Rans, detached Eddy simulation and large Eddy simulation of internal Torque converters flows: A comparative study

Chunbapo Liu, Changsuo Liu and Wenxing Ma*

School of Mechanical Science and Engineering, Jilin University, Changchun13002, China

(Received 10 July 2014; final version received 27 October 2014)

Comparative analysis among the capabilities of the RANS, DES, and LES models to predict flow and turbulence distribution was conducted to come up with guidelines for hydraulic torque converter (TC) transient simulation. To ensure the accuracy of calculation, the complex geometry of hydrodynamic elements was accurately represented and the computational meshes of the structured hexahedron were appropriately distributed. Wall shear stress, pressure–streamline structure were analyzed. Compared with RANS, the transient vorticity features, including the birth, development, formation of a scroll; transportation along the blade surface; shedding and rupture at the trailing edge could be clearly captured by the LES and DES models. Rothalpy was used to quantitatively evaluate the hydraulic loss and a new computational formula was proposed to predict the efficiency of each element in TC. After the comparison of relative computing time, DES model was proved be a feasible method for efficiently and accurately simulating 3D unsteady turbulent flow of TC.

Keywords: hydraulic torque converter; transient simulation; DES model

1. Introduction

A hydraulic torque converter (TC) is a turbomachine that is widely used in automatic transmissions for automobiles. It consists of three major elements, namely, a pump, a turbine, and a stator. The functions of a TC include damping engine torque fluctuation, damping noise and vibration in the drive line, and automatically amplifying torque according to the difference in rotational speed between the input and output shafts without requiring external control (Kim, Ha, Lim, & Cha, 2008).

Various intrinsic flow motions, such as swirl, flow separation, and secondary flow, occur inside a TC. The accuracy of performance prediction heavily depends on the capability of the TC to capture and represent flow patterns and turbulence. The k-epsilon model, which is based on the average Reynolds number, is the most commonly used method to conduct numerical TC simulations. In recent years, however, more precise methods, including detached eddy simulation (DES) and large eddy simulation (LES), have been used to study the flow mechanism inside fluid machinery. Accordingly, these approaches have become the most effective tools for exploring the flow mechanism in a complex passages (Johnsen, Larsson, & Bhagatwala, 2010). LES captures cycle-to-cycle variation as opposed to the cycle-averaged mean computations of the Reynolds-averaged Navier–Stokes (RANS) equation, and therefore, is more accurate than the latter (Schmidt & Thiele, 2002). Wu, Wen, Yen, Weng, and Wang (2004) analyzed the transient separation process in the boundary layer at low Reynolds numbers. Su, Li, Li, and Wei (2012) made an assessment of the performance of the large eddy simulation (LES) method in simulating the complex vortical flow in a Francis hydraulic turbine. In the last decade, due to the intensive growth of computer capacity and parallelized numerical modeling solutions, the development and application of large eddy simulation (LES) became a convenient tool for the calculation of flow patterns around single and multiple cylinders, which result in very complex unsteady, three-dimensional flows that are dominated by large-scale coherent vortices (Palau-Salvador, Stoesser, Froehlich, Kappler, & Rodi, 2010; Stoesser, Kim, & Diplas, 2010). Matsuura and Kato (2006, 2007) performed a series of investigations on compressible transition flow in the low-pressure turbo cascade, T106. They summarized the changes in transient characteristic, including the propagation of pressure waves and flow separation, after considering and neglecting the effects of turbulent intensity.

Since its inception in 1997, the DES model, whose main idea is to solve the RANS equations in near-wall regions while applying the LES method in other areas, has been gradually applied in the field of aviation and fluid machinery (Squires, Krishnan, & Forsythe, 2008). This model is suitable for simulating separated flow under any Reynolds number. Caruelle and Durcos (2003) clarified the application of the DES model at the boundary layer. Xu, Chen, and Lu (2007) simulated flow separation with

