \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\bar{S}_{ij} \bar{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}}, \quad (5)$$

where $C_w = 0.325$ is the model constant, V is the volume of a cell, and S_{ij}^d is the traceless symmetric part of the square of the velocity gradient tensor:

$$S_{ij}^d = \frac{1}{2} (\bar{g}_{ij}^2 + \bar{g}_{ji}^2) - \frac{1}{3} \delta_{ij} \bar{g}_{kk}^2, \quad (6)$$

where

$$\bar{g}_{ij} = \frac{\partial \bar{u}_i}{\partial x_j}, \quad (7)$$

$$\bar{g}_{ij}^2 = \bar{g}_{ik} \bar{g}_{kj}. \quad (8)$$

The flow is solved by pisoFoam. The convection, diffusion and time derivative terms of the Navier–Stokes equations are discretized, respectively, using a fourth-order cubic scheme, a second-order linear scheme, and a blended scheme consisting of the second-order Crank–Nicolson scheme and a first-order Euler implicit scheme. The same set of numerical schemes has been used in Jiang et al. (2016), Jiang and Cheng (2017) and Jiang (2020) for the direct numerical simulation (DNS) of flow past a circular cylinder over $Re = 50 - 1000$.

2.2. The Nektar++ model

In the framework of Nektar++, the governing equations are the continuity and incompressible Navier–Stokes equations:

$$\frac{\partial u_i}{\partial x_i} = 0, \quad (9)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j}. \quad (10)$$

Different from the conventional FVM, the Nektar++ model adopts a high-order spectral/ hp element method (Karniadakis and Sherwin, 2005) for the x - y plane, coupled with a Fourier expansion (Karniadakis, 1990) in the spanwise direction. The flow is solved by the unsteady Navier–Stokes solver, a second-order implicit–explicit time-integration scheme, a velocity correction scheme, and a continuous Galerkin projection.

Another difference between the OpenFOAM and Nektar++ models is that the latter does not use a subgrid scale model. Instead, an implicit LES (iLES) is performed in Nektar++ based on the spectral vanishing viscosity (SVV) technique (Kirby and Sherwin, 2006) for stabilising the solution, which is achieved by increasing the viscosity on the expansion modes with the highest frequencies.

2.3. Computational domain and mesh

For the OpenFOAM model, a circular computational domain, as shown in Figure 1(a), is used in the present study. A circular cylinder is placed at the centre of the domain, i.e. $(x, y) = (0, 0)$, while the radius of the domain is $r = 30D$. Figure 1(b) shows a close-up view of the mesh. A relatively high mesh resolution is used in the near-wake region. Specifically, in the circumferential direction, the downstream half of the cylinder is discretised equally with 300 nodes, while the upstream half of the cylinder is discretised (unequally) with 164 nodes. In the radial direction, the height of the first layer of cells next to the cylinder is $1 \times 10^{-4}D$, and a relatively high resolution is used within $7D$ of the cylinder to resolve the near wake. The three-dimensional (3D) mesh is constructed by replicating the mesh in the x - y plane along the spanwise direction. The spanwise domain length and spanwise cell size are $L_z = 3D$ and $\Delta z = 0.025D$, respectively.

For the Nektar++ model, the computational domain size and the general topology of the mesh are the same as those for the OpenFOAM model. However, owing to the use of the high-order spectral/ hp element method, the sizes of the macro-elements for Nektar++ are generally five times larger than those for OpenFOAM in both the radial and circumferential directions. The macro-elements are subdivided using 4th-order Lagrange polynomials (denoted $N_p = 4$) on the Gauss–Lobatto–Legendre points for the quadrilateral expansion. The 3D mesh is constructed by using a Fourier expansion in the spanwise direction. The spanwise domain length is $L_z = 3D$, while the number of Fourier planes is 64.

