Large eddy simulation for improvement of performance estimation and turbulent flow analysis in a hydrodynamic torque converter

Chunbao Liua,b, Jing Lia, Weiyang Bu a, Wenxing Ma a,b, Guang Shen c and Zhe Yuana

aSchool of Mechanical Science and Engineering, Jilin University, Changchun, People’s Republic of China; bState Key Laboratory of Automotive Simulation and Control, Jilin University, Changchun, People’s Republic of China; cState Grid Xiaoshan Power Supply Company, Hangzhou, People’s Republic of China

ABSTRACT
Computational fluid dynamics (CFD) has been widely applied as an effective tool for optimizing products and reducing production cycles in many industrial fields; consequently, engineers are constantly pursuing higher accuracy in the performance predictions of CFD methods. In this paper, an analysis for the flow field of a hydrodynamic torque converter (TC) is conducted to evaluate CFD applications in detail. In the past, Reynolds-averaged Navier–Stokes (RANS) simulations have always played a dominant role in the numerical modeling of TCs because of their efficient calculation speed. However, most RANS models are unable to capture the complicated transient flows whose performance estimation errors are generally greater than 10%. Therefore, large eddy simulation (LES) with various sub-grid scale (SGS) models are applied in order to explore feasible methods for improving numerical accuracy and capturing the detailed transient flow phenomena. The effectiveness of the LES method is verified by comparing the numerical results with experimental data. Although the grid resolution is not fine enough due to the limitations of the high-performance computer (HPC) used, LES with dynamic kinetic energy transport (KET) models were still able to obtain an excellent description of both the near-wall flow and the main-stream flow via quantitative and qualitative analyses. The maximum error in the capacity factor ($CF$) is remarkably reduced to 4.4%. It is therefore beyond doubt that applying LES methods using coarser grid resolutions can still guarantee higher prediction accuracy through the reasonable selection of SGS models, which can effectively reduce the computing capacity requirements and contribute to the design process of TCs.

ARTICLE HISTORY
Received 6 August 2017
Accepted 13 June 2018

KEYWORDS
hydrodynamic torque converter; transient simulation; LES; SGS models

Nomenclature

- $Re$: Reynolds number
- $CF$: capacity factor
- $SR$: speed ratio
- $TR$: torque ratio
- $CS$: Smagorinsky coefficient
- $k$: turbulent kinetic energy ($m^2/s^2$)
- $\kappa$: von Kármán constant
- $\mu_t$: turbulent viscosity ($m^2/s$)
- $\mu$: dynamic viscosity ($m^2/s$)
- $N$: rotating speed (rpm)
- $P$: static pressure (pa)
- $R$: radius (mm)
- $\omega$: relative velocity (m/s)
- $\eta$: efficiency
- $\tau_{ij}$: SGS stress tensor
- $\epsilon$: turbulent dissipation rate ($m^2/s^3$)
- $\rho$: density ($kg/m^3$)
- $v$: mean velocity (m/s)
- $L$: characteristic length scale (m)

1. Introduction
An important component of automatic transmissions, the hydrodynamic torque converter (TC) is comprised of three elements: (1) the pump, which directly connects to the engine and transfers the power of the input shaft to the automatic transmission fluid (ATF); (2) the turbine, which rotates under the impact action of the high-speed ATF and transfers the mechanical energy to the output shaft; and (3) the stator, which is stationary and redirects the ATF into the pump to generate the circulating flow in the blade cascade (Figure 1). The primary function of the TC is torque multiplication, but it also multiplies output torque at a low speed and absorbs torsional vibration. Due to their superior fluid power transmission, TCs are widely applied in construction engineering machinery, automobiles, heavy military vehicles, and other technologies (Cigarini & Jonnavithula, 1995). For instance, the use of TCs in automobiles results in better automatic adaptability, a longer service life, a more comfortable driving experience, simpler operation, and so on.
Figure 1 shows that the TC has a very complex enclosed three-dimensional geometry. The extremely complicated unsteady internal flow is difficult to capture without the use of sophisticated, high-cost flow visualization instruments such as a particle image velocimetry. Over the past two decades, CFD applications have increasingly been integrated into the TC design process. To begin with, the flow structures were simulated separately in a single element. Through modeling the pump element individually and using a low Reynolds number \((Re)\) two-equation model to close the time-averaged Navier–Stokes equations, the predicted result of the \(Re\) was achieved roughly 5100 (By, Kunz, & Lakshminarayana, 1995). Considering that only a few simulation results were in good agreement with the experimental values, it was necessary to simulate all the elements in the flow field simultaneously. Shin, Chang, and Athavale (1999) used a standard \(k-\epsilon\) model to investigate the influence that the tip bending near the exit of the pump blade has on the internal flow, finding that the pressure decrease near the exit leads to a reduction in the capacity factor \((CF)\). In order to improve the stall torque ratio, a modified stator structure was developed to effectively reduce the flow loss on the suction side of the blade surface (Dong, Korivi, Attibele, & Yuan, 2002). Shin, Lee, Hong, and Joo (2003) proposed a newly slotted stator blade to enhance the performance of the TC, and the \(CF\) increased by more than 15%. Nevertheless, the prediction accuracy was still not quite satisfactory compared to the experimental data.

