

Dynamic Three-Dimensional CFD Simulation of Closed Circuit Torque Converter Systems

Chiranth Srinivasan
Simerics Inc.

Darshak Joshi
Luk USA LLC

Sujan Dhar and De Ming Wang
Simerics Inc.

ABSTRACT

This paper details the capability of PumpLinx® and Simerics® in simulating both Steady-State (Multiple Reference Frame) and transient, three dimensional torque converter performance and predicting the coupling point in a closed torque converter system in automatic transmission. The focuses of the simulation are in predicting the performance characteristics of the torque converters at different turbine to impeller rotating speeds (speed ratios) for 7 different torque converter designs and determine the coupling point at 70°C temperature. The computational domain includes the complex 3D design of all the impeller, turbine and reactor blades, the path ways that the oil travels between the above three components and the leakage gaps between these components. The physics captured in the simulation include the turbulence in the flow field and the rigorous treatment of the Fluid Structure Interaction (FSI) for the one-way free wheel reactor in predicting coupling point. The one-dimensional rotating dynamic modeling of the reactor enables the simulation of the whole range of speed ratios starting from 0 to 0.99. The integrated values of the transient torque on all the rotating components are found out to determine the torque ratio, K-Factor and efficiency. The comparisons with the hardware measurements show less than 5% differences between the test and simulation results. The consistency of the numeric schemes used for the simulation combined with the extremely fast run times and close comparisons with the test measurements adds value to the use of PumpLinx as a tool for simulating full torque converter systems.

CITATION: Srinivasan, C., Joshi, D., Dhar, S., and Wang, D., "Dynamic Three-Dimensional CFD Simulation of Closed Circuit Torque Converter Systems," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(1):2016, doi:10.4271/2016-01-1345.

INTRODUCTION

Torque converters are commonly used in cars, buses, locomotives and construction equipment for smooth torque transmission between engine and automatic transmission. The typical torque converter cross-section, as shown in Figure 1, is a recirculating hydrodynamic, mixed-flow turbo machine (containing both axial and radial flow) with three independent blade cascade elements that control the internal flow field. In the torque converter, Pump assembly is connected to engine and turbine assembly is connected to transmission. Torque will be transferred from engine to transmission using torque converter during start-up, drive and coast conditions. The fuel efficiency of the car depends a lot on the torque converter's ability to transfer the torque effectively in all conditions. Each torque converter in the car has unique blade designs to match the effective balance between power and fuel efficiency. These designs are achieved by optimizing the blade angle to meet the required torque ratio (TR) and K-factor. Some of the other common uses of torque converters include torque amplifications during startup conditions, a

soft start from standstill condition, and the capacity for damping transmission through the absorption of torsional vibrations introduced from the engine.

The working of a torque converter is akin to the working of a centrifugal pump. For the impeller side of the torque converter: (a) the flow enters this domain axially from the inlet ports, (b) the impeller solid which is connected to the engine side rotates with the engine speed, and accelerates the fluid in the radial and tangential direction to the blades, (c) the fluid pressure increases, and the fluid exits the impeller domain to enter the turbine domain at an orthogonal angle to the turbine blades. In the turbine part of the torque converter: (a) the shape of the blades are designed in a way to change the pressure of the fluid into the rotation of the turbine components, (b) the blades are also oriented in a direction which lets the flow exit the turbine axially into the reactor before it reverts back to the impeller. In the reactor domain: (a) the blade design is such that the flow from the turbine is redirected into the pump, (b) the reactor has a device, clutch, ensuring that it rotates in only the direction of rotation of the

turbine, (c) the reactor initially will be stationary and when the coupling point is reached, the reactor starts rotating with certain speed dictated by the flow field. Figure 2 and 3 shows a close-up view of the blade architecture on the impeller, turbine and reactor components. For torque converters, the performance characteristics, like the torque ratio, k-factor and efficiency are typically studied from stall to cruise range of operations. However, torque converters can be operated at high positive speed ratios beyond cruise point. At high speed ratios, beyond 1, the turbine is overrunning and exceeding the speed of the impeller.