*Corresponding author. Email: mawx@jlu.edu.cn

© 2015 The Author(s). Published by Taylor & Francis. This is an Open Access article distributed under the terms of the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.
the large-eddy and detached-eddy models at the boundary layer of a cylinder and captured the transient characteristic of flow separation. So far, several numerical attempts have been made to investigate the flow field inside the torque converter passages. Most previous studies, however, adopted a simplified geometry such as the one-passage model and ignored temporal variation of the flow field such as the unsteady blade interaction, which characterizes the actual flow pattern inside the passage of a torque converter. (Flack, 2005) simulated unsteady flow with a moving grid and a complete TC model. They observed that unsteady effects are most important in the region wherein the interaction of the pump and the turbine occurs. This interaction is a flow phenomenon that cannot be predicted with steady calculation. Jung, Kang, and Hur (2011) performed a comparative study to investigate the effects of different methods for handling the relative motion of parts. The accuracy of the turbulence model applied in performance prediction of Jung et al., who also failed to investigate the accuracy of the turbulence model was not considered in the study for handling the relative motion of parts. The accuracy of the turbulence model applied in performance prediction and turbulence distribution in the boundary layer.

In the present study, the complete geometry of a TC passage was meshed into a hexahedron via the high grid resolution in near-wall regions. Because the blades were distorted seriously, much effort and time were used to improve the quality of grid in order to ensure the accuracy of calculation. Through the comparison with experimental results, influence of various turbulence models on predicting TC performance was evaluated. Wall shear stress, pressure-streamline structure, vorticity and velocity distribution were also analyzed. Compared with RANS, the LES and DES models can capture transient vorticity features, including the birth, development, and formation of a scroll; transportation along the blade surface; and shedding and rupture at the trailing edge. Moreover, DES model was proven to be a feasible method for accurately and efficiently simulating 3D unsteady turbulent flow in the complex passage of a TC.

2. Numerical simulation

2.1. Three kinds of turbulence models

The Reynolds-averaged continuity and momentum equations for incompressible flow are as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + \frac{1}{3} \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}
\]

where the Reynolds stress \( \tau_{ij} \) is:

\[
\tau_{ij} = -\rho \left(u_i u_j \right)
\] \tag{3}

LES is a popular technique for simulating turbulent flows. An implication theory of self-similarity states that large flow eddies depend on geometry, whereas smaller ones are more universal. This feature allows explicitly solving for large eddies during calculation and implicitly accounting for small eddies by using a subgrid-scale model (Yang, Gupta, Kuo, & Gopalakrishnan, 2014), the subgrid-scale Reynolds stress is

\[
\tau_{ij} = -2\nu_f S_{ij} + \frac{1}{3} \tau_{kk} \delta_{ij}
\] \tag{4}

where the deformation rate tensor \( S_{ij} \) is:

\[
S_{ij} = \frac{1}{2} \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\] \tag{5}

and eddy viscosity coefficient \( \nu_f \) is

\[
\nu_f = (C_s \Delta)^2 |S|
\] \tag{6}

where the magnitude of deformation rate tensor, \( |S| = (2S_{ij} S_{ij})^{1/2} \), \( \Delta \) is the filtering scale, \( C_s \) is Smagorinsky coefficient.

Difficulties associated with using the standard LES model, particularly in near-wall regions, led to the development of the DES model, which attempts to combine the best aspects of RANS and LES methodologies in a single solution strategy. In this study, the Realizable k-epsilon model was used in near-wall regions, whereas the Smagorinsky–Lilly subgrid-scale model in LES (Breuer, Jovičić, & Mazaev, 2003) was used in separated flow regions.

The length size of eddy in turbulence is:

\[
L_{DES} = k^{3/2}/\varepsilon
\] \tag{7}

It is used for replacing the item \( \varepsilon \) in equation of turbulent kinetic energy in Realizable k-epsilon model, then, equation of turbulent kinetic energy in DES model is:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G_k - \rho \frac{k^{3/2}}{L_{DES}}
\] \tag{8}

The renormalization group (RNG) model was developed by using RNG methods to renormalize the N–S equations to account for the effects of small scales of motion. In the standard k-epsilon model, eddy viscosity is determined from a single turbulence length scale. Hence, the calculated turbulent diffusion refers to diffusion that occurs only at a specified scale. By contrast, all scales of motion contribute to turbulent diffusion in reality. Similar to k-epsilon, the RNG approach is a mathematical technique that can be used to derive a turbulence model. This approach results in a modified form of the epsilon equation, which attempts to account for different scales of motion through changes in the production term (Galvan, Reggio, & Guibault, 2011). Equation of (RNG) \( k-\varepsilon \) is as followed:
Equation of turbulent kinetic energy:
\[
\rho \frac{\partial k}{\partial \tau} + \rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j}
\]
\[+ \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i} \right) - \rho \varepsilon \tag{9}
\]