CFD application is an ongoing priority for improving performance estimation and turbulent flow analysis. B. S. Kim, Ha, Lim, and Cha (2008) investigated transient flow distribution and energy loss by applying moving wall boundary conditions to the element interface, obtaining results that are more reliable and satisfactory than those obtained using a traditional single flow passage model. Liu and Ma (2010) used an Renormalization Group (RNG) \(k-\epsilon\) model and sliding mesh method to obtain the complicated unsteady flow field of the TC, with a deviation of less than 7.3%. Lei, Wang, Liu, and Li (2012) analyzed the flow rate, pressure, and flow loss by establishing a full three-dimensional passage model, providing a direction for TC design optimization. However, most of these Reynolds-averaged Navier–Stokes (RANS) studies suffer from a lack of clarity in the description of the flow structures.

Some typical applications of CFD are described in Figure 2, based on a review of the literature. A great number of studies have incorporated the RANS method into the TC design process, but there are only a few reports about the application of the large eddy simulation (LES) method. In fact, many transient phenomena – such as non-uniform velocity profiles, flow separation, secondary circulation, and the unsteady interactions of rotating cascades – are not accurately predicted by RANS steady calculations. The LES method resolves the turbulence motions of large-scale structures directly and models the small-scale eddies through the use of sub-grid scale (SGS) models. It has been demonstrated that
the LES method that provides transient results is a reasonable alternative to the performance simulation of the TC. Liu, Liu, and Ma (2015) concluded that the transient vorticity features, shedding, and rupture at the trailing edge can be clearly captured by the LES method. Tasaka, Oshima, Fujimoto, and Kishi (2017) evaluated the feasibility of LES unsteady calculation using coincident results in the oil flow pattern at the correct position. However, these two studies only employed Smagorinsky–Lilly (SL) models of the LES method, and failed to show significant differences compared to the SGS models.

The current study describes the entire process of accurately predicting TC performance. To achieve this, it is necessary not only to investigate the unsteady internal flow field of the TC but also to perform a comprehensive evaluation of various SGS models. Furthermore, the TC geometries, mesh generation, CFD numerical simulations, and flow structures are presented in great detail. The accuracy of the LES method is evaluated via a comparative analysis between the calculated results and experimental data. Additionally, a more accurate SGS model – i.e., a dynamic kinetic energy transport (KET) model – is provided in order to simulate the turbulence characteristics of the TC. Lastly, the feasibility of using different CFD methods as industrial diagnostic tools is evaluated according to the presence of obvious improvements in performance predictions.

2. Numerical model

2.1. Governing equations

The governing equations employed for LES are obtained by filtering the time-dependent Navier–Stokes equations in either configuration space or Fourier space. The filtering process effectively filters out the eddies, whose scale is smaller than the filter width and grid spacing in the computation. The filtering operation can be expressed as

$$\bar{\phi}(x) = \int_D \phi(x')G(x,x')dx', \quad (1)$$

where $D$ is the fluid domain and $G$ is the filtering function that determines the scale of the resolved eddies. The finite-volume discretization itself implicitly provides the filtering operation:

$$\bar{\phi}(x) = \frac{1}{V} \int_v \phi(x')dx', x' \in v, \quad (2)$$
where $V$ is the volume of the cell and $G(x, x')$ is the filter function, which is given by

$$G(x, x') = \begin{cases} 1/V, & x' \in v \\ 0, & x' \text{ otherwise} \end{cases}.$$  

(3)

The filtering procedure for the continuity and momentum equations used in LES (Moeng & Sullivan, 2002) is

$$\begin{align*}
\frac{\partial \tilde{u}_i}{\partial x_i} &= 0 \\
\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial (\tilde{u}_i \tilde{u}_j)}{\partial x_j} &= \mu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{1}{\rho} \frac{\partial \rho}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} ,
\end{align*}$$  

(4)

where $\mu$ is the dynamic viscosity and $\tau_{ij}$ is the SGS stress tensor, which is given by

$$\tau_{ij} \equiv \tilde{u}_i \tilde{u}_j - \bar{u}_i \bar{u}_j.$$  

(5)

Since the Boussinesq hypothesis is employed for $\tau_{ij}$, it can be computed from

$$\tau_{ij} = \frac{1}{3} \delta_{ij} \tau_{kk} - \mu_t \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right),$$  

(6)

where $\delta_{ij}$ is the Kronecker delta and $\mu_t$ is the turbulent viscosity. Due to the different definitions of $\mu_t$, various SGS models of the LES are proposed in the next section.

### 2.2. SGS models

#### 2.2.1. Smagorinsky–Lilly (SL) model

In this model, the eddy viscosity is defined as follows (Smagorinsky, 1963):

$$\mu_t = C_S^2 \Delta^2 \sqrt{2 \tilde{S}_{ij} \tilde{S}_{ij}},$$  

(7)

where $C_S$ (the Smagorinsky constant) is 0.1, the filter width $\Delta = \frac{\sqrt{V}}{L_1}$, and the overbar denotes time-averaged quantities. The corresponding SGS stress tensor is defined as

$$T_{ij} = -2 \mu_t \tilde{S}_{ij} + \frac{1}{3} T_{kk} \delta_{ij},$$  

(8)

where $\tilde{S}_{ij}$ is the resolved strain-rate tensor, which is given by

$$\tilde{S}_{ij} = \frac{1}{2} \left[ \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right].$$  

(9)