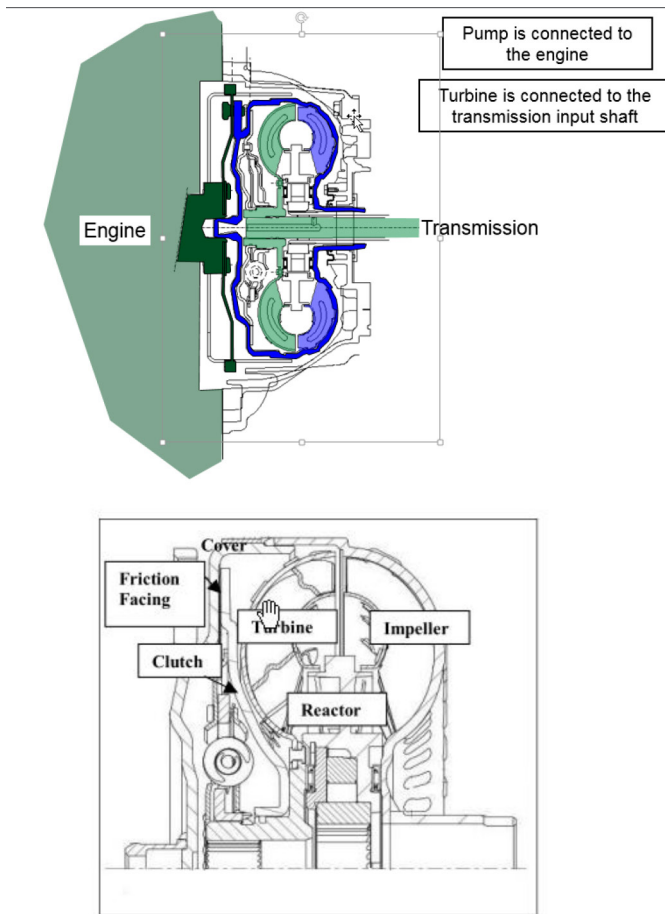


Figure 1. Cross sections of a torque converter

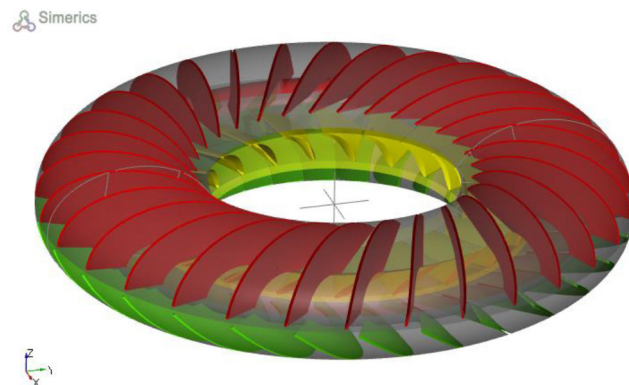


Figure 2. Complete torque converter circuit, with the impeller blades marked with red, reactor blades with yellow and turbine blades with green.

In this paper, numerical simulations are performed on model setup using the designs of torque converters manufactured by Schaeffler -LuK USA. The performance curves of 7 different torque converters are studied using the commercial CFD code Simerics MP® and PumpLinx®. The complete architecture of the three components, impeller, reactor and turbine, along with the gaps between these components are also included in the model domain. The model has no approximation and the results of the test are compared with the simulation.

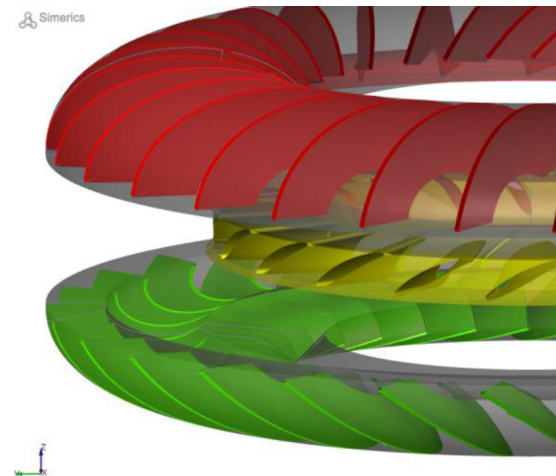


Figure 3. Expanded view of the torque converter blade architecture, the impeller (in red) rotates in +z direction, causing the concave side of the blades to push the fluid onto the concave side of the turbine blades (in green). The reactor blades (in yellow) are oriented such that the flow circulates back into the impeller side to help further work on the impeller blades.

BACKGROUND

The ancestry of the first designs of fluid coupling systems similar to torque converters dates back to early 1900s with the works of Dr. Hermann Föttinger. The first attempts to implement this system in vehicle transmission dates back to 1920s when English engineer Harold Sinclair along with Dr. Bauer of Vulcan-Werke tried to use them in London buses. In 1930, Daimler Company of Coventry, England started introducing fluid coupling systems into their flagship cars. In 1939, General Motors Corporation introduced the Hydramatic drive, which was the first fully automatic transmission system installed in a mass production automobile. Spannhake [1] devised the first theoretical one-dimensional analysis of the torque converter. Ziebart [2] applied photographic techniques and a large water table to visualize flow cascades in different blade configurations of the Föttinger torque converter. His studies showed that the interactions between the turbine and reactor become very important when the gaps between these elements become very small.

