A numerical study of a torque converter with various methods for the accuracy improvement of performance prediction

Jae Hyuk Jung, Seongwon Kang and Nahmkeon Hur*

Department of Mechanical Engineering, Sogang University, Seoul, Korea
E-mail: jaeh@sogang.ac.kr
E-mail: skang@sogang.ac.kr
E-mail: nhur@sogang.ac.kr
*Corresponding author

Abstract: A comparative study was carried out on numerical methods for simulating a flow inside a torque converter. To investigate the effect of different methods for handling the relative motion of the parts, three methods were considered – the frozen rotor, sliding mesh and mixing plane methods. To improve the accuracy of performance prediction, the influence of viscosity variation with the temperature was studied by a thermo-fluid analysis. From parametric studies on the numerical scheme and the mesh resolution, it is observed that the results with the frozen rotor and sliding mesh methods agree well with the experimental data, whereas the mixing plane method induces a larger difference. The effect of viscosity variation on the accuracy of simulation is also investigated.

Keywords: torque converter; performance prediction; thermo-fluid analysis; capacity factor; CFD.


Biographical notes: Nahmkeon Hur is a Professor of the Department of Mechanical Engineering at Sogang University in Seoul, Korea. He received his PhD from Stevens Institute of Technology in 1988. Before joining the university in 1996, he had been working as a Senior Engineer at Analysis and Design Application Company in USA and as a senior/principal researcher at Korea Institute of Science and Technology. He served as a President of Korean Society of Computational Fluids Engineering from 2007 to 2010 and as a Vice President of the Korean Society of Mechanical Engineers from 2009 to 2010. He is also a Director of Multi-phenomena CFD Engineering Research Center (ERC) funded by National Research Foundation of Korea. His research interests include multiphysics and multidynamics CFD and its application.

Seongwon Kang is an Assistant Professor of the Department of Mechanical Engineering at Sogang University. He received his PhD from Stanford University in 2008 and joined Sogang University in 2010. His research interest includes numerical methods for a complex geometry and turbulent flows with combustion.

Jae Hyuk Jung received his Bachelor degree (2006) in Mechanical Engineering from Myong Ji University in Korea. From March 2009 to February 2011, he had been studying as a master student at Sogang University where he is actively involved in the research on turbomachinery such as fans and pumps. Currently, he is employed by LG Electronics Co.

This paper is a revised and expanded version of a paper entitled ‘A Performance Prediction of a Torque Converter with Various Numerical Methods to Simulate the Relative Motion of the Blades’ presented at Asian Symposium on Computational Heat Transfer and Fluid Flow, Jeju, Korea, 20–23 October, 2009.

1 Introduction

The torque converter is a complex turbomachinery composed of the pump, turbine and stator. It has a role of transferring power from an engine to a transmission. The pump is connected directly to the engine and converts the mechanical power into the fluid power. The fluid power rotates the turbine directly connected to the transmission, and the flow direction is changed by the stator so that the
resulting torque increases. These systematic operations significantly affect the performance characteristics and fuel economy of an automobile. It is therefore very important to analyse the flow characteristics of a torque converter.

So far, many experimental and analytical studies have been done on the flows inside the torque converter passages. By and Lakshminarayana (1991, 1995a, 1995b) measured the static pressure field inside the stator and pump. Their results showed that the primary factor on the static pressure rise around the pump is the centrifugal force, and the static pressure was not rigorously defined at the blade core section. Moreover, they argued that the potential flow theory is applicable to prediction of the static pressure along the blade mid-span with a sufficient accuracy but not at the core and shell sections. On the static pressure field around the turbine, similar conclusions to those for the pump have been derived in several experiments under various rotating speed ratios (0–0.8) between the pump and turbine. The previous studies reported that the centrifugal force has the dominant effect on the static pressure drop in the turbine due to the viscous effect and diffusion of the relative velocity in the low speed ratios. Although there have been several previous studies, most of them focused on the overall performance and did not provide the data of detailed flow fields essential for designing high performance torque converters.

Recent development in CFD makes it possible to predict the flow field by modelling the detailed physics and geometry of a torque converter. Denton (1992) established a 3-D flow calculation method using a basic Euler solver and extended the flow through multiple stages of blade rows. Schulz et al. (1996) used a finite volume method to calculate a 3-D incompressible turbulent flow with the multi-stage model for a torque converter. Both steady and unsteady calculations were carried out to examine the interaction of flows from different stages. Their results showed that, even though unsteady interaction of the stator with the pump or the turbine is negligible, it is necessary to treat the interface region between the pump and turbine accurately. Although their results described the flow physics reasonably well, various physical effects are not fully captured and the circulating flow rate inside the torque converter was not accurately predicted due to the simplified geometry of the multi-stage model.

