Computational Fluid Dynamics Simulation of Hydraulic Torque Converter for Performance Characteristics Prediction

S. Jeyakumar, M. Sasikumar

Research and Development Division, BEML Ltd, KGF, Karnataka, India

ABSTRACT

In this paper the Computational Fluid Dynamics (CFD) simulation work being done on single stage, three element two phase torque converter using a commercially available software STAR-CCM+. A Computational Fluid Dynamics (CFD) simulation has used to determine the overall steady state torque converter performance characteristics in-terms of speed ratio, torque ratio, K-Factor and efficiency. The internal flow fields distribution rule were analysed in inlet and outlet of the torque converter elements. The CFD STAR CCM+ software solves the incompressible Reynolds-Averaged-Navier-Stokes (RANS) equations on the entire flow domain using the realisable k-Ɛ turbulence model. To verify the accuracy of CFD simulation, the results had validated experimentally by using dynamometer load testing. The torque converter performance characteristics predicted by CFD simulation, is exactly coincidence with experimental results. Based on the validation study, it has proved that CFD can use as a design and analysis tool to make decisions about design direction. This paper demonstrates the confidence level in hydraulic torque converter fluid flow simulations using CFD STAR CCM+ software and describes how it can uses to assist torque converter design today.

Keywords:Computational Fluid Dynamics (CFD), Reynolds-Averaged-Navier-Stokes (RANS), Realisable k-Ɛ turbulence model).

I. INTRODUCTION

Torque converter is an important component of the vehicle automatic transmission system. The primary function of hydraulic torque converter is torque multiplication. It multiplies the torque based on load demand by the vehicle. It transfers power, changes torque through the conversion of mechanical energy between hydraulic energy, and can make vehicle start and change gear smoothly. It provides various advantages, such as making prime mover easily start, protecting prime mover in case of overload, cutting or absorbing the torsional vibration and enabling automatic speed change, and shows excellence in protecting prime mover. The flow through the torque converter is one of the most difficult to analyse. It is highly three dimensional due to complex three-dimensional geometries, with three elements rotating at different speeds. Many methods of experiment using visualization have performed to clarify the internal flow phenomena of torque converter, but experiment needs to be repeated until desired solution has obtained. Thus, it

is difficult to consider them as the method for investigating optimum proposal. In the last years CFD tools make possible to reduce the cost and time for the flow field analysis and have been used to predict the performance characteristics of torque converter because of its property to deliver fast predictions of instantaneous flow field states by numerical calculation. CFD tools use the computer resources to calculate the flow by complex mathematical models. CFD code by STAR-CCM+ software used in present CFD simulations. So far, several numerical attempts have made to investigate the flow field inside the torque converter passages. Most of them, however, adopted a simplified geometry. In this paper, the entire flow region has modelled and the coupling of the mesh block has applied by using arbitrary interface technique in STAR CCM+ to each interface of elements. The relative locations of blades in the present simulation were not sensitive for CFD flow analysis. The hydraulic torque converter used in this paper has designed by BEML. The specification of torque converter (BTC500) is single stage, three elements and two phases has an outer

diameter of 500 mm. The torque converter contains impeller Z_p =25, turbine Z_t=40 and stator Z_s=47 blades respectively. The operating fluid is C4-SAE30 whose density is ρ=832 kg/m3 and the dynamic viscosity is υ=0.0088 Pa.s at 100°C. All simulations and calculations have performed at a constant temperature of the fluid.

II. ASSUMPTIONS AND SIMPLIFICATIONS FOR CFD SIMULATION

The following assumptions and simplifications have made in current hydraulic torque converter CFD simulation.

1. The flow field in the blade passage is steady state. Therefore, the flow parameters are time independent.
2. The fluid is incompressible with constant physical properties (density and viscosity).
3. The fluid flow has simulated at a constant temperature, with no heat transfer simulated.
4. The Realisable k-Ɛ turbulence model has used to simulate turbulence features of the flow.
5. No leakage flow exists inside the torque converter.
6. All the surfaces in the model have assumed to be smooth.
7. The overall flow models have generated with different regions with in-place interface option in CFD STAR CCM+ software.
8. The flow around the lock-up clutch is not included in the CFD model.