#### 2.2.2. Dynamic Smagorinsky–Lilly (DSL) model

In order to dynamically resolve the equations of motion, an extra second filter (or test filter) is employed in this model, where the new filter width $\tilde{\Delta}$ is twice the grid filter width $\Delta$. The coefficient $C_S$ is not specified by the user in advance, as it is dynamically calculated based on the resolved scales of motion (Germano, Piomelli, Moin, & Cabot, 1991). Thus, two scale filters generate the SGS stress tensors:

$$\begin{align*}
\tau_{ij} &= -2 C_D \rho \Delta^2 \tilde{S}_{ij} \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right) \\
T_{ij} &= -2 C_D \rho \tilde{\Delta}^2 \tilde{S}_{ij} \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right),
\end{align*}$$  

(10)

where $\wedge$ denotes terms that are filtered by the sub-test filter and $\sim$ denotes Favre-averaged quantities. The second-order tensor $L_{ij}$ is the stress produced by the turbulence movement between the main filter $\Delta$ and the second filter $\tilde{\Delta}$:

$$L_{ij} = T_{ij} - \hat{T}_{ij} = \tilde{u}_i \tilde{u}_j - \bar{u}_i \bar{u}_j.$$  

(11)

The model coefficient $C_D$ with the contraction can be obtained from the least-squares analysis of Lilly (1992) as follows:

$$C_D = \frac{(L_{ij} - L_{kk} \delta_{ij} / 3)^2 M_{ij}}{M_{ij} M_{ij}},$$  

(12)

where

$$M_{ij} = -2(\Delta^2 \rho \tilde{S}_{ij} \Delta^2 - \Delta^2 \hat{\rho} \tilde{\Delta}^2 \tilde{S}_{ij}).$$  

(13)

#### 2.2.3. Wall-adapting local eddy viscosity (WALE) model

In this formulation, the eddy viscosity is modeled as follows (Nicoud & Ducros, 1999):

$$\mu_t = \bar{\rho} L_s^2 \frac{(S_{ij} S_{ij}^d)^{3/2}}{(\tilde{S}_{ij} S_{ij})^{5/2} + (S_{ij} S_{ij}^d)^{5/4}},$$  

(14)

The length scale is defined as

$$L_s = \min \{ \kappa y, C_w V^{1/3} \},$$  

(15)

where $\kappa$ is the von Kármán constant, $y$ is the distance to the closest wall, $C_w = 0.325$ is the model coefficient, and $S_{ij}^d$ is the traceless symmetric part of the square of the velocity gradient tensor, which is given by

$$S_{ij}^d = \frac{1}{2} (\tilde{S}_{ij}^2 + \tilde{S}_{ji}^2) - \frac{1}{3} \delta_{ij} \tilde{S}_{kk}, \quad \tilde{S}_{ij} = \frac{\partial \tilde{u}_i}{\partial x_j}.$$  

(16)
\[
S_{ij}^{el} = \frac{1}{6} (S^2 S^2 + \Omega^2 \Omega^2) \\
+ \frac{2}{3} \Omega^2 \Omega_i \Omega_j, \quad \Omega^2 = \Omega_j \Omega_j, \quad IVS = S_{ij} S_{ij} S_{ij} S_{ij}.
\]

2.2.4. Algebraic wall-modeled LES (WMLES) model and WMLES S–Ω model

In order to overcome the Re scaling limitations of LES, the algebraic wall-modeled LES (WMLES) model is proposed (Shur, Spalart, Strelets, & Travin, 2008). Here, the eddy viscosity is calculated by using a hybrid length scale:

\[
\mu_t = \min\{(\kappa d_w)^2, (C_{Smag}^3)^2\} \cdot S, \quad \Delta = \min\{(\kappa d_w), (C_{Smag}^3)^2\} \cdot S,
\]

where \(d_w\) is the wall distance, \(S\) is the strain rate tensor, \(C_{Smag} = 0.2\) is a constant, and \(y^+\) is the normal to the wall inner scaling. The SGS model is based on a modified grid scale that accounts for the grid anisotropies in wall-modeled flows:

\[
\Delta = \min\{(C_w \cdot d_w), (C_w \cdot h_{max}), (h_{nw}), (h_{max})\}, \quad (20)
\]

where \(h_{max}\) is the maximum edge length for a rectilinear hexahedral cell, \(h_{nw}\) is the wall-normal grid spacing, and \(C_w = 0.15\). Considering the zero eddy viscosity, WMLES S–Ω model is devised (Piomelli, Moin, & Ferziger, 1988). In this model, Eq. (17) is calculated with \(abs(S–\Omega)\) instead of \(S\), where \(\Omega\) is the rotation rate tensor.

2.2.5. Dynamic kinetic energy transport (KET) model

By accounting for the transport of the SGS turbulence kinetic energy, the eddy viscosity is defined as follows (S. E. Kim, 2004):

\[
\mu_t = C_k \rho k_{sgs}^{12} \Delta, \quad (21)
\]

where \(k_{sgs}\) is the SGS kinetic energy calculated from (W. W. Kim & Menon, 1997):

\[
\rho \frac{\partial k_{sgs}}{\partial t} + \rho \frac{\partial \tilde{K}_{sgs}}{\partial x_j} = -\tau_{ij} \frac{\partial u_i}{\partial x_j} - C_k \rho k_{sgs}^{3/2} \Delta_f + \frac{\partial}{\partial x_j} \left( u_i \frac{\partial k_{sgs}}{\partial x_j} \right).
\]

Then the SGS stress tensor can be written as

\[
\tau_{ij} = \frac{2}{3} \rho k_{sgs} \rho_{ij} - 2C_k \rho \Delta \sqrt{k_{sgs} S_{ij}}.
\]

In this model, \(C_k\) and \(C_\varepsilon\) are calculated dynamically, and \(\sigma_k = 0.1\) is a constant.