To investigate the design of torque converters in a hydraulic sense, Kim et al. (1998) carried out a 1-D analysis using the velocity triangle at the inlet and outlet of each component. To determine the circulating flow rate inside a torque converter, they derived the Euler equation using the meridional velocity profile. They also developed a one-passage model consisting with the pump and turbine elements for 3-D steady simulation and used the circulating flow rate from the 1-D analysis as the boundary condition. However, their results showed a discrepancy from the experimental data due to the approximate boundary condition instead of considering the stator passage accurately.

To investigate the core leakage flow, Dong et al. (2002) investigated the effects from a sealing gap between the crown and pump (or turbine) core. The core region was modelled and coupled with the other components such as the pump, turbine and stator. Their results showed that the core leakage flow affects the overall performance such as the torque ratio and the efficiency of a torque converter. They also observed that cavitation occurs inside the torque converter with a high input torque and a low speed ratio condition. From these results, they proposed a new concept of the maximum input torque and an engineering solution to reduce undesirable cavitation.

Kim et al. (2008) suggested a performance estimation model for a torque converter using correlation between the internal flow field and energy loss coefficient. This model is based on the change of the operating condition due to variation of the energy loss coefficient and the least mean square method. The estimated equivalent model resulted in a better agreement between experiments and the theoretical model.

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 characterises the actual flow pattern inside the passage of a torque converter.

Won (2004) performed an unsteady flow simulation with a moving grid and an entire torque converter model. He observed that the unsteady effects are most important in the region where interaction of the pump and turbine is strong. Flow phenomena that cannot be predicted with steady calculation, e.g., the back-flow at the pump outlet, the periodic flow and the unsteady pressure variation are predicted by the moving grid method. These flow phenomena are induced by interaction of the pump and the passing turbine blade. The effect of the blade passing frequency and the harmonic frequency were also investigated.

The present study is based on the study of Won (2004), which presented fairly accurate results with the experimental data. In this paper, we first use similar approaches as Won and perform a parametric study on the numerical method and mesh resolution. Then, the influence of viscosity variation on the performance of a torque converter was investigated by a thermo-fluid analysis in order for an improved accuracy of performance prediction.

2 Methods to simulate the rotating motion

2.1 Analysis model

The torque converter used in the present study is composed of three elements of the pump, turbine and stator as shown in Figure 1. The flow region around each part is modelled by a polyhedral mesh. To handle the rotating parts, three computational methods are considered – the frozen rotor, sliding mesh and mixing plane methods. The effect of different methods on prediction of the performance was investigated. All the simulations in the present study
were performed by using a commercial code STAR-CD V4.08 (CD-adapco Group, 2008).

**Figure 1** Analysis model of torque converter: (a) assembled model and (b) explode view for each part

(a) (b)

2.1.1 Frozen rotor method

The frozen rotor method of STAR-CD enables a user to model cases where the entire mesh is rotating at a constant angular velocity around a prescribed axis. The same feature is extended to multiple rotating frames in which different angular velocities are assigned to different 3-D mesh blocks within the model such as the pump, turbine and stator. The entire flow region was modelled and the coupling of the mesh blocks was applied by using arbitrary interface technique in STAR-CD to each interface of elements for a steady analysis.

In practice, different relative locations of the blades in computational mesh give different results. However, as the number of blades of pump, stator and turbine used in the present simulation are 37, 30 and 29, respectively, the relative location of the blades naturally covers all possible combinations, that is, at the exit plane of one pitch of compressor blades, the turbine blades are located at 29 different positions and so on. Therefore, the relative locations of blades in the present simulation with frozen rotor method are not sensitive to the computational results.

2.1.2 Sliding mesh method

For simulating the rotating parts, the computational grids of the pump and turbine are rotated in space at each time step corresponding to their rotational speeds. The connection of the two computational grids across the interface should be updated every time step, and the attached boundary condition of STAR-CD is adopted. The computational time step is determined so that the pump rotates by one degree at every time step. Similar to the frozen rotor method, the entire flow region of the torque converter was modelled.

