Comparisons of Subgrid-Scale Models for OpenFoam Large-Eddy Simulation

Zhipeng Feng*, Huanhuan Qi, Xuan Huang, Shuai Liu, Jian Liu

Science and Technology on Reactor System Design Technology Laboratory, Nuclear Power Institute of China, Chengdu, Sichuan, China

*Corresponding author: zhipengfeng@npic.ac.cn

Abstract. OpenFOAM is a free, open-source software package that can be used for the solutions of computational fluid dynamics and simulation of various fluid flow processes. Nevertheless, OpenFOAM still lacks default settings and a large number of different numerical schemes and turbulent models should be validated. In this paper, the unsteady flow around a cylinder (Re=3900) is calculated by the large eddy simulation of OpenFOAM. The predictions include the drag and lift coefficient, the pressure distribution around the cylinder, the velocity distribution and Reynolds stress distribution in the wake region, as well as the prediction of the recirculation length and separation angle. Thanks to several simulations, these five subgrid-scale (SGS) models are compared and studied: The Smagorinsky SGS model, wall adaptive local eddy visibility SGS model, dynamic SGS model with Lagrangian averaging, dynamic one equation eddy viscosity model, one equation eddy viscosity model. The numerical results are verified with the published experimental data.

1. Introduction

The OpenFOAM uses the finite volume method (FVM) to solve the Navier-Stokes equations, and provides a series of numerical schemes, solution methods and turbulence models. Users can freely define a separate scheme for each discrete item, such as pressure, velocity, viscosity, etc.

Flow over a circular cylinder is the most typical benchmark case to evaluate the performance of models in the prediction of separated flows [1]. Lourenco and Shih [2] used PIV to measure the flow field. Parnaudeau et al. [3] also conducted experiments and numerical simulations with particle image velocimetry (PIV) and hot-wire anemometry (HWA) methods. Their PIV data is inconsistent with the data obtained by Lourenco and Shih in the very near wake, but both data sets are consistent with the HWA data obtained by Ong and Wallace [4], far away from the near wake. More precisely, a first flow state in the near wake of a circular cylinder is characterized by a U-shaped profile of the stream-wise velocity, while a second flow state is characterized by a V-shaped profile. Several existing studies [1][3] confirmed that the U-shaped state is correct. Therefore, this specific flow problem has become a classic case for testing numerical simulation.

Although many researchers have validated LES in OpenFOAM, there is still a lack of a comprehensive benchmark case to compare and study the performance of LES SGS models quantitatively, so the comparative study of numerical benchmark cases is of great significance for researchers in the field of CFD. Therefore, the purpose of this paper is to quantitatively compare the performance of SGS model...
in OpenFOAM by studying this benchmark problem, and to establish a new comprehensive benchmark case.

2. Control equations
In LES, large eddies are resolved directly, while small eddies are modelled. The control equations of large eddy can be obtained by filtering the N-S equation in physical space.

\[ \frac{\partial (\rho \hat{u}_i)}{\partial x_i} = 0 \]  
\[ \frac{\partial (\rho \hat{u}_i)}{\partial t} + \frac{\partial (\rho \hat{u}_i \hat{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \hat{u}_i}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left[ \rho (u_j \hat{u}_j - \bar{u}_j \bar{u}_j) \right] \]

Where, (\( \tilde{\cdot} \)) denotes the filtering operation. The filtered velocity in the \( i \) th direction is \( \bar{u}_i \), the unclosed terms are the subgrid-scale stresses \( \tau_{ij} = \rho \left( u_j \bar{u}_j - \bar{u}_j \bar{u}_j \right) \) and need to be modeled by different subgrid-scale models. Modelling of the subgrid-scale stress terms are performed in terms of the eddy viscosity using the common Boussinesq approximation:

\[ \tau_{ij} = -\frac{1}{3} \tau_{kk} \delta_{ij} = -2 \mu_t \bar{S}_{ij} \]

Where, \( \mu_t \) is the subgrid turbulent viscosity \( \bar{S}_{ij} \) is the strain rate tensor, defined as \( \bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \).

For the modelling of subgrid turbulent viscosity \( \mu_t \), OpenFOAM provides five models:
- Smagorinsky model (SMAG)
- Dynamic Smagorinsky-Lilly model (DSL)
- WALE model (WALE)
- One-equation turbulent kinetic energy model (KEqn)
- Dynamic kinetic energy Subgrid-scale model (DKE)

3. Numerical model

3.1. Numerical solutions
The main discretization schemes used in present study are as follows: implicit Euler scheme for temporal term; Second order central difference scheme for gradient discretization; linear upwind (linearUpwind), Limited linear difference(limitedLinear), Linear interpolation (linear) scheme for different variables of divergence terms; Gauss linear limited corrected schemes for Laplacian terms; Linear interpolation (linear) for interpolation scheme; The explicit non-orthogonal correction schemes for surface normal gradient schemes(corrected).