2.2.6. Shear-Stress Transport (SST) k–ω model

The turbulence kinetic energy \(k\) equation and specific dissipation rate \(\omega\) equation are expressed as follows (Menter, Kuntz, & Langtry, 2003):

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k U_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \left( \mu + \sigma_k \mu_t \right) \frac{\partial k}{\partial x_j} \right) + P_k - \beta^* \rho k \omega, \quad (23)
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega U_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \left( \mu + \sigma_\varepsilon \mu_t \right) \frac{\partial \omega}{\partial x_j} \right) + 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, \quad (24)
\]

where the blending function \(F_1\) is defined by

\[
F_1 = \tanh \left\{ \left[ \min\left[ \sqrt{\kappa} \frac{500 v}{\beta^* \omega}, \frac{4 \rho \sigma_{\omega 2} k}{CD_{kw} y^2} \right] \right]^4 \right\},
\]

where

\[
CD_{kw} = \max \left( 2 \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right).
\]

At the boundary layer, the \(k–\varepsilon\) model is used away from the wall, \(F_1 = 0\), and the inner layer is switched to the \(k–\omega\) model. The turbulent eddy viscosity is defined by:

\[
\mu_t = \frac{a_1 k}{\max(\eta_1 w, SF_2)}, \quad (25)
\]

where \(S\) is the invariant measure of the strain rate and the second blending function \(F_2\) is defined by:

\[
F_2 = \tanh \left[ \left[ \max \left( 2 \sqrt{\kappa} \frac{500 v}{\beta^* \omega y^2}, \frac{CD_{kw} y^2}{\eta^2} \right) \right]^2 \right]. \quad (26)
\]

In the stagnation zone, the shear-stress transport (SST) \(k–\varepsilon\) model defines a production limiter to prevent turbulence generation, which is defined by:

\[
P_k = \mu_t \frac{\partial U_i}{\partial x_j} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_j} \right) \rightarrow P_k
\]

\[
= \min(P_k, 10 \beta^* \rho k \omega). \quad (27)
\]

The other setting parameters in the model are as follows:

\[
\sigma_k = 0.85, \quad \sigma_\varepsilon = 0.5, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 0.856, \quad \beta_1 = 3/40, \quad \beta_2 = 0.0828, \quad \beta^* = 0.09, \quad a_1 = 5/9, \quad a_2 = 0.44.
\]
2.2.7. Summary of the advanced properties of SGS models

In the SL model, $C_S$ is defined as a fixed value, which greatly simplifies the governing equations but results in a deviation from the actual situation. This is the reason why the prediction error is larger in the SL model than in the other SGS models. In the dynamic Smagorinsky–Lilly (DSL) model, $C_S$ is dynamically calculated according to the information acquired by the resolved scales of motion; hence, the prediction accuracy of the flow states is improved. The wall-adapting local eddy viscosity (WALE) model is based on the square of the velocity gradient tensor and considers the effects of local strain and rotation rates simultaneously; as a result, it obtains better eddy viscosity near the wall scale without using a dynamic model.

The WMLES model is a kind of hybrid RANS/LES method which employs the RANS method to treat the much finer near-wall region and then switches to the LES method once the grid spacing becomes sufficient to resolve the local scales. Since the WMLES model does not take into account zero eddy viscosity in the constant shear flow, instead of $S$, the WMLES $S-\Omega$ uses $\text{abs}(S-\Omega)$ to compute the LES portion, thus capturing the transition behaviors between the laminar and turbulent flows. Based on the DSL model, the KET model employs the resolved velocity scales to parameterize the SGS stresses, and it assumes that the dissipation of kinetic energy at small sub-grid scales is locally balanced with the transferred energy after filtration. Accordingly, it can simulate the turbulence motions more effectively.

3. Numerical method and simulation details

3.1. Test setup

A physical experiment was conducted to validate the effectiveness of the LES method, rather than employing flow visualization. The standard layout was manufactured according to the current Chinese national standard – GB/T 7680-2005 (2006), which was adopted in the flow machinery. A TC with a nominal diameter of 345 mm designed by the authors and fabricated by Guangxi Liugong Machinery Co., Ltd was chosen for this study (Figure 1). The detailed specifications of the pump, turbine, and stator are summarized in Table 1.

Figure 3 shows the basic layout of the test bench used in this study, comprised of the oil compensation system (Part I), the main test system (including the drive motor, sensors, dynamometer, etc.; Part II), and the control and data acquisition system (Part III). The operating conditions of the TC are determined through the joint work of the drive motor (4) and the dynamometer (7). Torque sensors (5) were used to quantitatively obtain the required torque under different speed ratio conditions. The speed of the input shaft was kept constant while the speed of the output shaft slowly rose to a specified level. When the measurements were taken, the temperature of the inlet oil was 80 to 100°C and the temperature of the outlet oil was a maximum of 120°C. The measurement of all parameters was carried out simultaneously after the condition had been stable for 5 s. The instruments used in the experiment were adapted to the required ranges of the measured parameters, and their precision is listed in Table 2.