Equation of turbulent dissipation rate:
\[
\rho \frac{\partial \varepsilon}{\partial \tau} + \rho u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j}
\]
\[+ \frac{c_1 \varepsilon}{k} \mu_i \frac{\partial \mu_i}{\partial x_k} \left( \frac{\partial u_i}{\partial x_k} + \frac{\partial u_i}{\partial x_i} \right) - c_2 \rho \frac{\varepsilon^2}{k} \tag{10}
\]

where the turbulent viscosity coefficient \(\mu_t\) is:
\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \tag{11}
\]

the fluctuation kinetic energy dissipation rate of per unit in turbulence \(\varepsilon\) is:
\[
\varepsilon = \nu \left( \frac{\partial \mu'_i}{\partial x_k} \right) \frac{\partial \mu'_i}{\partial x_k} \tag{12}
\]

The following values are used for RNG \(k-\varepsilon\) model:
\[C_{\mu} = 0.09, \ C_1 = 1.44, \ C_2 = 1.91, \ C_K = 1.00, \ \text{and} \ C_\varepsilon = 1.30\]

The enhanced wall function method in the RANS model that employs the double zone model to divide turbulent flow into the viscous effect region and the fully turbulent flow region was adopted to ensure that the viscosity effect in the boundary layer can be reflected. Herein, the high Reynolds number RNG \(k-\varepsilon\) model was used to solve equations. However, the enhanced wall function method increases the requirement for mesh density in near-wall regions for grid encryption and smooth growth. In this study, the boundary layer had an initial size of 0.05 mm, a growth ratio of 1.1, and a maximum size of 0.60 mm.

2.2. Hexahedral mesh generation and mesh test

The complex geometry of hydrodynamic elements was accurately represented and the computational meshes were appropriately distributed with ANSYS ICEM computational fluid dynamics (CFD) software to generate the structured hexahedral grid of the entire TC. Mesh-point clustering was performed near the boundary layer, and cell deformity was checked with the software. Figure 1 describes the generation process of the structured grid of entire passage. A mesh sensitivity test in the RNG \(k-\varepsilon\) model was performed to minimize the effect of number error. The number of grids is 720,866, 1,530,270, 2,378,562 and 3,032,834 respectively. Figure 2 shows the grid distribution of 2.3 million.

The performance of a TC is defined by several operating parameters that are described as follows:

\[
SR \text{ (speed ratio)} : \quad SR = \frac{N_{\text{turbine}}}{N_{\text{pump}}}, \tag{13}
\]
\[
TR \text{ (torque ratio between the pump and turbine)}:
\[
TR = \frac{T_{\text{turbine}}}{T_{\text{pump}}}, \tag{14}
\]
\[
CF \text{ (capacity factor)} : \quad CF = \frac{T_{\text{pump}} \times 10^6}{N_{\text{pump}}^2}, \tag{15}
\]
\[
\eta \text{ (efficiency)} : \quad \eta = \frac{N_{\text{turbine}} \times T_{\text{turbine}}}{N_{\text{pump}} \times T_{\text{pump}}} = SR \times TR, \tag{16}
\]

where \(N\) is the rotating speed (rpm) and \(T\) is the torque of each component. To compare prediction accuracy under different grid densities, the absolute error of prediction

![Figure 1. Generation process of the structured grid: (a) entire passage, (b) single passage, (c) single structured grid model and (d) entire passage of the structured hexahedral.](image-url)
among different turbulence models is defined as follows:

\[
E_I = \frac{|I_{TM} - I_{Exp}|}{I_{Exp}} \times 100\%, \quad (17)
\]

where \( I \) represents the capacity coefficient, torque ratio, efficiency, and other physical quantities; and the subscripts \( TM \) and \( Exp \) denote the turbulence model and test data, respectively.