III. BASIC EQUATIONS AND TURBULENCE MODELLING

Control equation and turbulence model of fluid motion for the incompressible flow, its control equation [8] is,

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \qquad \text{-----------------------(1)}$$

$$\rho \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\mu + \varepsilon_m)\frac{\partial \overline{u_i}}{\partial x_j}\right] + \rho \overline{F_i} - \frac{\partial \overline{P}}{\partial x_i} \quad \text{-----------(2)}$$

In the equations 1 &2 : u_i(i=1,2,3) expresses the component of velocity in x, y, z direction; Likewise, u_j(i=1,2,3) expresses the component of velocity too, x_i (i=1,2,3) expresses coordinate in x axis, ρ is density, u is dynamic viscosity, ε_m is turbulence viscosity coefficient, F is mass force in unit mass, P is pressure. The above two equations are called N-S equation or Reynolds equation. The finite volume method has used

to solve the control equation. The realisable K-Ɛ model has selected as turbulence model.

IV. FLOW FIELD NUMERICAL ANALYSIS

The entire internal flow field analysis of torque converter divides in to three stages. Flow field numerical analysis procedure has explained in Fig. 1. The quality of the mesh directly affects the convergence, and the accuracy of simulation is highly related to the convergence. To some extent, the simulation results are highly dependent on the torque converter flow passage modelling. The pre processing stage has the important influence on astringency and accuracy of the numerical calculation. Solution computing stage belongs to internal fluid flow calculation using STAR-CCM+ software and post processing stage refers to further analysis on the numerical calculation result by which deep understanding on the flow field in torque converter by various graphical plots.
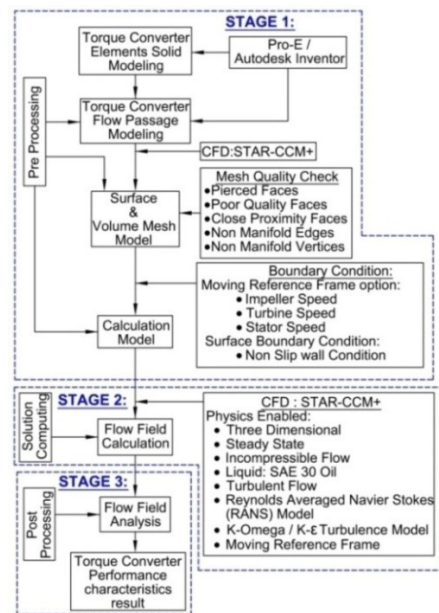

Figure 1: Flow Field Numerical Analysis Procedure

1. Pre Processing (Stage 1):

The pre-processing stage has controls the time consumption and accuracy of the torque converter CFD analysis. The procedure for creating an analytical model in pre-processing stage mentioned in fig. 2.
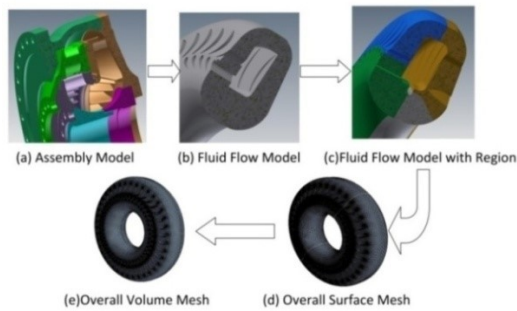
Figure 2 (a, b, c, d &e): Procedure for Creating an Analytical Model

The virtual model of the hydraulic torque converter involves the torque converter assembly model and the circulating three dimensional fluid flow models. The three dimensional model generated using Autodesk Inventor/Pro-E(Fig.2(a)). Utilizing 3D-CAD model software, blade surface contour and the internal surface contour of shell and core have extracted from a torque converter model in Pro-E to configure a fluid flow passage shape. In this way, a fluid flow model that is interrelated with the torque converter model was created (Fig.2(b)). By enabling Moving Reference Frame (MRF) in STAR-CCM+, the fluid flow model was divided in to various regions based on torque converter elements (Fig.2(c)). In-place interface options used for transferring momentum between the elements regions in STAR-CCM+ software.