Lots of experimental work has gone into understanding the workings, flow field and characteristics of a torque converter. Bahr et al. [3] and Gruver et al. [4], separately, in their laser velocimeter measurements observed a low velocity wake/separation region. Dong et al. [5] in their experiments found high losses due to wake-flow in the core-suction corner, while also observing strong secondary flow in the impeller mid-cord. Flow separation was also a salient characteristic observed in the experiments of Liri Y [6]. Experimental works by Gb and Lakshminarayana [7,8,9] show the centrifugal force as the main

factor behind the static pressure rise inside the torque converter in the impeller section. They applied the potential flow theory to accurately predict the pressure field in the blade mid-section but not at the shell and core regions of the torque converter. Several other works have looked at the turbine domain to apply the potential flow theory at varying speed ratios (0-0.8) and have arrived at similar conclusions to that of the pump domain described above.

By et al. [9] used a modified form of the Navier-Stokes equation to solve for the flow field in the impeller side of the torque converter. The conclusion from the study showed the inlet velocity profile to be of huge importance in the pressure loss and the secondary flow is dictated by the impeller rotation. A two-dimensional, steady, incompressible Navier-Stokes equation code was used by Marathe et al. [10] to get the flow field of the reactor. The values of the flow field obtained for the mid-span flow was accurate for the designed conditions but not for the off-design points.

Recent advances in CFD have enabled successful modeling of torque converter with varying blade geometries, gaps between components and with solving for all the applicable physics. Cigarini and Jonnavihula [11] used the commercial code STAR-CD to solve for the three-dimensional flow field of a torque converter using the steady interaction technique implemented in that code. The performance characteristics matched well with the test data. Shultz et al. [12] used a finite volume approach to solve for the 3D turbulent incompressible flow in a multi-stage torque converter. Conclusions from the study show that unsteady interaction between impeller and turbine is negligible but the interface treatment between the both is important. Their results captured the flow physics accurately but their geometry was largely simplified to capture physical effects and circulating flow rates precisely. STAR-CD was also used by Ejiri and Kubo [13] to optimize the turbine bias angle and the contraction ratio for maximum torque converter efficiency. Works of S. Shin, H. Chang and M. Athavale [14] use the modified Navier-Stokes equation with the k- ϵ model for turbulence to obtain the flow characteristics of a torque converter. Kim et al. [15] in their works developed both a 1-D and 3-D model for their analysis. They ran their 1D model to supply the circulating flow rates to their 3D one-passage model. Their results showed discrepancy with the experimental data because of the disregard of the reactor domain in the simulation. Good agreements are observed with the test measurements. S.A. Talukder and B.P. Huynh [16, 17] used the code CFD-ACE+ of CFDRC and ESI Group to perform 3D simulations of a complete closed torque converter. Their studies were aimed at understanding the change in efficiency of the torque converter with the change in number of blades in the reactor component. They also analyzed the use of shear-thinning and shear-thickening fluids as their operating fluid and documented the behavior of the torque converter with these changes. Dong et al. [18] in their numerical works investigated the effects of the leakage gap between the crown and the impeller/turbine core. The core regions were included and modeled in their simulation. The results showed the performance characteristics like the torque ratio and efficiency are affected when the core regions are included in the simulation. Their study also showed cavitation effects on the torque converter at high input torques and low speed ratios.

Work of J. Schweitzer and J. Gandham [19], applied the commercial CFD package CFX TASCflow to validate the CFD results with experiments. Their results showed good comparisons with the test data with a promise that CFD could be a tool for design optimization before manufacturing. The limitation with this numerical setup is the use of a section of the model to perform the analysis with a periodic boundary condition applied over the repeating boundaries. This sort of process becomes a little tricky for a device like torque converter which has an odd number of impeller, turbine and reactor blades and rarely symmetrical in nature.

Most of the CFD research in the area of torque converter has been concentrated on simplified geometry and have ignored temporal variations of the flow field such as blade interactions. Won [20] performed an unsteady flow simulation with the entire torque converter using a moving grid. Results of Won showed the prediction of back-flow at the pump-outlet, the periodic flow and unsteady pressure fluctuations using this method. Reasonable comparison to the experimental data is presented in these results. J.H. Jung, S. Kang, and N. Hur [21] used the commercial code STAR-CD V4.08 to perform simulations of a 3D torque converter flow. They tried three different mesh approaches, the frozen rotor, sliding mesh and mixing plane method. Results of comparisons of the capacity factor and torque ratios show fairly good agreements with experimental data for frozen rotor and sliding mesh approaches. However, the mixing plane approach shows large discrepancy. However, the study reports that the simulation times for the frozen rotor and sliding mesh approaches are around 5 and 11 days, respectively, while the mixing plane approach takes only 3.3 hours. Jung et al. [21] also concluded that the results with RNG k- ϵ turbulence model showed better comparisons with the test data compared to k- ϵ model. Also, the results show a smaller difference from the test data when the viscosity is varied as a function of temperature.