The YJ315X manufactured in Hangzhou Advance Gearbox Group Co., Ltd. is used in this paper, which had been tested in the Supervise and Inspect Center of Hydraulic Parts in China National Machinery Industry Corporation in August 2011. Figure 3 illustrates that the simulated result moved closer to the experimental data with increasing element numbers. Only a subtle change was observed when element numbers increased from 2.3 million to 3.0 million. After an evaluation of computation amount and accuracy, 2.3 million was used in subsequent CFD simulations of three turbulent models.

### 2.3. Computational setup

ANSYS FLUENT software was employed to solve the incompressible Navier–Stokes equations without heat transfer. During calculation, the pump and the turbine rotate along the grid interface according to a certain time step size (0.0005s). The solution was assumed to have converged when the rate of change in the mass flow rate in two consecutive circulatory iterations and all normalized residuals in mass and momentum conservation equations were less than \( 10^{-4} \) for the differences of results in these criteria is not obvious (Wood, Yan, & Wei, 2013), the total time
3. Results and analysis

3.1. Verification of the wall function model

The value of y-plus should be discussed through the post-process to investigate whether the grid satisfies the requirements of the enhanced wall function method or the mesh in near-wall regions. Figure 4 shows that all values of y-plus on the blades are less than 2. This result indicates that the mesh density in near-wall regions is acceptable (Barony, Olsen, Stoesser, & Sturm, 2012).

3.2. Wall shear stress on blade surfaces

The pressure surface of the pump blade was regarded as the positive face of the blade group (a single pump blade, turbine blade, and stator blade). Figure 5 indicates the wall shear stress distribution in the blade group.

Viscous effects facilitate the emergence of wall shear stress between the surface of the blades and the fluid. This stress attaches the fluid in near-wall regions to the surface of the blades, and thus, energy is consumed. The higher wall shear stress is located in the entrance region of the stator and the pump, whereas the lower one is located on the pressure surface of the turbine. The distributions of the RNG k-epsilon [Figure 5(c)] and DES models [Figure 5(b)] are similar, but significantly different from that of the LES model [Figure 5(a)]. This finding indicates that energy consumption is relatively larger in LES model than in other two models. The LES approach is always used to calculate the flow region at high Reynolds numbers. An extremely fine boundary layer mesh and a huge computational expense are required when the LES approach is adopted in near-wall regions at low Reynolds numbers. Therefore, the RANS model with enhanced wall function method or the DES model is recommended to simulate the near-wall region in most cases.

3.3. Pressure–streamline structure

Under stall condition (SR = 0), fluid flow from the turbine affects the leading edge of stator wherein the region with velocity stagnation and high vorticity is located. A part of fluid moves along the suction surface of the stator. Under the viscous force and inverse interaction of the pressure gradient, the velocity of the fluid particle within the boundary layer gradually decreases, whereas the fluid particle...
remains stagnant. Subsequently, the separation point is formed. The downstream fluid particle at the boundary layer is reversed to form the vortex flow, which pushes the upper fluid out of the surface and simultaneously causes flow separation, that is, the two-flow phenomenon in a wide range of downstream area. Figure 6 describes the distributions of pressure and streamline of fluid flow under four kinds of operating conditions, which are stuck in the transient time of 0.1 s. the direction in which the liquid flow from turbine blades impacts the stator leading edge deflects clockwise with the increase of SR. Meanwhile, the pressure gradient is evidently becoming gentle. A slight difference in the pressure gradient was observed among RANS, LES, and DES models only when SR = 0 (the stall condition). Overall flow, pressure trends, and flow separation region described in the three turbulent models are basically the same, which suggests that all simulations perform efficiently in capturing the main flow structures.

3.4. Vorticity distribution

The vorticity (1/s) is a description of the physical quantity of vortex motion, which is defined as the curl of a velocity field, and used to measure the magnitude and direction of vortex. In an enclosed turbo-machinery, the cyclical phenomenon like flow separation and reattachment always occurs in boundary layer of the suction surface due to the effect of upstream impeller. Figures 7 and 8 depict the transient evolutionary process of vorticity (1/s) on the suspended cylindrical surface in the stator and the interaction between the pump and the turbine under the stall condition (SR = 0) at different transient time respectively. In particular, Figure 8 shows that the outflow with high velocity in circular motion from the pump affects the leading edge of the turbine blades, which results in the flow separation phenomenon on the suction surface of the turbine blades.