1.1. Mesh Model of Fluid Passage:

Surface and volume mesh was generated using STAR CCM+ CFD tool (Figure 1(d) and 1(e)). The flow has discretised in a finite volume mesh to carry out the numerical calculation. For fluid flow simulation, convergence properties and the result of analysis (analytical accuracy) greatly influenced by the shape of created mesh. Mesh size has made uniform as possible to prevent an analytical accuracy from dropping. Concerning the blade surface, that requires high accuracy; fine mesh was created along blade shape to create appropriate mesh. The surface quality of the mesh has ensured using surface checking options in STAR-CCM+ software. The torque converter flow calculation CFD model made with approximately One million polyhedral mesh elements for fluid flow fields analysis.

1.2. Boundary Conditions:

Blade surface, shell and core surface are the three different boundary surfaces in fluid flow path.

- The surfaces of the shell and core in fluid flow path are non-deformable solid wall, so the surface boundary conditions are non-slip wall conditions. Meanwhile ignored the influence of surface roughness, these solid walls are hydraulic smooth, its roughness will not affect the flow condition in internal flow field.
- Numerical settings in the internal flow field are the space in flow passage is full of fluid. The impeller velocity remains the same in calculating process, N_i=1000 RPM; the turbine speed changes with the different speed ratio. The stator speed remains zero at different speed ratio until coupling point reaches.

2. Solution Computing (Stage 2) :

The flow field numerical calculation carried out by using STAR-CCM+ Computational fluid dynamics (CFD) software. Three dimensional, liquid (C4 SAE30 oil), steady state, In-compressible fluid flow, turbulence model (RANS: realisable k-Ɛ turbulence model) and Moving Reference Frame (MRF) options are enabled for fluid flow simulation. When the residual error on control equation is in between 10^{-4} to 10^{-10}, the results converge.

3. Post Processing (Stage 3) :

The post processing stage refers to further analysis on the numerical calculation result, by which the deep understanding on the flow field in hydraulic torque converter. The hydraulic torque converter performance characteristics had predicted by using the fluid flow simulation.

V. RESULT AND DISCUSSION

Flow field numerical calculation adopts STAR-CCM+ software, RANS, realisable k-Ɛ turbulence model has selected for prediction of hydraulic torque converter. The basic terminology used in hydraulic torque converter is speed ratio, torque ratio, K-Factor and torque converter efficiency.

1. Essential Relationship for Torque Converter:

1.1. Speed Ratio (i):

It is the relationship between turbine speed and impeller speed. It mentioned in equation 3.

$$i = \frac{N_t}{N_i} \text{-- (3)}$$

i = Speed ratio

N_t = Turbine speed in RPM

N_i = Impeller speed in RPM

1.2. Torque Ratio (T_r):

It is the relationship between turbine torque and impeller torque. It mentioned in equation 4.

$$T_r = \frac{T_t}{T_i} \text{-- (4)}$$

T_r = Torque ratio

T_t = Turbine torque in (N-m)

T_i = Impeller torque in (N-m)

1.3. K-Factor(K):

It is influencing parameter for finalising the stall characteristics. It is representing the relationship of torque and speed of a converter for particular size and mentioned in equation 5.

$$K = \frac{N}{\sqrt{T}} \text{------------------------------------- (5)}$$

N = Impeller speed in $\left(Rad / Sec \right)$

T = Impeller torque in (N-m)

1.4. Efficiency (η):

It is the product of torque ratio and speed ratio. It mentioned in equation 6.

$$\eta = (T_r \times S_r) \times 100 \text{----------------------------------- (6)}$$

η = Efficiency in (%)

T_r = Torque ratio

S_r = Speed ratio

The speed ratio $i = 0$, 0.40 and 0.80 are chosen as typical working conditions, and the rotation rate for different boundary conditions are shown in Fig. 3.
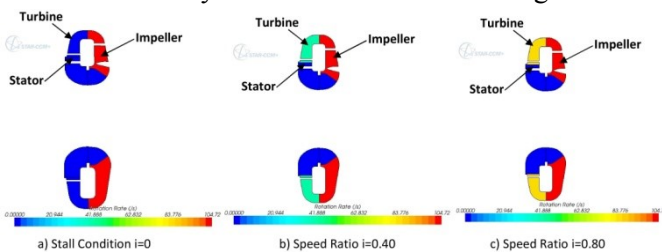