Table 1. Specifications for the three elements.

| Element | Inlet angle | Outlet angle | Number of blades |
|---------|-------------|--------------|-----------------|
| Pump    | 108°        | 123°         | 28              |
| Turbine | 35°         | 150°         | 27              |
| Stator  | 96°         | 20°          | 17              |

Table 2. Precision of various sensors.

| Sensor type: | Torque | Speed | Pressure | Temperature | Flow |
|--------------|--------|-------|----------|-------------|------|
| Precision    | ±0.5%  | ±2.0% | ±1.5%    | ±1.5%       | ±0.5%|

Figure 3. Rig test for the TC.
The experimental performance was obtained by measuring the input $N_{pump}$ and the output $N_{turbine}$:

$$SR \text{ (speed ratio)} : SR = \frac{N_{turbine}}{N_{pump}}$$

$$TR \text{ (torque ratio)} : TR = \frac{T_{turbine}}{T_{pump}}$$

$$CF \text{ (capacity factor)} : CF = \frac{T_{pump} 	imes 10^6}{N_{pump}^2}$$

where $CF$ is the capacity of the power transmission, and the efficiency was obtained as

$$\eta \text{ (efficiency)} : \eta = \frac{N_{turbine} \times T_{turbine}}{N_{pump} \times T_{pump}} = SR \times TR$$

where $N$ is the rotation speed (rpm) and $T$ is the torque of the element.

### 3.2. Three-dimensional computational domain and mesh layout

For the fluid flow simulation, the object of analysis is the space through which the fluid passes. A full passage model was created in Unigraphics (UG) to extract the contours of the blade surface, the shell, and the core surface. To minimize the deviation of the mesh shape among components with the same shape, the mesh was modeled by extracting the minimum periodic unit. Because a high degree of accuracy is required for modeling the blade surfaces, a local-grid refinement method was applied to obtain sufficient resolution near the blades. Finally, an overall model was created by copying the periodic model. The procedure for creating an analytical model is shown in Figure 4(a), and a grid independence study was carried out as shown in Figure 4(b). In order to maximize the advantages of the LES method and express the flow structures in detail, 9 million grid cells were applied in this study. The maximum and minimum grid sizes are 3.000 mm (the maximum element of global size) and 0.025 mm (the first layer height near the blade), respectively.

### 3.3. Computational settings

Prior to calculation, appropriate simplifications were made to reduce the workload of mesh generation and computation time: (1) the density and viscosity of the transmission oil were set as constants, (2) the computational domain was defined as an enclosed region without an inlet or an outlet, and (3) the oil leakage between the impellers was ignored. Furthermore, the sliding mesh method of ANSYS Fluent (2011) was applied to deal with the real-time interaction between the upstream and downstream flows. Mesh interfaces were set up to enable the transmission of variables and parameters between the adjacent impellers. The internal flow field was full of fluid and all the components moved at the prescribed rotational velocity. The blade, shell, and core surfaces...
were defined as non-deformable solid walls and the boundaries used no-slip wall conditions, meaning that the fluid had zero velocity relative to the boundaries.

A PC with an Intel Xeon E5-2620 V3 2.4 GHz CPU and 32 GB of memory was employed for the calculations. In addition, the solution was assumed to have converged when the monitored turbine torque tended toward stability and the defined residuals for all equations were less than $1 \times 10^{-4}$ (Figure 5). Details of the other properties of the CFD simulation are presented in Table 3.

![Figure 5. Time histories of the monitored velocity and torque.](image)

| Analysis type                  | Transient state       |
|--------------------------------|-----------------------|
| Pressure–velocity coupling    | SIMPLEC               |
| Momentum                      | Bounded central differencing |
| Transient formulation         | Second-order implicit |
| Pump status                   | 2000 rpm              |
| Turbine status                | 0 \sim 1600 rpm       |
| Stator status                 | Stationary            |
| Viscosity                     | 0.0258 Pa·s           |
| Density                       | 860 kg/m³             |
| Timestep size                 | 0.0005 s              |
| Number of time steps          | 400                   |

![Figure 6. Energy spectra of the monitored velocity.](image)
3.4. Evaluation of temporal variations

For a comprehensive study of turbulence with LES, Figure 5 also shows the time history of the velocity monitored at a location in the stator region (87 mm, −55 mm, 5 mm). After 0.04 s, the flow data seems to fully develop into a random fluctuation of turbulence.
Therefore, the time-averaged evaluation of the torque forces of the three cascade components was performed within a fixed time range of 0.1 to 0.2 s.

The energy spectra obtained by monitoring the instantaneous velocity at this location using each SGS model are plotted in Figure 6. In the simulation, the velocity energy spectra are all appropriately characterized by the theoretical power law of Kolmogorov’s (1941) \(-5/3\) slope, showing that the LES models are well resolved in both the time and frequency domains. However, it is difficult to make specific comments on the superiority of any SGS model based only on the energy spectra.

### 4. Results and analysis

#### 4.1. Prediction estimations of the LES method and comparison with the RANS method

Figure 7 shows a comparison of the CFD results and the test data. It can be seen that the KET and WALE models generated the best predictions. The maximum error in CF is reduced to 4.4%, compared to values of 15% (Lee, Jang, Lee, & Lim, 2000), 9% (Jung, Kang, & Hur, 2011), and 10% (Lei et al., 2012) for the RANS method. These results demonstrate that the use of the LES method produces a significant improvement in performance estimation. A comparison of the computation times for the SGS models is listed in Table 4, which shows that the SL and KET models require the shortest and longest simulation times, respectively.

Although the RANS method has been widely used for the numerical simulation of TCs, there are still some limitations on how it captures the transient turbulence characteristics. As a comparison, the RANS method was applied in this investigation using an SST \(k-\omega\) model. The simulation procedure and convergence targets are the same as those used for the LES method. Figure 8(a) presents the whole streamlines between the impellers \((SR = 0)\). It can be seen that the flow states predicted by the KET model are highly complex and that the vortices are clearly captured, illustrating that the unsteady simulation of the LES method might be suitable for TC design.