In Figures 7 and 8, the LES and DES models clearly display transient vortex features, including the birth development, and formation of the scroll; transportation along the suction surface; and shedding and rupture at the trailing edge. LES and DES full geometry capture the local peaks and troughs in contours. Whereas RANS contours miss out on that and give out the relatively smooth pictures, the features exhibited during the evolution of vortices in the transient simulation in the RANS model may or may not be entirely accurate. Hence, RANS fails to capture this important trends.
3.5. Velocity distribution

The observations in Figure 9 suggest that all simulations perform well in predicting the magnitude and levels of velocity, however, only LES and DES models capture the low-velocity zones in the downstream region of suction surface which is reasonable owing to the shedding and rupture of vortex features at the trailing edge expressed in Figure 7. In Figure 10, three kinds of turbulent models differ relatively small in velocity distribution, however, after the scrutiny of low-velocity structures, it indicates the impetus to separate from the suction surface in LES and DES models while the stability to stay still in RANS models respectively. These results further reveal that LES and DES models are better able to predict the transient flow structures as compared to the RANS model.

Rothalpy was introduced to quantitatively evaluate loss and conduct the direct analysis on flow. For viscous flow, the difference of the parameter between two points on a streamline represents hydraulic loss, Rothalpy is defined as follows:

\[ R = \frac{P}{\rho} + \frac{1}{2} \frac{\omega^2}{\rho} - \frac{1}{2} \frac{\mu^2}{\rho}, \]

where \( P \) is the pressure, \( \rho \) is the density, \( \omega \) is the relative velocity, and \( \mu \) is the circumferential velocity.

Figure 11 displays the change in Rothalpy from the inlet to the outlet on the suction surface of the stator under stall condition (SR = 0). “0” and “1” represent the location of inlet and the outlet of stator respectively. Considering the retarded effect of fluid along the surface of the blades, Rothalpy value was gradually reduced. However, Rothalpy fluctuated because of the continuous evolution of the vortex on the suction surface under instantaneous time. The region between 0 and 0.7 was considered as the flow separation region wherein the kinetic energy of the fluid is low. Meanwhile, the position 0.7 was the reattachment point of the flow. Where after the Rothalpy increases significantly because of fluid flow along the surface. Losing the obvious characteristics of the vortex in the region of the outlet,
Figure 8. Vorticity distribution during the interaction between the pump and the turbine.

Figure 9. 2D contours of velocity magnitude at $Y = 0.09$ m plane ($SR = 0, T = 0.075s$)
Figure 10. 2D contours of velocity magnitude at $Y = 0.140$ m plane (SR = 0, T = 0.075s).

Figure 11. The change of Routhalpy from the inlet to the outlet on the suction surface of the stator as exhibited in Figure 7(c), resulted in a significant difference between the value of the RANS model and those of the DES and LES models.

3.6. Efficiency of elements

When operating, the performance of a TC can be predicted through experiments. However, the efficiency of each element can only be evaluated by performing a numerical calculation. The DES model was used as the example to predict the efficiency of each element.

The individual efficiencies of the pump, turbine, and stator ($\eta_p$, $\eta_T$, and $\eta_S$, respectively) and the overall efficiency $\eta_{TC}$ are defined by the following expressions:

$$\eta_p = \frac{(p_{p2} - p_{p1})}{\rho w_1(r_{p2}V_{p2} - r_{p1}V_{p1})},$$

$$\eta_T = \frac{\rho w_2(r_{t2}V_{t2} - r_{t1}V_{t1})}{(P_{t1} - P_{t2})},$$

$$\eta_S = 1 - \frac{(P_{s1} - P_{s2})}{(P_{s2} - P_{s1})}$$

$$\eta_{TC} = \eta_p \eta_T \eta_S,$$

where subscripts 1 and 2 refer to the inlet and outlet of each element, respectively; $P$ is the total pressure; $r$ is the radius; and $V$ is the velocity. Figure 12 displays the efficiency of various elements under different speed ratios.
ratios. The highest efficiency of a single element is approximately 93%, whereas TC efficiency is 81.2%. The overall efficiency $\eta_{TC}$ undoubtedly coincides well with the experimental data. Thus, this method can be used to assess the efficiency of individual elements in a TC, which can clearly indicates the reasons for inefficiency.