Figure 3: Rotation Rate at Different Boundary Conditions

2. Residual and Torque Monitoring Plots:

The residual and torque monitoring is one of the most fundamental measures of the solution's convergence, as it directly quantifies the error in the solution of the system equations. In a CFD simulation analysis, the residual measures the local imbalance of a conserved variable in each control volume. Similarly torque at different speed ratios (i=0, 0.4 &0.8) were monitored for verifying the local imbalance of a conserved variable in each control volume. Residual and torque monitoring graphs is plotted in Fig. 4, 5 and 6.
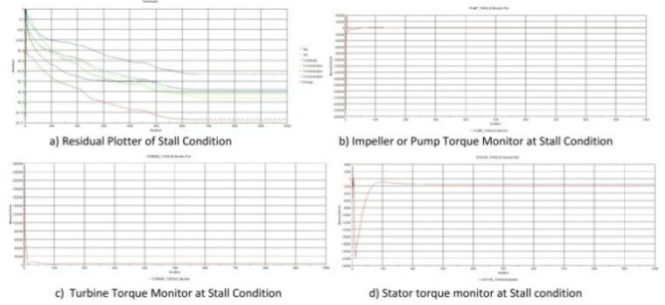

Figure 4: Residual and Torque Monitoring Plots at Stall Condition
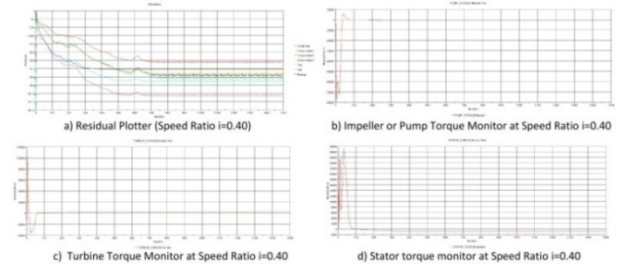

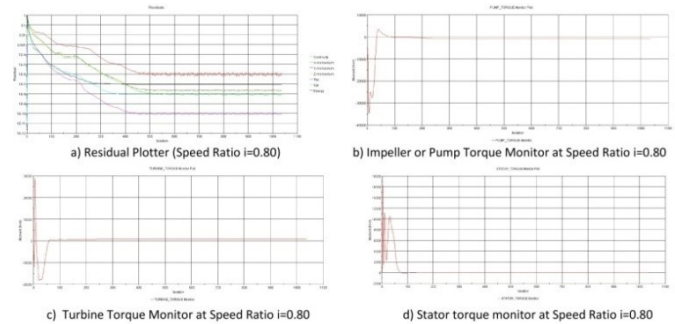Figure 5: Residual and Torque Monitoring Plots at Speed Ratio i=0.40

Figure 6: Residual and Torque Monitoring Plots at Speed Ratio i=0.80

The results of Fig. 4, 5 and 6 shows that, meshing and boundary conditions were more accurate for the above-converged solution. The overall imbalance is less than 0.1% of all variables.

3. Investigation of Velocity and Pressure Distribution:

The velocity distribution of internal flow fields calculated, as shown in Fig. 7, 8 and 9.
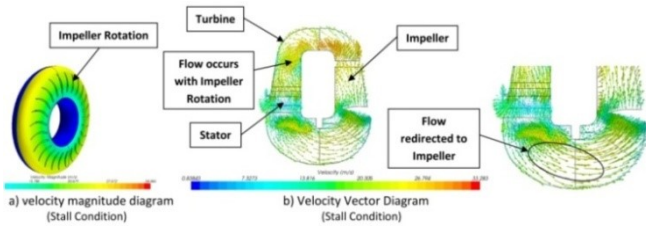
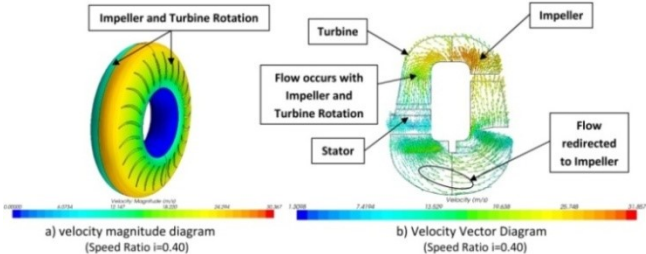Figure 7: Velocity Magnitude and Velocity Vector Diagram for Stall Condition