2.1.3 Mixing plane method

A mixing plane is a special boundary that simplifies the problem and reduces the required computational effort. The boundary should lie between two mesh blocks, which may be of the same size. This method assumes that the flow in a block does not depend strongly on the instantaneous flow field in the other blocks. The flow fields should depend on time-averaged flow properties of the neighbouring blocks. If there is a correlation between the variation of dependent variables in time and space, the mixing plane averages the flow conditions on both sides and yields averaged space/time values. As the result, two blocks communicate with each other by the quasi time-averaged data. Therefore, the analysis model of each part is divided by the blade pitch angle as shown in Figure 2.

**Figure 2** Analysis model and basic concept of the mixing plane method for torque converter analysis: (a) schematic of the mixing plane method for torque converter and (b) analysis model and boundary conditions for the mixing plane method

The main difference in frozen rotor method and mixing plane approach is in the space averaging process. In the mixing plane approach, the flow field at the exit of a single blade pitch is averaged and the information is passed over as a boundary condition to neighbouring blocks, whereas in frozen rotor method, one does not need to average the field variables since all the flow passages are modelled. In the present study, the mixing plane condition is applied to each interface of the pump/turbine, turbine/stator and stator/pump for a 3-D steady simulation.

2.2 Parameters for performance

The performance of a torque converter is defined by several parameters described below:

\[
SR = \frac{N_{\text{turbine}}}{N_{\text{pump}}}. \tag{1}
\]
CF (capacity factor)
\[
\text{CF} = \frac{T_{\text{pump}}}{N_{\text{pump}}} \times 10^6
\]  
(2)

\(\tau\) (torque ratio between the pump and turbine)
\[
\tau = \frac{T_{\text{turbine}}}{T_{\text{pump}}}
\]  
(3)

\(\eta\) (efficiency)
\[
\eta = \frac{N_{\text{turbine}} \times T_{\text{turbine}}}{N_{\text{pump}} \times T_{\text{pump}}} = SR \times \tau
\]  
(4)

where \(N\) is the rotating speed (rpm) and \(T\) is the torque of each component.

2.3 Material properties and analysis conditions
A torque converter is filled with the Auto Transmission Fluid (ATF). In this study, the density of the fluid is 800 kg/m\(^3\) and viscosity is 0.00566 kg/m-s. The pump and turbine revolve and the stator is fixed. The rotating speed of the pump is 2000 rpm. The rotating speed of the turbine is 0–1800 rpm based on \(SR = 0–0.9\). Incompressible turbulent flow simulations were carried out with STAR-CD V4.08 and the standard \(k-\varepsilon\) model.

2.4 Test of mesh resolution and discretisation scheme
A mesh sensitivity test was carried out to minimise the effect of the numerical error. The meshes used are shown in Table 1. The frozen rotor method along with the Upwind Difference Scheme (UDS) and Central Difference Scheme (CDS) was used for the momentum equation. Figure 3 shows the results of the test. The experimental data in the figure were obtained by using the torque converter dynamometer (Jang and Kwon, 2009). The results with CDS gave overall much better agreement compared to that with UDS as expected. It is important to note from the figure that the results of capacity factor with UDS not only less accurate but also failed to follow the experimental trend of increasing and followed by decreasing capacity factor as speed ration increases. As the total number of meshes is increased from 0.8 million to 3.9 million, the simulation results with the UDS move closer to the experimental data. With the CDS, however, only a marginal improvement is observed with improved mesh resolution. Based on this result, the 3.9 million mesh was used for subsequent simulations.

Additionally, a parametric study on the spatial discretisation method was tried to find the most appropriate method for the present study. Four discretisation methods available in STAR-CD v4.08 – the UDS, CDS, monotone advection and reconstruction scheme and linear upwind difference scheme were considered.

<table>
<thead>
<tr>
<th>Total cell number</th>
<th>812,964</th>
<th>1,907,605</th>
<th>3,897,176</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>Stator leading edge</td>
<td>Pump leading edge</td>
<td></td>
</tr>
</tbody>
</table>

The results with the different schemes are presented in Table 2. The SR of the torque converter was set to 0.1. As shown in the table, STAR-CD predicted the most
accurate result for the capacity factor with the CDS. In contrast, the result with the UDS showed the largest discrepancy from the experimental data. Figure 4 shows comparison of the performance curves between the UDS and CDS. The results with the UDS and CDS are somewhat different from each other. In the experimental result, the capacity factor rises when SR is small and begins to drop when SR is over 0.5. The CDS predicted this tendency reasonably well. However, the UDS showed an earlier drop of the capacity factor. Especially, when SR is small, the UDS shows a large discrepancy.