3.7. Prediction performance of the three turbulence models

As shown in Figure 13(a), the capacity factor of the LES and RNG k-epsilon models are relatively large, whereas that of the DES model is relatively small. However, both DES and LES models are more accurate in predicting CF than the RANS model, whose average absolute error is 2%. In predicting torque ratio, the absolute errors of the three models are high when SR is between 0 and 0.4, particularly under stall condition. The absolute errors of the RANS, LES, and DES models are 7.96%, 6.78%, and 6.12%, respectively. When SR is between 0.4 and 0.9, the absolute errors of the LES and DES models are less than 2%. By contrast, the data of the RANS model remain large although less than 5% [Figure 13(b)]. Only a slight difference exists between the highest efficiency data of the three
models and the experiment data. [Figure 13(c)] Overall, the original characteristics of three turbulence models are basically the same, which can satisfy engineering requirements. Compared with the absolute error of RNG k-epsilon model, those of DES and LES models are closer to the experimental data.

The computation time was compared in Table 2, the RANS simulation is about the 2.5 times faster than the LES simulation. In industries, most of the CFD simulations are carried out through RANS approach which is computationally less expensive but gives lower accuracy levels than LES. LES models are more accurate, however, large computational time makes it impractical to use it for routine engineering applications. In addition, by combing the advantage of LES and RANS, DES model appears to be a good option in CFD simulations of the complex passage for the less computation time, but the more computational accuracy.

| Model | CPU time (h) |
|-------|-------------|
| LES   | 22.2        |
| DES   | 14.7        |
| RANS  | 8.6         |

4. Conclusion

This paper performs an unsteady flow simulation with an entire passage model of the hexahedral mesh, a comparative analysis among the capabilities of the RANS, DES, and LES models in general to predict flow and turbulence distribution is conducted to come up with guidelines to use RANS/DES/LES for TC transient simulation. Scholars may resort to using the RANS model if they are interested in obtaining reasonably accurate estimates of velocity magnitudes and flow structures. However, to acquire the more accurate turbulent distributions, LES or DES approach of complete geometries should be recommended. A torque converter is a closed geometry. Typically, the temperature of Auto Transmission Fluid rises up to $80 \sim 120^\circ C$ by the shear heating effect, which would result in a decrease of the torque by a small viscosity. In this study, the accuracy of performance prediction will be improved if the change of temperature could be considered. By combining the advantages of the RANS and LES methods, a complex vortex flow can be accurately simulated by DES model, which exhibits the characteristic of simulating large-scale vortices within the boundary layer of the LES method. The DES model has been proved be a feasible method for accurately and efficiently simulating 3D unsteady turbulent flow in complex passages. Yet, theoretical researches on this aspect are rather little, which would be a direction of future work in CFD numerical simulation of hydraulic torque converter.

Acknowledgements

This work was supported by the National High-tech R&D Program of China (2014AA041502) and Basic Research Programs of Jilin Province (120102JC01032095).

Nomenclature

| CF | Capacity factor |
| Cs | Smagorinsky coefficient |
| $k$ | Turbulent kinetic energy $[m^2/s^2]$ |
| N | Rotating speed [rpm] |
| P | Fluid pressure [pa] |
| r | Radius [m] |
| SR | Speed ratio |
| T | Torque [n-m] |
| TR | Torque ratio between pump and turbine |
| $u_i$ | Time-averaged velocity [m/s] |
| $X_i$ | Coordinate [m] |

Greek letters

| $\eta$ | Efficiency |
| $\varepsilon$ | Turbulent dissipation rate $[m^2/s^3]$ |
| $\mu$ | Circumferential velocity [m/s] |
| $\mu_t$ | Turbulence viscosity coefficient |
| $\rho$ | Density [kg/m] |
| $\nu_T$ | Eddy viscosity coefficient |
| $v$ | Turbulent viscosity $[m^2/s]$ |
| $\omega$ | Relative velocity [m/s] |

Subscripts

| P | Pump |
| S | Stator |
| T | Turbine |
| TC | Torque converter |

ORCID

Chunbao Liu □ http://orcid.org/0000-0001-5513-7707
Changsuoo Liu □ http://orcid.org/0000-0002-5079-5206
Wenxing Ma □ http://orcid.org/0000-0002-2865-8643

References

Barony, S., Olsen, N. R. B., Stoesser, T., & Sturm, T. (2012). Three-dimensional RANS modeling of flow around circular piers using nested grids. *Engineering Applications of Computational Fluid Mechanics*, 6(4), 648–662.