![Figure 8. Streamlines between the impellers obtained by the LES and RANS methods: (a) whole streamlines; (b) local streamlines.](image)
processes. As expected, it is clear from the local streamlines shown in Figure 8(b) that the flow track between the three cascade components is well described, which also proves the validity of LES method.

The Reynolds number ($Re$) is an important dimensionless quantity in fluid mechanics that is used to evaluate flow patterns. It represents the ratio of the inertia force to the viscous force in the fluid:

$$Re = \frac{\rho v L}{\mu}, \quad (32)$$

where $v$ is the mean fluid velocity and $L$ is the characteristic length scale. Figure 9 shows the $Re$ distribution in the pump–turbine interface ($SR = 0$), illustrating that the internal flow of the TC is fully developed turbulence with high Reynolds numbers. When compared with the reference RANS model, it can be seen that the $Re$ results between the blades obtained by the KET model are more complex and constantly changing, which might be due to the properties of the unsteady simulation. The LES method was able to capture the transient flow phenomena, which is the key reason for the improvement in performance prediction, while the time-averaged method seems to be inaccurate and unreliable.

In addition, Tasaka et al. (2017) applied the LES method to predicting the overall performance of TCs, and the maximum prediction error of the torque forces was 4.7%. However, approximately 26 million grid cells were used in their study, which resulted in a very high requirement in terms of processing power. By comparison, similar and slightly better results were obtained in the current study using fewer grid cells, which might be related to the selection of the SGS models. These findings for LES simulations could help with better understanding the turbulent flows within TCs.

4.2. Flow field description

4.2.1. Pressure–streamline distribution

As external characteristics are determined by internal flow structures, this study also focuses on how – based on the chosen mesh scales – the specific numerical algorithms of various SGS models reflect performance differences. Figure 10(a) describes the pressure–streamline distribution on the suspended cylindrical surface of the stator ($R = 100$ mm). The pressure difference between the sides of the blade is the main reason for torque generation. Owing to the large incident angle at the stall condition ($SR = 0$), severe impacts occur on the inlet of the pressure side, leading to an increase in pressure and a decrease in speed. Meanwhile, a large-scale backflow region forms near the boundary layer of the suction side;
as the flow leaves the blade surface, more unsteady flow phenomena – such as vortex flow, flow separation, and secondary flow – can be observed. It was found that the WALE, WMLES S–Ω, and KET models capture the flow separation and vortex flow near the suction side relatively clearly, whereas the flow separation zone predicted by the WMLES S–Ω lags slightly behind the predictions of the other two models.

4.2.2. Vorticity distribution

Vorticity is conceptually defined as the curl of a velocity field that describes the direction and magnitude of a vortex. Due to the direct action of the upstream impeller, high-vorticity regions always appear at the surface of the near wall on the suction side, accompanied by the phenomena of flow separation and reattachment. Figure 10(b) shows the transient variations of vorticity magnitude in the stator (SR = 0). It can be seen that the numerical magnitudes obtained by the WMLES S–Ω model are largest, followed by the KET model. Furthermore, the wake region in the outlet predicted by the KET model is more pronounced, while the results of the others are indistinguishable. Overall, the WMLES, WMLES S–Ω, and KET models show advantages in analyzing the vorticity distribution. The differences in the flow structures between each blade passage also illustrate that unsteady simulations are more accurate for predicting the complicated turbulent reaction mechanisms than the time-averaged method.

4.2.3. Velocity distribution

A comparative analysis of the velocity distribution in the stator (SR = 0) is shown in Figure 10(c). It can be seen that the high-speed transmission oil flowing out of the turbine causes rapid deceleration due to a severe impact interaction with the blade surface on the pressure side. The flow rapidly accelerates as it passes the blade’s leading edge, producing flow separation and backflow. Subsequently, the transmission oil accelerates on the pressure side while decelerating on the suction side. Although the trends predicted by each SGS model are consistent, it is clear that the ability of the WMLES model to capture the low-speed region is unsatisfactory compared with the other models. In the WMLES S–Ω model, the low-speed region of the wake flow is not accurately predicted, while the analog values of the SL model are much too small. Figure 11 shows the variation of the velocity distribution for radial distances ranging from R = 155 mm to R = 170 mm in the pump–turbine interface (SR = 0). The speed fluctuation curve achieved by the SL model is relatively smooth, which illustrates that the simplified governing equations lead to a more uniform prediction of the flow field. The WALE and KET models obtained larger speed ranges, proving that the flow field information is simulated in detail.

4.2.4. Vortex distribution

Coherent structures in a turbulent flow can be visualized using the Q criterion proposed by Dubief and Delcayre
Figure 11. Variation of the velocity distribution in the interface.

(2000). Figure 12 shows the coherent structures for the SGS models near the turbine blades under $Q = 3 \times 10^6$. It can be seen that the flow speed gradually declines along the main stream. The comparison shows that the DSL, WALE, and KET models are most effective at capturing the vortex structures, whereas the description generated by the SL model is quite rudimentary.