Table 2  Comparison of the capacity factor between different discretisation schemes for the momentum equation

<table>
<thead>
<tr>
<th></th>
<th>CF (SR = 0.1)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>30.51</td>
<td>–</td>
</tr>
<tr>
<td>UDS</td>
<td>34.83</td>
<td>14.16</td>
</tr>
<tr>
<td>CDS</td>
<td>33.03</td>
<td>8.26</td>
</tr>
<tr>
<td>MARS</td>
<td>34.18</td>
<td>12.03</td>
</tr>
<tr>
<td>LUDS</td>
<td>33.35</td>
<td>9.32</td>
</tr>
</tbody>
</table>

Figure 4  Effect of the discretisation scheme between UDS and CDS with 3.9 million meshes: (a) capacity factor; (b) torque ratio and (c) efficiency

Based on the results in this section, the 3.9 million mesh model and the CDS was selected for the present study. Because the domain for the mixing plane method is significantly smaller than the other methods, the total number of meshes for the mixing plane method is 128,680.

2.5 Comparison between various methods to simulate rotating blades

In this section, different methods for simulation of rotating parts are compared based on the performance parameters. Except for the method for the rotating parts, all the other conditions are kept the same.

The results with different methods are compared in Figure 6. As shown in this figure, the numerical results with the frozen rotor and sliding mesh methods showed a good agreement with the experimental data. However, the mixing plane method showed a larger difference from the experimental data than the others. This trend is observed consistently from all performance curves investigated. For the capacity factor, the sliding mesh method shows the smallest difference when SR is small. In addition, the frozen rotor method showed the smallest difference when SR is large. The sliding mesh and frozen rotor methods show less than 8% difference from the experimental data in the capacity factor. For the torque ratio and efficiency, the difference is less than 2%.

Figure 5  Comparison of the velocity distribution between CDS and UDS in two sections R = 0.07, 0.105. SR = 0.1

Figure 6  Comparison of the performance curves between various numerical methods: (a) capacity factor; (b) torque ratio and (c) efficiency (continues on next page)
The computation time was compared in Table 3. The computation time of the mixing plane method is smallest. It is about 30 times faster than the frozen rotor method. The total number of meshes for the mixing plane method is only 1/30 of the number of meshes for the other methods. The frozen rotor method is about 2.5 times faster than the sliding mesh method.

### Table 3 Comparison of the computation time between different methods (2.0 GHz, 1CPU)

<table>
<thead>
<tr>
<th>Method</th>
<th>CPU time (h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frozen rotor</td>
<td>112</td>
</tr>
<tr>
<td>Sliding mesh</td>
<td>267</td>
</tr>
<tr>
<td>Mixing plane</td>
<td>3.3</td>
</tr>
</tbody>
</table>

### 3 Turbulence model

A torque converter has a complex geometry, which results in a complex turbulent flow. Thus, the choice of the turbulence model is very important. In the present study, a comparison was made between the standard \( k-\varepsilon \) and RNG \( k-\varepsilon \) models. As shown in Figure 7, the RNG \( k-\varepsilon \) model showed a slightly better agreement with the experimental data than the standard \( k-\varepsilon \) model. A further study on the effect of the turbulence model seems to be desirable but this is not the main interest in the present study. Thus, we leave it for the future work.

### 4 Thermo-fluid analysis with temperature variation

#### 4.1 Material property for thermo-fluid analysis

In the present study, the influence of viscosity variation on the performance of a torque converter was investigated by a thermo-fluid analysis for an improved accuracy of performance prediction. The effect of viscosity variation is given by setting the density of the fluid as 800 kg/m\(^3\) and the viscosity as a function of the temperature as shown in Figure 8.

#### 4.2 Results of thermo-fluid analysis

A torque converter is a closed geometry. Typically, the temperature of ATF rises up to 80~120°C by the shear heating effect. A high temperature like this results in a decrease of the torque by a small viscosity. In this study, the initial temperature of ATF is set to 80°C. The boundary temperature of the core and shell is fixed to 80°C. All the other computational conditions are kept the same as the previous simulations.

The comparison of the velocity distribution between the cases with and without viscosity variation from the temperature is shown in Figure 9. The results from both cases are similar to each other. The temperature distribution in two sections is shown in Figure 10. When SR was 0.1, the temperature increases from 80°C to 111°C as the...
solution converges. As shown in Figure 10, the temperature is low in the core and shell regions and high around the turbine blade tip.