Breuer, M., Jovičić, N., & Mazaev, K. (2003). Comparison of DES, RANS and LES for the separated flow around a flat plate at high incidence. *International Journal for Numerical Methods in Fluids*, 41(4), 357–388.

Caruelle, B., & Ducros, F. (2003). Detached-eddy simulations of attached and detached boundary layers. *International Journal of Computational Fluid Dynamics*, 17(6), 433–451.

Chen, H. C., & Patel, V. C. (1988). Near-wall turbulence models for complex flows including separation. *AIAA Journal*, 26(6), 641–648.
Flack, R. (2005). Experimental flow fields in an automotive torque converter—an invited summary and review paper. *International Journal of Vehicle Design, 38*(2), 240–258.

Galvan, S., Reggio, M., & Guibault, F. (2011). Assessment study of k-ε turbulence models and near-wall modeling for steady state swirling flow analysis in draft tube using FLUENT. *Engineering Applications of Computational Fluid Mechanics, 5*(4), 459–478.

Johnsen, E., Larsson, J., & Bhagatwala, AV. (2010). Assessment of high-resolution methods for numerical simulations of compressible turbulence with shock waves. *Journal of Computational Physics, 229*(4), 1213–1237.

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), 261–268.

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.

Matsuura, K., & Kato, C. (2006). Large-eddy simulation of compressible transitional cascade flows with and without incoming free-stream turbulence. *JSME International Journal Series B, 49*(3), 660–669.

Matsuura, K., & Kato, C. (2007). Large-eddy simulation of compressible transitional flows in a low-pressure turbine cascade. *AIAA Journal, 45*(2), 442–457.

Palau-Salvador, G., Stoesser, T., Froehlich, J., Kappler, M., & Rodi, W. (2010). Large eddy simulations and experiments of flow around finite-height cylinders. *Flow Turbulence Combustion, 84*(2), 239–275.

Schmidt, S., & Thiele, F. (2002). Comparison of numerical methods applied to the flow over wall-mounted cubes. *International Journal of Heat and Fluid Flow, 23*(3), 330–339.

Squires, K. D., Krishnan, V., & Forsythe, J. R. (2008). Prediction of the flow over a circular cylinder at high Reynolds number using detached-eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics, 96*(10), 1528–1536.

Stoesser, T., Kim, S., & Diplas, P. (2010). Turbulent flow through idealized emergent vegetation. *Journal of Hydraulic Engineering, 136*(12), 1003–1017.

Su, W. T., Li, F. C., Li, X. B., & Wei, X. Z. (2012). Assessment of LES performance in simulating complex 3D flows in turbomachines. *Engineering Applications of Computational Fluid Mechanics, 6*(3), 356–365.

Wood, H. G., Yan, Q., & Wei, W. (2013). Parametric analysis and optimization of inlet deflection angle in Torque converters. *Journal of Fluid Engineering, 137*, 031101.

Wu, M. H., Wen, C. Y., Yen, R. H., Weng, M. C., & Wang, A. B. (2004). Experimental and numerical study of the separation angle for flow around a circular cylinder at low Reynolds number. *Journal of Fluid Mechanics, 515*, 233–260.

Xu, C., Chen, L., & Lu, X. (2007). Large-eddy and detached-eddy simulations of the separated flow around a circular cylinder. *Journal of Hydrodynamics, Ser. B., 19*(5), 559–563.

Yang, X., Gupta, S., Kuo, T. W., & Gopalakrishnan, V. (2014). RANS and large Eddy simulation of internal combustion engine flows—a comparative study. *Journal of Engineering for Gas Turbines and Power, 136*(5), 051507.