The boundary conditions for both of the models include a uniform streamwise velocity U at the inlet boundary (i.e. the left half of the outer boundary), a Neumann condition for the velocity at the outlet boundary (i.e. the right half of the outer boundary), and a no-slip condition for the cylinder surface. In addition, the periodic boundary condition is applied to the two lateral boundaries perpendicular to the spanwise direction, owing to its better performance than the symmetry boundary condition (Jiang et al., 2017).

Since the 3D flows are highly turbulent and chaotic, each case is simulated for a period of at least 1400 non-dimensional time units (defined as $t^* = tU/D$). The first 400 time units are used to wash out the transients, while the remainder is used for the statistics and analysis of the flow. The time-step size for the time integration is chosen based on a Courant–Friedrichs–Lewy (CFL) limit of no more than 0.5.

Figure 1. (a) Computational mesh in the x - y plane for the OpenFOAM model, and (b) a close-up view of the mesh.

3. CIRCULAR VERSUS RECTANGULAR DOMAINS

For the numerical modelling of bluff-body flows, some commonly used types of the computational domain in the x - y plane are shown schematically in Figure 2. Figure 2 also shows key partitions (in black solid lines) for the three domains. A common practice for the three domains is that, for the area close to the cylinder (the blue area highlighted in Figure 2), orthogonal mesh is used. Outside the blue area, the mesh needs to conform to the outer boundaries. In particular, the rectangular and C-shaped domains involve a transition of the mesh topology from concentric circles in the blue area towards rectangular cells parallel to the outer boundaries, leading to

non-orthogonality of the cells at the transition of mesh topology, in particular the areas highlighted by red circles in Figure 2(b,c).

Naturally, the circular domain shown in Figure 2(a) and Figure 1(a) has an advantage of minimising non-orthogonality of the mesh. However, for a circular domain, the cell size in the circumferential direction increases linearly with increasing distance of the cell to the cylinder, such that a sufficient mesh resolution may be only available in the near wake. To resolve the relatively far wake, a rectangular (Figure 2b) or a C-shaped (Figure 2c) domain is commonly used. However, due to the introduction of the mesh non-orthogonality, the numerical results may be affected by the mesh (depending also on the numerical method and schemes). As shown by the red circles in Figure 2(b,c), the effect of mesh non-orthogonality takes place at both upstream and downstream sides of the cylinder for a rectangular domain, and occurs at the downstream side only for a C-shaped domain.

Figure 2. Types of the computational domain in the x - y plane: (a) a circular domain, (b) a rectangular domain, and (c) a C-shaped domain. The black solid lines show key partitions for the three domains, while the red dashed circles highlight the areas where mesh non-orthogonality takes place.

Figure 3. Time- and span-averaged streamwise velocity field for $Re = 3900$, simulated with: (a) OpenFOAM and the C-shaped domain, (b) OpenFOAM and the circular domain, (c) Nektar++ and the C-shaped domain, (d) Nektar++ and the circular domain. For the C-shaped domains, the key partitions of the mesh are shown by the blue dashed lines.

Figure 4. Close-up view of the time- and span-averaged streamwise velocity field for $Re = 3900$, simulated with: (a) OpenFOAM and the C-shaped domain, where key partitions of the mesh are shown by the blue dashed lines, and (b) Nektar++ and the C-shaped domain, overlaid with the macro-element mesh.

It is worth noting that the effect of the mesh non-orthogonality on the numerical results may be minor for relatively small Reynolds numbers, such that all three types of the domain shown in Figure 2 may be appropriate. However, such effect enhances with increasing Reynolds numbers, such that it is worthwhile to investigate this effect at a relatively large Reynolds numbers. The Reynolds number of 3900 is chosen for the present study, since this case has been a popular benchmarking case in the literature, where both experimental and numerical results are available.