PISO (pisoFoam Solver) is used to solve pressure-velocity coupling. The momentum equation is solved by geometric algebraic multigrid solver (Geometric-Algebraic Multigrid) and Gauss-Seidel smoother. The other quantities are solved by smooth solver (smootherSolver) and symGaussSeidel smoother. The message passing interface (MPI) method is used for Parallel computing.

3.2. Computational domain and grid
The domain size is an important parameter that affects the steady and unsteady flow around a cylinder. Figure 1 shows the computational domain. Two domains are established to study the influence of the
flow field size: \( L_x \times L_y \times L_z = 20D \times 10D \times \pi D \), \( L_x \times L_y \times L_z = 40D \times 20D \times \pi D \). \( L_x \), \( L_y \), \( L_z \)-spatial dimension of a computational domain in stream-wise (x), transverse (y) and span-wise (z) coordinates.

The details of the computational domain and grid are shown in Figure 2. The number of elements in the circumferential direction is 1% \( D \) on the cylinder surface, which is covered by approximately 316 around the cylinders. Other detailed parameters are shown in Table 1. Four different structured grids are established for the two computational domains:

Mesh1 consists of \( 1.13 \times 10^6 \) cells. The grid details of the blocks of flow field and around cylinder are shown in Figure 2.

Mesh2 consists of \( 3.94 \times 10^6 \) cells and it was obtained by doubling the grid numbers in x-y direction of the far field and increasing from 32 cells to 48 cells in the span-wise direction.

The domain of Mesh1 and Mesh2 is doubled to create Mesh3 and Mesh4. The grid resolution of Mesh3 and Mesh4 is consistent with Mesh1 and Mesh2 respectively. But the grid size in span-wise direction is 32 for both Mesh3 and Mesh4.

![Figure 1. computational domain](image1.png)

![Figure 2. domain blocks and grid details](image2.png)

### Table 1. Details of the Grid

| Mesh   | \( L_x \) | \( L_y \) | \( N_c \) | \( N_z \) | \( N_T \times 10^6 \) |
|--------|----------|----------|-----------|-----------|---------------------|
| Mesh1  | 20D      | 10D      | 316       | 32        | 1.13                |
| Mesh2  | 20D      | 10D      | 316       | 48        | 3.94                |
| Mesh3  | 40D      | 20D      | 316       | 32        | 2.40                |
| Mesh4  | 40D      | 20D      | 316       | 32        | 5.27                |

### 3.3. Boundary conditions

The boundary conditions used in this paper are as follows:

A Uniform, constant, undisturbed velocity profile at the inlet boundary is specified. The velocity is defined by \( Re=3900 \).

At the outlet boundary, the normal gradient of pressure is set to 0, the velocity and \( k \) (if the kinetic energy model is used) are set to inletOutlet.

The top, bottom and side boundary are considered as no-slip condition and set to symmetry.

For the cylinder surface, no slip condition is applied. The wall functions are \( kqRWallFunction \) and \( nutkWallFunction \).
4. Results and discussions

In the practice, different researchers use different averaging time to calculate statistical data. For example, the statistical data of Kravchenko and Moin [5] considered 7 vortex shedding periods, Dong et al. [6] used 40-50 vortex shedding periods, Ma et al [7] used 131 vortex shedding periods. Parnaudeau [3] observed that the convergence level of a set of statistical data is difficult to evaluate in practice. 180 vortex shedding periods are used.

The lift and drag coefficients are defined as:

\[ C_l = \frac{F_l}{0.5 \rho U_0^2 A_x}, \quad C_d = \frac{F_d}{0.5 \rho U_0^2 A_y} \]  

(4)

where, \( F_l \) and \( F_d \) are the lift force and drag force acting on the cylinder, \( \rho \) is the fluid density, \( U_0 \) is the inlet velocity, \( A_x \) and \( A_y \) are the projection areas of the cylinder in the direction of drag and lift.