4.3. Flow mechanism analysis

4.3.1. Sub-grid effective viscosity

It was necessary to analyze the turbulent viscosity ($\mu_t$) because of the different definitions of the eddy viscosity coefficient used in each SGS model. Figure 13 presents the apparent differences in the sub-grid effective viscosity between the near-wall region and the main-stream region. Through comprehensive comparison, it is shown that the predicted values of the WMLES and WMLES S–$\Omega$ models are higher compared to those of the other SGS models, resulting in greater energy loss and reduced prediction accuracy. However, the smaller values simulated by the SL model demonstrate greater deviation from the actual situation. Similar results are also found in the pump–turbine interface, as shown in Figure 14 ($SR = 0$).

4.3.2. Near-wall treatment

The near-wall treatment used in the LES method leads to highly accurate predictions, but it requires a very fine grid resolution in the boundary layer. Moreover, the magnitude of the wall shear stress ($\tau_w$) distribution can to a certain extent reflect energy dissipation in the flow process. For the LES method, it can be obtained from the laminar stress–strain relationship if the laminar sublayer is well resolved under a mesh with sufficiently fine resolution:

$$\begin{align*}
    u &= \rho \frac{u_t y}{\mu}, \\
    \frac{u_t}{\mu} &= \sqrt{\frac{\tau_w}{\rho}}.
\end{align*}$$

(33)

Alternatively, when the mesh is too coarse to resolve the laminar sublayer it is assumed that the centroid of the wall-adjacent cell is located in the logarithmic region, and the law of the wall is used:

$$\frac{u}{u_t} = \frac{1}{\kappa} \ln \left( \frac{\rho u_t y}{\mu} \right),$$

(34)

where $y$ is the distance to the wall, $\kappa$ is the von Kármán constant, and $E = 9.793$.

Figure 12. Vortex distribution near the turbine blades.
Figure 13. Sub-grid effective viscosity distribution in the stator region.

Figure 14. Variation of the sub-grid effective viscosity in the interface.

Figure 15 displays the wall shear stress distribution on the blades of the three impellers ($SR = 0$). The viscous effect between the transmission oil and the blade surface generates wall shear stress. Generally, more wall shear stress occurs in the inlets of the stator and the turbine, and less occurs on the pressure side of the pump blade surface. However, the SL model predicts significantly greater levels of wall shear stress compared to the other models, indicating relatively high levels of energy dissipation. This illustrates that the WALE and KET models predict
Figure 15. The wall shear stress on the blades of the pump, the stator, and the turbine.

Figure 16. The $y^+$ distribution on the stator blade surface.
lower levels of energy dissipation, and thus are predicting the performance with better accuracy.

Figure 16 shows the ability of the SGS models to accurately analyze the boundary layer \((SR = 0)\). It can be seen that all values of \(y^+\) on the stator blade surface are less than 2, indicating that the grid resolution satisfies the calculation requirements of the near-wall regions (Gharbi, Absi, Benzaoui, & Amara, 2010) and that the viscous sub-layer was resolved instead of using wall functions. This is one of the main reasons that the LES method can generate high-accuracy performance predictions under the chosen grid resolution. In addition, the KET model generated the smallest values and the SL model generated the largest values, which is also in accordance with the prediction accuracy of the experimental data.

It is of note that the predictions of the WMLES and WMLES-S-\(\Omega\) models are often unsatisfactory. This is because they use hybrid RANS–LES methods; hence, when the computational domain of the RANS method extends beyond the viscous sublayer, less of the turbulence spectrum is resolved than in the other SGS models, resulting in larger calculation errors. If the grid resolution, especially in the near-wall region, is sufficiently fine to demonstrate the advantages of these two SGS models, this results in a huge increase in the computational cost.

## 5. Conclusion

Transient CFD simulations were implemented to capture the internal flow field of the TC, and a comprehensive assessment of a number of SGS models was carried out. In order to ensure the calculation accuracy, a full three-dimensional passage model with block-structured grids was used to appropriately characterize the complex geometry of the TC. The following conclusions are drawn from this study:

1. The LES method can predict the variation tendencies in TC performance at different speed ratios. The results show that the absolute errors in the performance estimations are less than 4.4%, which proves that the LES method is more accurate than the RANS method.
2. For transient simulations, the LES method can capture the essential flow information. However, the SL model is not suited to capturing the dynamic flow process due to a constant coefficient that simplifies the governing equations.
3. Using the LES method with coarse grids can reduce the computing requirements and contribute to the TC design process through a reasonable selection of SGS models. When both experimental performance and internal flow structures are regarded as the research targets, the KET model is recommended. Given its computational efficiency and accuracy, the WALE model is also a good alternative.
4. The grid resolution of the computational domain is an important factor. The WMLES and WMLES-S-\(\Omega\) models require finer grids, especially in the near-wall region. Moreover, these two models calculate larger sub-grid effective viscosity than the other SGS models, which results in more energy dissipation and lower prediction accuracy.

Further work on the requirements of CFD tools is needed in order to fully evaluate TC performance, including the numerical investigation of the effects of the cavitation mechanism on overall performance, and multi-objective shape optimization for the geometric parameters of the blade cascade to improve torque capacity and transmission efficiency.

### Disclosure statement

No potential conflict of interest was reported by the authors.

### Funding

This work was supported by Key Scientific and Technological Projects of Jilin Province [grant no. 20170204066GX]; Science and Technology Projects of the Education Department of Jilin Province [grant no. JJKH20170785K]; the Open Fund of Key Laboratory of Road Construction Technology and Equipment of Chang’an University, Ministry of Education [grant no. 310825171104]; Green Design Platform Construction Project for High-End Earthwork Machinery, Ministry of Industry and Information [grant no. [2017]327]; and the Advanced Manufacturing Projects of Government and University Co-construction Program funded by Jilin Province [grant no. SXGJSF2017-2].