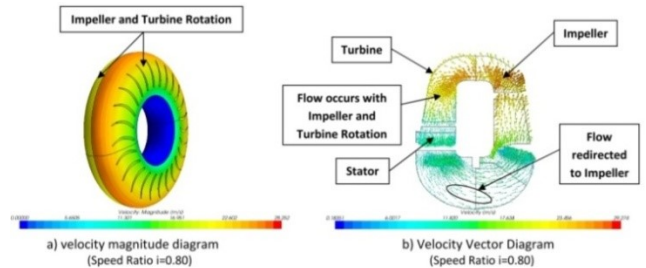Figure 8: Velocity Magnitude and Velocity Vector Diagram for 0.40 Speed Ratio Condition

Figure 9: Velocity Magnitude and Velocity Vector Diagram for 0.80 Speed Ratio Condition

As shown in Fig. 7, 8 and 9, due to the stator static, the flow direction of the impeller at the entrance changes very little under different speed ratio conditions, so the velocity distribution in the inlet surface basically the same under different speed ratio conditions. The velocity magnitude is peak at exit angle portion of impeller and entry angle portion of turbine. Flow occurs at different speed ratio conditions with the help of impeller and turbine rotation by using centrifugal force. Flow also redirected from stator exit portion to impeller suction side for carry over the momentum in-terms of torque and speed variation. Fig. 7, 8 and 9 shows the condition of internal flow passage and the magnitude of flow velocity, it is very tedious to analyse the difference because of flow losses. The pressure distribution at stall, speed ratio i=0.40 & 0.80 mentioned in Fig. 10, 11 & 12.

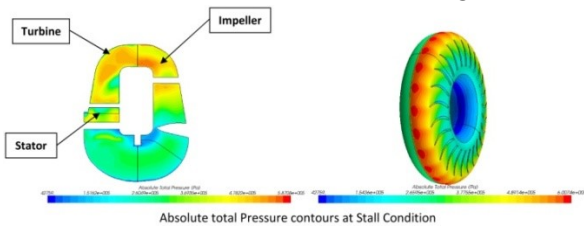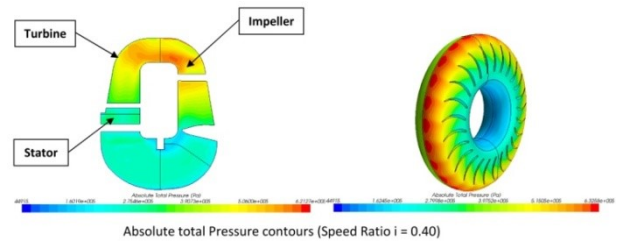Figure 10: Pressure Distribution at Stall Condition
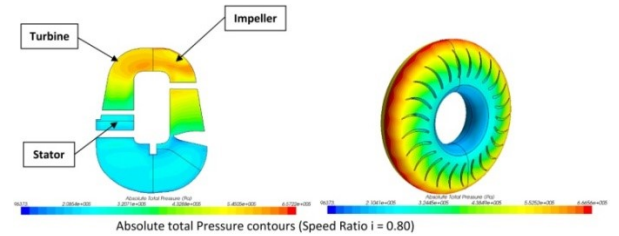

Figure 11: Pressure Distribution (Speed Ratio i=0.40)

Figure 12: Pressure Distribution (Speed Ratio i=0.80)

As per figure 10, 11 and 12; under three different operating conditions, pressure distribution in the entry surface of impeller is almost same, where is proportional distribution along the radial direction from core portion to shell portion and the pressure gradually increases. Pressure is high in the pressure surface area and low in the suction surface area. The severe non-uniform flow angle distribution causes high flow losses in pump passage. To overcome that, almost uniform flow angle distribution made to reduce the flow loss and vortex at stator exit has largely eliminated to reduce the flow losses.