4.1. Study of the Grid and computational domain

Table 2 summarizes the main averaging flow parameters, where, \( \bar{C}_d \) is the mean drag coefficient, \( C_{RMS} \) is the root mean square value of lift coefficient, \( St \) is Strouhal number, \( \bar{\theta}_s \) is the separation angle, \( \bar{C}_{pb} \) is the negative value of back pressure coefficient, \( \bar{u}_{min} \) is the minimum value of average stream-wise velocity on the wake centerline, and \( L_r \) is the recirculation length. It can be seen that the time averaged flow parameters obtained by the four grids are basically consistent with the existing results.

| Data from | \( \bar{C}_d \) | \( C_{RMS} \) | \( S_t \) | \( \bar{\theta}_s \) | \( \bar{C}_{pb} \) | \( \bar{u}_{min} / U_0 \) | \( L_r / D \) |
|-----------|----------------|--------------|--------|--------------|----------------|------------------|----------|
| Kravchenko and Moin.[5], LES | 1.04 | 0.21 | 88.0° | 0.94 | -0.37 | 1.35 |
| Lourenco and Shih.[2], PIV | 0.99 | 0.04-0.15 | 0.22 | 86° | -0.24 | 1.19 |
| Lysenko et al.[1], LES (SMAG) | 1.18 | 0.44 | 0.19 | 89° | 0.8 | -0.26 | 0.9 |
| Lysenko et al.[1], LES (TKE) | 0.97 | 0.09 | 0.209 | 88° | 0.91 | -0.27 | 1.67 |
| Ma et al.[7], Case II, DNS | 0.84 | 0.22 | - | - | - | 1.59 |
| Ong and Wallace[4], experiment | 0.98 | 0.21 | - | - | - | 1.51-1.56 |
| Parnaudeau et al.[3], PIV | - | 0.21 | - | -3.4 | 1.56 |
| Mesh1, LES (SMAG) | 1.07 | 0.093 | 0.228 | 87.3° | 1.006 | -0.301 | 1.692 |
| Mesh2, LES (SMAG) | 1.05 | 0.074 | 0.222 | 86.6° | 1.000 | -0.280 | 1.570 |
| Mesh3, LES (SMAG) | 1.00 | 0.066 | 0.213 | 85.6° | 0.866 | -0.319 | 1.694 |
| Mesh4, LES (SMAG) | 1.01 | 0.073 | 0.209 | 85.5° | 0.860 | -0.313 | 1.808 |

The distribution of the mean pressure coefficient on the cylinder’s surface is plotted in Figure 3. The surface pressure coefficient is defined as follows:

\[ C_p = 1 - \frac{P_0 - p_d}{0.5 \rho U_0^2} \]  

(5)
Where, $p_0$ is the mean pressure at $\theta=0$ (stagnation point), $p_\theta$ is the mean pressure at any surface position angle ($\theta$), $U_0$ is the inlet velocity, $\rho$ is the fluid density. Figure 4 shows the inflow direction and schematic of angular position.

The surface pressure distribution reveals that the present grid resolution has no obvious effect on the surface pressure, but the influence of the computational domain is greater. The smaller computational domain, especially upstream, will lead to a larger absolute value of the mean back pressure coefficient, which will lead to a larger drag coefficient. Therefore, the reasonable simulation should be carried out on the computational domain $L_x \times L_y \times L_z = 40D \times 20D \times \pi D$.

![Figure 3. Surface pressure distribution](image1)

![Figure 4. Schematic of angular position](image2)

### 4.2. Time Averaged Flow Characteristics

Mesh4 is selected to study the influence of subgrid-scale model. The five SGS models (SMAG, DSL, WALE, KEqn, DKE) are used to predict the flow statistics, and the results are then compared to the experimental data in the following literature: PIV data in Lourenco and Shih [2], PIV data in Parnaudeau et al. [3], HWA data in Ong and Wallace [4].

Figure 5 compares the mean flow parameters of different SGS models. The error bar in the figure represents the relative error of each SGS model relative to SMAG. It can be seen from the comparison that for different SGS models, $C_{l_{RMS}}$ and $U_{\text{min}}$ are most sensitive to SGS models, with the maximum relative errors of 19.9% and 22.5% respectively, followed by $L_\tau$, with the relative error within 7%, while the difference of $\tilde{C}_d$, $\tilde{U}_r$, $\tilde{C}_{pb}$ and $S_l$ (except DSL, with the relative errors of -6.1%) are all within 3%.