### ORCID

Wenxing Ma [http://orcid.org/0000-0002-2865-8643](http://orcid.org/0000-0002-2865-8643)

### References

By, R. R., Kunz, R., & Lakshminarayana, B. (1995). Navier–Stokes analysis of the pump flow field of an automotive torque converter. *Journal of Fluids Engineering, 117*(1), 116–122.

Cigarini, M., & Jonnavithula, S. (1995). Fluid flow in an automotive torque converter: Comparison of numerical results with measurements (SAE Technical Paper No. 950673).

Dong, Y., Korivi, V., Attibele, P., & Yuan, Y. (2002). Torque converter CFD engineering – Part I: Torque ratio and K factor improvement through stator modifications (SAE Technical Paper No. 2002-01-0883).

Dubief, Y., & Delcayre, F. (2000). On coherent-vortex identification in turbulence. *Journal of Turbulence, 1*(1), N11.

Fluent Inc. (2011). *Fluent user’s guide*. Lebanon, NH: Fluent Incorporated.
GB/T 7680-2005. (2006). Torque converters-Methods of performance tests. (in Chinese).

Germano, M., Piomelli, U., Moin, P., & Cabot, W. H. (1991). A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics, 3*(7), 1760–1765.

Gharbi, N. E., Absi, R., Benzaoui, A., & Amara, E. H. (2010). Effect of near-wall treatments on airflow simulations. *Journal of Applied Fluid Mechanics, 5*(14), 63–70.

Jung, J. H., Kang, S., & Hur, N. (2011). A numerical study of a torque converter with various methods for the accuracy improvement of performance prediction. *Progress in Computational Fluid Dynamics, An International Journal, 11*(3–4), 261–268.

Kim, S. E. (2004). Large eddy simulation using unstructured meshes and dynamic subgrid-scale turbulence models (AIAA paper No. 2004-2548).

Kim, B. S., Ha, S. B., Lim, W. S., & Cha, S. W. (2008). Performance estimation model of a torque converter part I: Correlation between the internal flow field and energy loss coefficient. *International Journal of Automotive Technology, 9*(2), 141–148.

Kim, W. W., & Menon, S. (1997, January). Application of the localized dynamic subgrid-scale model to turbulent wall-bounded flows. *Proceedings of the 35th aerospace sciences meeting and exhibit* (pp. 210). Reno, NV.

Kolmogorov, A. N. (1941). The local structure of turbulence in incompressible viscous fluid for very large reynolds numbers. *Doklady Akademii Nauk SSSR, 30*(4), 299–303.

Lee, C., Jang, W., Lee, J. M., & Lim, W. S. (2000). Three dimensional flow field simulation to estimate performance of a torque converter (SAE Technical Paper No. 2000-01-1146).

Lei, Y. L., Wang, C., Liu, Z. J., & Li, X. Z. (2012). Analysis of the full flow field of torque converter. *Advanced Materials Research, 468–471*, 674–677.

Lilly, D. K. (1992). A proposed modification of the germano subgrid-scale closure method. *Physics of Fluids A: Fluid Dynamics, 4*(3), 633–635.

Liu, C., Liu, C., & Ma, W. (2015). RANS, detached eddy simulation and large eddy simulation of internal torque converters flows: A comparative study. *Engineering Applications of Computational Fluid Mechanics, 9*(1), 114–125.

Liu, C., & Ma, W. (2010). 3D transient calculation of internal flow field for hydrodynamic torque converter. *Journal of Mechanical Engineering, 46*(14), 161–166. (in Chinese).

Menter, F. R., Kuntz, M., & Langtry, R. (2003). Ten years of industrial experience with the SST turbulence model. *Turbulence, Heat and Mass Transfer, 4*(1), 625–632.

Moeng, C. H., & Sullivan, P. P. (2002). Large-eddy simulation. In R. N. Gerald, P. John, & F. Q. Zhang (Eds.), *Encyclopedia of atmospheric sciences* (2nd ed., Vol. 4, pp. 232–240). Cambridge, MA: Academic Press.

Nicoud, F., & Ducros, F. (1999). Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion, 62*(3), 183–200.

Piomelli, U., Moin, P., & Ferziger, J. H. (1988). Model consistency in large eddy simulation of turbulent channel flows. *Physics of Fluids, 31*(7), 1884–1891.

Shin, S., Chang, H., & Athavale, M. (1999). Numerical investigation of the pump flow in an automotive torque converter (SAE Technical Paper No. 1999-01-1056).

Shin, S., Lee, B. C., Hong, J. H., & Joo, I. S. (2003). Performance improvement using a slotted stator of an automotive torque converter (SAE Technical Paper No. 2003-01-0247).

Shur, M. L., Spalart, P. R., Strelets, M. K., & Travin, A. K. (2008). A hybrid RANS–LES approach with delayed-DES and wall-modelled LES capabilities. *International Journal of Heat and Fluid Flow, 29*(6), 1638–1649.

Smagorinsky, J. (1963). General circulation experiments with the primitive equations: I. The basic experiment. *Monthly Weather Review, 91*(3), 99–164.

Tasaka, T., Oshima, N., Fujimoto, S., & Kishi, Y. (2017). Application of Large Eddy Simulation to a Torque Converter to Predict its Fluid Performance (SAE Technical Paper No. 2017-01-1116).