**Figure 9** Comparison of the velocity distribution between fluid analysis and thermo-fluid analysis in two sections $R = 0.07, 0.105$. SR = 0.1

**Figure 10** Temperature distribution in two sections $R = 0.07, 0.105$. SR = 0.1

In Figure 11 and Table 4, the capacity factor is compared between the cases with and without viscosity variation from the temperature. When SR is small, the case with viscosity variation predicted a smaller capacity factor than the other. As the result, the difference from the experimental data decreases by 1.55–2.17% with viscosity variation. When SR is large, the case with viscosity variation predicted a slightly larger capacity factor than the other. Thus, the difference from the experimental data decreased by 0.44–0.85%. In conclusion, these results showed that an improved accuracy is possible by including the effect of viscosity and temperature variation.

**Figure 11** Comparison of the capacity factor between numerical results with and without temperature variation

**Table 4** Effect of the temperature variation on the capacity factor

<table>
<thead>
<tr>
<th>SR</th>
<th>Experiment</th>
<th>Numerical analysis (without temperature condition)</th>
<th>Numerical analysis (with temperature condition)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>CF</td>
<td>Difference (%)</td>
<td>CF</td>
</tr>
<tr>
<td>0.0</td>
<td>29.43</td>
<td>31.95</td>
<td>31.49</td>
</tr>
<tr>
<td>0.1</td>
<td>30.51</td>
<td>33.40</td>
<td>32.88</td>
</tr>
<tr>
<td>0.2</td>
<td>31.98</td>
<td>34.68</td>
<td>34.07</td>
</tr>
<tr>
<td>0.3</td>
<td>32.96</td>
<td>35.41</td>
<td>34.71</td>
</tr>
<tr>
<td>0.4</td>
<td>33.84</td>
<td>34.73</td>
<td>33.99</td>
</tr>
<tr>
<td>0.5</td>
<td>34.24</td>
<td>32.66</td>
<td>32.82</td>
</tr>
<tr>
<td>0.6</td>
<td>31.78</td>
<td>30.04</td>
<td>30.18</td>
</tr>
<tr>
<td>0.7</td>
<td>28.25</td>
<td>27.18</td>
<td>27.32</td>
</tr>
<tr>
<td>0.8</td>
<td>25.02</td>
<td>24.12</td>
<td>24.33</td>
</tr>
<tr>
<td>0.9</td>
<td>18.34</td>
<td>19.38</td>
<td>19.29</td>
</tr>
</tbody>
</table>

Figure 12 shows the predicted mean temperature of ATF in the torque converter for various SR. The mean temperature is about 114°C when SR is 0.0 and 86°C when SR is 0.9. This temperature change shows the opposite trend to the efficiency curve because the energy loss increases the temperature of ATF. Considering that the properties of ATF at 100°C are used in the case without viscosity variation, we can see a consistent behaviour for various SR between the temperature difference and the error in the efficiency curve for the case without viscosity variation, which justifies the approach of the thermo-fluid analysis.
5 Concluding remarks

A numerical parametric study for a torque converter was done to evaluate various numerical methods for the moving blades such as the frozen rotor, sliding mesh and mixing plane methods. The results with the frozen rotor and sliding mesh methods agree fairly well with the experimental data such as the capacity factor and torque ratio, whereas the mixing plane method induces a larger discrepancy. About the computation time of these methods, the mixing plane method is fastest because of the simple geometry of single passage. In contrast, the sliding mesh method requires a very long time because of the unsteady analysis.

As the overall performance of a torque converter strongly depends on the average torque acting on all blades, the frozen rotor and sliding mesh methods, which consider the full passages, produced very similar results in the capacity factor. In addition to the small discrepancy, an advantage of the frozen rotor method is that it uses a steady-state analysis, which requires a smaller computational resource compared to a transient analysis. Thus, the frozen rotor method is appropriate for performance prediction of a torque converter in a practical sense.

To further improve the performance prediction, the effect of choice of turbulence models and thermo-fluid simulation was tested. Simulation with RNG $k$-$\varepsilon$ model gave slightly better results compared to that with standard $k$-$\varepsilon$ model. The case with viscosity variation with temperature resulted in a somewhat smaller discrepancy from the experimental data than the case with constant viscosity. These results show the possibility of further improvement of accuracy in performance prediction of a torque converter.

Acknowledgements

This work was supported by the National Research Foundation of Korea (NRF) grant No. 2009-0083510 funded by the Korea Government (MEST) through Multi-phenomena CFD Engineering Research Center (ERC) and the Special Research Grant of Sogang University.

References