To this end, the case of flow past a circular cylinder at $Re = 3900$ is simulated with both OpenFOAM and Nektar++, and both the circular and C-shaped domains are tested. Figure 3 shows the time- and span-averaged streamwise velocity field for the four simulations. For the C-shaped domain, the key partitions of the mesh are shown by the blue dashed lines. As shown in Figure 3(a), the flow field simulated with OpenFOAM and the C-shaped domain is strongly affected by the mesh non-orthogonality. Unphysical fluctuations of the contours are observed at the locations where the topology of the mesh changes (i.e. the locations highlighted in red circles in Figure 2(c)). Figure 4(a) shows a close-up view of the flow field at one of these locations, which illustrates more clearly that the fluctuations in the velocity contours indeed originate from the intersection of the partition lines. In addition, small fluctuations are also observed upstream of the cylinder (Figure 3a). In contrast, by using OpenFOAM with the circular domain,

the contours shown in Figure 3(b) are all smooth and there are no unphysical fluctuations. The comparison between the C-shaped and the circular domains further confirms that the unphysical fluctuations observed in Figure 3(a) are indeed due to mesh non-orthogonality. It is noted that in the OpenFOAM model a non-orthogonality correction step has been used, yet it does not resolve the issue. Nevertheless, the hydrodynamic forces predicted with the C-shaped and the circular domains agree well, since the locations of mesh non-orthogonality are relatively far away from the separating shear layer and the recirculation region (Figure 3a).

Figure 3(c,d) shows the velocity fields obtained by using Nektar++ and the C-shaped and the circular domains. Unlike the OpenFOAM case, the spectral/ hp element method adopted by Nektar++ handles the mesh non-orthogonality well. Even by using the C-shaped domain, the velocity contours are highly smooth (Figure 3c). A close-up view of Figure 3(c) is shown in Figure 4(b), where the macro-element mesh is also presented. It is seen that the flow is hardly affected by the mesh pattern.

In summary, for the numerical simulation of flow past a circular cylinder at $Re = 3900$, a circular domain is required for the OpenFOAM model to minimise the footprint of mesh non-orthogonality on the simulated flow, while the Nektar++ model does not have strict requirements for the orthogonality of the mesh.

4. NUMERICAL RESULTS AT DIFFERENT RE VALUES

In this section, the case of flow past a circular cylinder is investigated at three subcritical Re values 1000, 3900 and 7000. The numerical simulations are performed on Nektar++ with the circular domain, owing to the following reasons:

- (i) Based on a comparison of the case $Re = 3900$ simulated with OpenFOAM and Nektar++ (both using the circular domain so as to minimise the effect of mesh non-orthogonality), it is found that the computational cost for the Nektar++ model is only less than 10% of that for the OpenFOAM model. More details on the OpenFOAM and Nektar++ results at $Re = 3900$, including a systematic investigation of the numerical convergence and the computational cost, and a detailed validation against the experimental results, will be reported in a follow-up journal publication.
- (ii) The present study focuses on the flow characteristics in the near wake, such that the circular domain is appropriate.

For the cases of $Re = 1000, 3900$ and 7000 , some key parameters for the numerical simulations are listed in Table 1. The adequacy of $N_p = 4$ is checked by additional simulations with $N_p = 5$ for $Re = 3900$ and 7000 . The hydrodynamic forces predicted by $N_p = 4$ and 5 agree well. The spanwise domain length (L_z) and the number of Fourier planes used for each Re are also determined based on a mesh dependence study (details omitted for simplicity). In general, with the increase in Re the L_z value is reduced, while the spanwise resolution (represented by the number of Fourier planes per unit span length) increases.

Table 1. Key numerical parameters for the cases $Re = 1000, 3900$ and 7000 .

Re	N_p	L_z	Fourier planes
1000	4	6	128
3900	4	3	64
7000	4	3	128

Figure 5. The (a) $\overline{C_D}$ - Re , (b) C_L' - Re , and (c) St - Re relationships for flow past a circular cylinder.