For $C_{l_{RMS}}$, which is the most sensitive to subgrid-scale model, Norberg [8] observed the dispersion of measurement data, especially for $Re=3000$ to $5000$, which varies from $C_{l_{RMS}}=0.03$ to 0.1. The fluctuating lift coefficient obtained by the present numerical model is $C_{l_{RMS}}=0.056$ to 0.094. This result supports Norberg [8]. $L_\tau$ shows greater dispersion, $L_\tau / D = 0.9$–1.808. The predicted values of the five SGS models are $L_\tau / D = 1.67$–1.808, which is consistent with the existing research conclusions.

The present numerical results show that $C_{l_{RMS}}$ obtained by Dynamic k-equation eddy-viscosity model is only 18% larger than that of SMAG model. Furthermore, it will be seen from the later analysis that all the five subgrid-scale models have successfully predicted the U-shape of mean streamwise velocity at $x/D = 1.06$. It is not the case that only Dynamic k-equation eddy-viscosity model did as pointed out by Lysenko [1].

Figure 6 shows the distribution of the mean pressure coefficient in the circumferential direction of the cylinder surface under different SGS models. The influence of the subgrid-scale model on the cylinder surface pressure is not obvious.
Figure 5. The variation of time averaged flow parameters with different SGS models

Figure 6. Surface pressure distribution

4.3. Flow characteristics along wake centerline

Figure 7 lists the mean streamwise velocity and variance of the streamwise velocity in the wake centerline. It shows that the mean streamwise velocity in the wake centerline is not sensitive to the SGS model, but the SGS model has a significant impact on the variance of the streamwise velocity in the wake centerline. Because of the large dissipation of Smagorinsky-Lilly model, the predicted value of Smagorinsky-Lilly is the smallest, the predicted value of Dynamic k-Equation Model is the closest to the experimental value, and the predicted value of other SGS models lies between them.
5. Conclusions
The purpose of this paper is to compare the performance of OpenFOAM sub grid model quantitatively by studying the benchmark problem, and to discuss some key factors that affect the prediction accuracy. The conclusion is as follows:

(1) For the mean flow parameters, $C_{\text{RMS}}, \bar{u}_{\text{min}}, \bar{E}$, are more sensitive to grid resolution, while $S_{\text{r}}, C_{p}$ are more sensitive to computational domain size; For the distribution of statistics ($\bar{u}, \bar{v}, \bar{u'u'}, \bar{v'v'}, \bar{u'v'}$) in the wake region, the influence of grid resolution is more significant than that of the computational domain size; The influence of the computational domain type (such as rectangular domain and circular domain) can be ignored.

(2) The subgrid-scale model has little influence on flow characteristics of flow past a circular cylinder. In general, Smagorinsky-Lilly has the lowest predicted value due to its large dissipation. The predicted value of Dynamic k-Equation Model is closest to the experimental value, and the predicted value of other SGS models lies between them; All of the five subgrid-scale models in the present work have successfully predicted the U-shape of the mean streamwise velocity at $x/D=1.06$, which is not the case that only Dynamic k-equation eddy-viscosity model did as pointed out by Lysenko [1].

Acknowledgement
This work was financially supported by the National Science Foundation of China (No. 51606180, 11872060, 11902315) and China Scholarship Council.

References
[1] Dmitry, A Lysenko, et al, 2012, Large-Eddy Simulation of the Flow Over a Circular Cylinder at Reynolds Number 3900 Using the OpenFOAM Toolbox. Flow Turbulence Combust 89: 491-518.
[2] Lourenco L., Shih C., 1993, Characteristics of the plane turbulent near wake of a circular cylinder, a particle image velocimetry study. Published in: Beaudan, P., and Moin, P., Report No. TF62, Thermo sciences Division, Department of Mechanical Engineering, Stanford University.
[3] Parnaudeau, P., et al, 2008, Experimental and numerical studies of the flow over a circular cylinder at Reynolds number 3900. Physics of Fluids 20(085101): 1-13.
[4] Ong, L., Wallace, J., 1996, The velocity field of the turbulent very near wake of a circular cylinder. Experiments in Fluids 20(6): 441-53.
[5] Kravchenko, A. G., Moin, P., 2000, Numerical studies of flow over a circular cylinder at...
ReD=3900. Phys Fluids 12(2): 403-417.

[6] Dong, S., et al, 2006, A combined direct numerical simulation-particle image velocimetry study of the turbulent near wake. Journal of Fluid Mechanics 569: 185-207.

[7] Ma, X., et al, 2000, Dynamics and low-dimensionality of a turbulent near wake. Journal of Fluid Mechanics 410: 29-65.

[8] Norberg, C., 2001, Flow around a circular cylinder: aspects of fluctuating lift. Journal of Fluids and Structures, 15: 459-469.