4. Flow Loss Investigation using Rothalpy Concept:

For improving the performance of torque converter, it is important to minimize the flow loss of the fluid that flows through the torque converter three elements. It is very much essential to clarify the internal flow of the passage and improve the shape of flow passage, so that the flow velocity variation and stagnation flow has minimized. To overcome the flow losses in the fluid passage, the concept called rothalpy (I) enabled in STAR CCM+. Rothalpy (I) is a property that remains constant throughout a rotating machine. It is a function of static pressure (P), density of oil (ρ), relative velocity (W) and peripheral or tangential velocity (U) of fluid. It mentioned in equation 7.

$$I = \frac{P}{\rho} + \left(\frac{1}{2} * W^2\right) - \left(\frac{1}{2} * U^2\right)\text{-----------------------}(7)$$

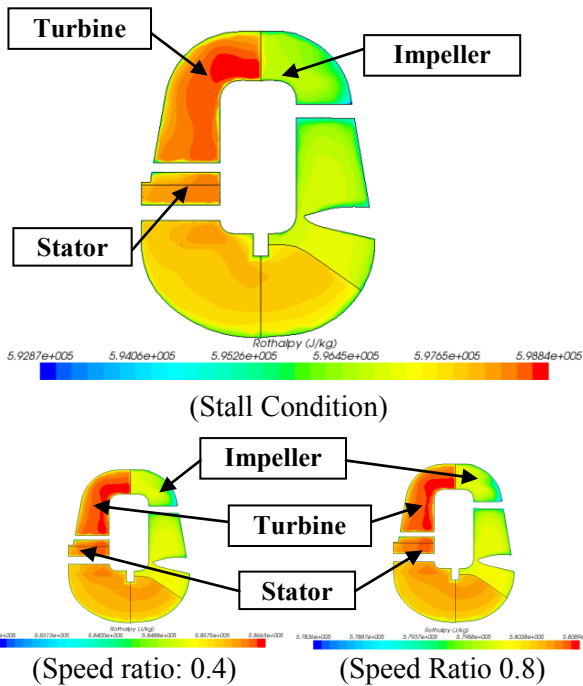(Stall Condition)

(Speed ratio: 0.4)   (Speed Ratio 0.8)

Figure 13: Rothalpy Distribution at Different Speed Ratio Conditions (Stall, Speed Ratio 0.4 & 0.8)

From figure 13, it understood that energy has effectively balanced in torque converter elements at different speed ratio conditions. Fluid circulation has more balanced in torque converter by using these blade geometries. It has confirmed that the method using rothalpy is an effective evaluation tool for finalizing the blade shapes in torque converter.

5. Dynamometer Load Testing:

Torque converter performance characteristics have verified by water dynamometer load testing. 1050 hp Engine used as a prime mover and Water dynamometer D2100-2e used as loading unit. Engine driven power take off (PTO) has used to drive the Charging pump for providing inlet oil supply (C4-SAE30) to the torque converter. Torque converter outlet oil flow passes through the oil cooler to sump. Test carried out by using speed constant mode for different speed ratio conditions by using water dynamometer loading. Torque converter relief valve pressure $(8\sim9.5\text{Kg/cm}^2)$ and torque converter regulator valve pressure $(3\sim5\text{Kg/cm}^2)$ is maintained as per requirement to avoid oil starvation in torque converter. Reading has been captured for torque converter performance characteristics like torque ratio, K-Factor and efficiency at 90~100°C oil temperature. Testing arrangements of hydraulic torque converter with prime mover (engine) and water dynamometer mentioned in Fig. 14.