Figure 5 shows the time-averaged drag coefficient ($\overline{C_D}$), the root-mean-square lift coefficient (C_L') and the Strouhal number (St) obtained at $Re = 1000, 3900$ and 7000 , where

$$C_D = \frac{F_D}{\frac{1}{2}U^2 DL_z} \quad (11)$$

$$C_L = \frac{F_L}{\frac{1}{2}U^2 DL_z} \quad (12)$$

$$C_L' = \sqrt{\frac{1}{N} \sum_{i=1}^N (C_{L,i} - \overline{C_L})^2} \quad (13)$$

$$St = \frac{f_L D}{U} \quad (14)$$

where F_D and F_L are the drag and lift forces integrated over the cylinder surface (divided by the fluid density already), respectively, N is the number of values in the time history of C_L , and f_L is the peak frequency derived from the fast Fourier transform (FFT) of the time history of C_L .

The hydrodynamic forces and the Strouhal number shown in Figure 5 are also compared with the experimental results of Wieselsberger (1922) and Norberg (2003). In general, the present numerical results agree well with the experimental data.

Figure 6 shows the frequency spectra of C_L for $Re = 1000$, 3900 and 7000. For each case, a well-defined peak is observed in the frequency spectrum, which corresponds to the St value shown in Figure 5(c). For $Re = 7000$, the peak in the frequency spectrum is less sharp-peaked than the other two cases, which is likely because the flow at $Re = 7000$ is more chaotic.

Figure 6. Frequency spectra of C_L for $Re = 1000$, 3900 and 7000.

Figure 7 shows the instantaneous streamwise (ω_x) and spanwise vorticity (ω_z) fields for $Re = 1000$, 3900 and 7000, where ω_x and ω_z are defined as:

$$\omega_x = \left(\frac{\partial u_z}{\partial y} - \frac{\partial u_y}{\partial z} \right) \frac{D}{U} \quad (15)$$

$$\omega_z = \left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right) \frac{D}{U} \quad (16)$$

For all three cases, the primary vortex shedding is coupled with strong influence of the streamwise vortices. With the increase in Re , the streamwise location for the development of the

primary and streamwise vortices moves closer to the cylinder. At $Re = 1000$, the streamwise vortices develop a short distance away from the cylinder (Figure 7a), whereas at $Re = 7000$ the streamwise vortices emerge right next to the cylinder (Figure 7c). At $Re = 3900$, both conditions may be observed (even at the same time instant) (Figure 7b). In particular, traces of the shear-layer instability are observed in Figure 7(b) at the far-end of the span.

Figure 7. Instantaneous streamwise and spanwise vorticity fields for (a) $Re = 1000$ ($\omega_z = \omega_x = \pm 3$), (b) $Re = 3900$ ($\omega_z = \omega_x = \pm 8$), and (c) $Re = 7000$ ($\omega_z = \omega_x = \pm 12$). The translucent iso-surfaces represent spanwise vortices, while the opaque iso-surfaces represent streamwise vortices. The positive and negative vortices are shown in red and blue, respectively. The flow is from left to right past the cylinder on the left.

Figure 8 shows the side view of the instantaneous vorticity fields. Figure 8(a) shows the spanwise vorticity field for $Re = 1000$, where the separating shear layers and the alternating shedding of the primary vortices are observed. Owing to the development of turbulence alongside vortex shedding, the primary vortices dissipate very quickly in the near wake. Figure 8(b–d) shows the streamwise vorticity field for $Re = 1000$, 3900

and 7000. The streamwise vorticity field shows both the traces of the primary vortex shedding and the streamwise vortices in between of the primary vortices. The streamwise vorticity fields are also presented in Figure 9 in the top view. With the increase in Re , the streamwise vortices become smaller in size and increasingly chaotic, which is consistent with the increasingly turbulent nature of the wake.

Figure 8. Side view of the instantaneous vorticity fields for (a) $Re = 1000$ ($\omega_z = \pm 3$), (b) $Re = 1000$ ($\omega_x = \pm 3$), (c) $Re = 3900$ ($\omega_x = \pm 8$), and (d) $Re = 7000$ ($\omega_x = \pm 12$). The positive and negative vortices are shown in red and blue, respectively. The flow is from left to right past the cylinder on the left.

5. CONCLUSIONS

The canonical case of flow past a circular cylinder at $Re = 3900$ is computed with both OpenFOAM and Nektar++. It is found that the computational cost for the Nektar++ model is only less than 10% of that for the OpenFOAM model. It is also found that a circular domain is required for the OpenFOAM model to minimise the footprint of mesh non-orthogonality on the simulated flow, while the Nektar++ model does not have strict requirements for the orthogonality of the mesh. In addition, LES results of $Re = 1000$, 3900 and 7000 are examined and compared.

ACKNOWLEDGEMENTS

The first author would like to acknowledge the support from the Australian Research Council through the DECRA scheme (Grant No. DE190100870). This work was supported by resources provided by the Pawsey Supercomputing Centre with funding from the Australian Government and the Government of Western Australia.

Figure 9. Top view of the instantaneous streamwise vorticity fields for (a) $Re = 1000$ ($\omega_x = \pm 3$), (b) $Re = 3900$ ($\omega_x = \pm 8$), and (c) $Re = 7000$ ($\omega_x = \pm 12$). The positive and negative vortices are shown in red and blue, respectively. The flow is from left to right past the cylinder on the left.

REFERENCES

- [1] Beaudan, P., Moin, P., 1994. Numerical experiments on the flow past a circular cylinder at a sub-critical Reynolds number', *Report No. TF-62*, Thermosciences Division, Department of Mechanical Engineering, Stanford University.
- [2] Breuer, M., 1998. Large eddy simulation of the subcritical flow past a circular cylinder: numerical and modeling aspects. *International Journal for Numerical Methods in Fluids* 28, 1281–1302.
- [3] Cantwell, C.D., et al., 2015. Nektar++: An open-source spectral/hp element framework. *Computer Physics Communications* 192, 205–219.
- [4] Jiang, H., 2020. Separation angle for flow past a circular cylinder in the subcritical regime. *Physics of Fluids* 32, 014106.
- [5] Jiang, H., Cheng, L., 2017. Strouhal–Reynolds number relationship for flow past a circular cylinder. *Journal of Fluid Mechanics* 832, 170–188.
- [6] Jiang, H., Cheng, L., An, H., 2017. On numerical aspects of simulating flow past a circular cylinder. *International Journal for Numerical Methods in Fluids* 85, 113–132.
- [7] Jiang, H., Cheng, L., Draper, S., An, H., Tong, F., 2016. Three-dimensional direct numerical simulation of wake transitions of a circular cylinder. *Journal of Fluid Mechanics* 801, 353–391.
- [8] Karniadakis, G.E., 1990. Spectral element-Fourier methods for incompressible turbulent flows. *Computer Methods in Applied Mechanics and Engineering* 80, 367–380.
- [9] Karniadakis, G.E., Sherwin, S.J., 2005. Spectral/hp Element Methods for CFD, *Oxford University Press*.
- [10] Kirby, R. M., Sherwin, S. J., 2006. Stabilisation of spectral/hp element methods through spectral vanishing viscosity: Application to fluid mechanics modelling. *Computer Methods in Applied Mechanics and Engineering* 195, 3128–3144.
- [11] Norberg, C., 2003. Fluctuating lift on a circular cylinder: review and new measurements. *Journal of Fluids and Structures* 17, 57–96.
- [12] Parnaudeau, P., Carlier, J., Heitz, D., Lamballais, E., 2008. Experimental and numerical studies of the flow over a circular cylinder at Reynolds number 3900. *Physics of Fluids* 20, 085101.
- [13] Wieselsberger, C., 1922. New data on the laws of fluid resistance. *NACA Technical Note 84*. National Advisory Committee for Aeronautics, Washington, DC, United States.
- [14] Williamson, C.H.K., 1996. Vortex dynamics in the cylinder wake. *Annual Review of Fluid Mechanics* 28, 477–539.