Figure 14: Dynamometer Load Testing Arrangements

Since the amount of core leakage flow in this torque converter is relatively less, the correlation of the torque ratio, K-Factor and efficiency between the CFD simulation prediction and dynamometer load test results are almost equal for this torque converter. The torque converter performance characteristics comparison results with CFD and Dynamometer load test results are shown in Fig. 15.
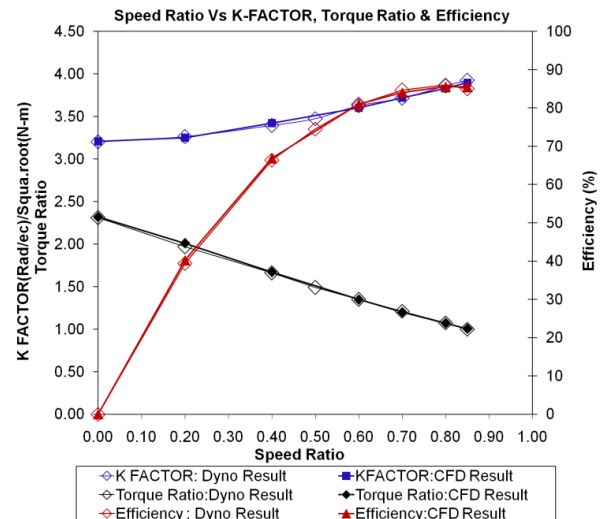

Figure 15: The Torque Converter Performance Characteristics Comparison
(CFD & Dynamometer Load-Test Results)

VI. CONCLUSION

The flow in the torque converter computed and visualized by using Computation Fluid Dynamic software, STAR CCM+, which is difficult by traditional experimental and analytical methods. A systematic torque converter performance characteristics prediction method has formulated by using STAR CCM+ CFD Software. It has high accuracy, reliability and can replace the part of the experimental work of torque converter. The computationally evaluated torque converter performance qualitatively coincides with the experimental results. This methodology used for the

simulation analysis of the flow field of the torque converter, which can reduce the development costs, shorten the research cycle, then improve design, and manufacture process.

VII. REFERENCES

[1] V.J. Jandasek, "Design of Single-Stage, Three Element Torque Converter", Design Practice : Passenger Car Automatic Transmission, Third Edition, AE-17, SAE Transaction, vol.5, 1970, pp.201-226.

[2] Sehyun Shin, Kyung-Joon Kim and Dong-Jin Kim "The Effect of Reactor Blade Geometry on the Performance of an Automatic Torque Converter", SAE 2002-01-0885

[3] Haruo Sakamoto, Kiyoto Suyama and TokimoriSaka "Study on Torque Converter Circuit Profile" SAE 920765

[4] Joaquim Emanuel Santini and Hans Inge Weber "Mathematical Model to Predict the Performance Characteristics of the Torque Converters", SAE 921452E.

[5] Kiyoyuki Minato, Kenichi Sakamoto, Michitoshi and KatsuroFujitani : "A Performance Prediction of Hydrodynamic Torque Converter", SAE 900555

[6] Robert R. By and John E. Mahoney "Technology Needs for the Automotive Torque Converter-Part 1: Internal Flow, Blade Design and Performance", SAE 880482.

[7] E Joaquim E. Sanitini and Hans L.Weber" Theoretical Formulation For Power Losses In Torque Converters",   SAE 931702E.

[8] GE Anlin, TIAN Hua, WANG Jian "Amending redesign of torque converter based on three dimensional flow analysis". Journal of Jiangsu University (Natural Science Edition), pp. 206-208, March 2005.

[9] Jae Hyuk Jung, Seongwon Kang and NahmkeonHur "A numerical study of a torque converter with various methods for the accuracy improvement of performance prediction" Progress in Computational fluid Dynamics, Vol.11, Nos.3/4, 2011

[10] Seunghan Yang, Sekyun Shin and IncheolBae School of Mechanical Eng. Kyungpook National University, Taegu, Korea, Taekyung Lee, Korea Powertrain corp. Taegu, Korea, SAE Technical paper series, 1999-01-1046.

[11] E.W. Upton "Application of Hydrodynamic Drive Units to Passenger Car Automatic Transmissions" SAE, 1961 Summer Meeting, 359A.

[12] Yu Dong, Vamshi Korivi, Pradeep Attibele and Yiqing Yuan, DamlerChrysler Corp. "Torque Converter CFD Engineering Part 1: Torque Ratio and K Factor Improvement Through Stator Modifications, SAE 2002-01-0883