Even with the vast amount of research going on in the area of CFD to study torque converters, there are still some limitations with the current state of CFD for torque converters. The efficient operation of the torque converter has a huge impact on the fuel economy of the automobile. Hence, understanding the flow characteristics inside a torque converter is of tremendous importance in optimizing design and improving its efficiency. Few of the limitations of current state of the art CFD includes: (a) the difficulty in CFD 3D grid generation with the complex impeller, turbine and reactor blades, and the tiny gaps between these components, (b) accuracy of the numerical handling of the moving/deforming grid and the interface between moving and stationary parts, and finally (c) the wall clock time for simulation, which for accurate solutions are in the order of days, rather than hours.

One of the main aims of this paper is to demonstrate and validate that the use of CFD as a productive diagnostic tool can be expanded through good implementation of CFD software together with advanced numerical algorithms and physical models for different design studies of torque converters. With accurate solutions of this complex problem along with the fast turnover times, we demonstrate the capacity of the new generation CFD package (Simerics® and PumpLinX®) to be a workforce in modeling complex torque converter designs.

MESH GENERATION AND MODEL SETUP

Mesh Generation

Simerics MP® and PumpLinx®, which are the same base codes with some template options available in the latter, both have the automated binary-tree meshing capabilities to provide a fast and effective means of mesh generation without sacrificing accuracy. The mesh for the torque converter will be created as three domains, impeller, turbine and reactor, and all the mesh generated will be body fitted binary-tree meshes. This type of grid belongs to the family of unstructured body-fitted Cartesian grids. In this type of meshing, an overlaid cubic grid is refined by factors of two as it approaches regions requiring higher geometric resolution. The important point to note here is the marching of the mesh from finer to coarser is always from the surface, hence the surface meshes are always at least half the size of the inside mesh. [Figure 4](#) shows a -x section view of the same mesh and we can see the surface refinements of the cells near the blades, shell and core. [Figure 4a](#) shows a zoomed in view of the same. At the boundary, the grid is cut to conform to the surfaces defining the fluid domain. As we can see from the [Figure 5](#), the mesh is predominantly hexahedral cells with the cells cut to fit the surface architecture. The algorithm is structured to handle the unstructured binary tree scheme with more details of the same provided in the works of Ding et al. [22]. The advantages that the binary tree mesh offers are multifold and are discussed in detail by Ding et al. [22].

In order to create the mesh for the torque converter, we follow a consistent procedure so that we can have a control on the size and distribution of cells in the domain. The meshing is completely automated. We initially create a STL bounding box surface, cubical in shape with one side of the cube 1.2 times the diameter of the torque converter. For mesh creation, the user selects the set of closed surfaces that defines a particular component, along with the upper and lower bound of this bounding box surface, and then clicks on a button to create a high quality grid in less than a minute on a personal computer (PC). The selection of the box ensures that we enforce this artificial bounding box while meshing the components of the torque converter. The cubical shape of the bounding box ensures that the mesh created for all components are all isotropic, with same size in all three directions. The cell size selected is such that the cells near the surfaces of the blades and shell are at least half that of the mid-plane, enabling us to have good aspect ratio and skewness of these cells for our solution. In the inner region, the faces of the binary tree cells are perfectly perpendicular to each other, and at the boundary of the grid, orthogonality ranges between 45 and 135 deg.

For mesh creation in PumpLinx®, the smaller cell size on the surfaces ensures a smaller mesh size near the blade and solid boundaries. These small cells along with the predominantly hexagonal shaped cells and orthogonality of the mesh takes care of any boundary layer effect, and ensures the solution consistency. We have used seven different designs of torque converter obtained from Schaeffler-LuK, having varying torus diameters, number of impeller, turbine and reactor blades, shape and curvatures of the impeller, turbine and reactor blades. [Table 1](#)

shows the mesh count, the number of blades, for these 7 different cases of torque converters that we have simulated for this study. The order of the mesh is always around 1.5 to 3 million cells with the standard procedure for outlined for meshing.

Table 1. For the seven different torque converter cases, the mesh size, torus diameter, number of impeller blades, turbine blades and reactor blades are shown

	Mesh Count	Torus Diameter (mm)	Impeller Blades	Turbine Blades	Reactor Blades
Case 1	2429656	430	31	27	29
Case 2	3126511	263.25	31	29	25
Case 3	2186426	260	31	29	21
Case 4	2627683	260	31	29	25
Case 5	2098897	260	31	29	25
Case 6	2193966	260	31	29	21
Case 7	3702100	260	31	29	34

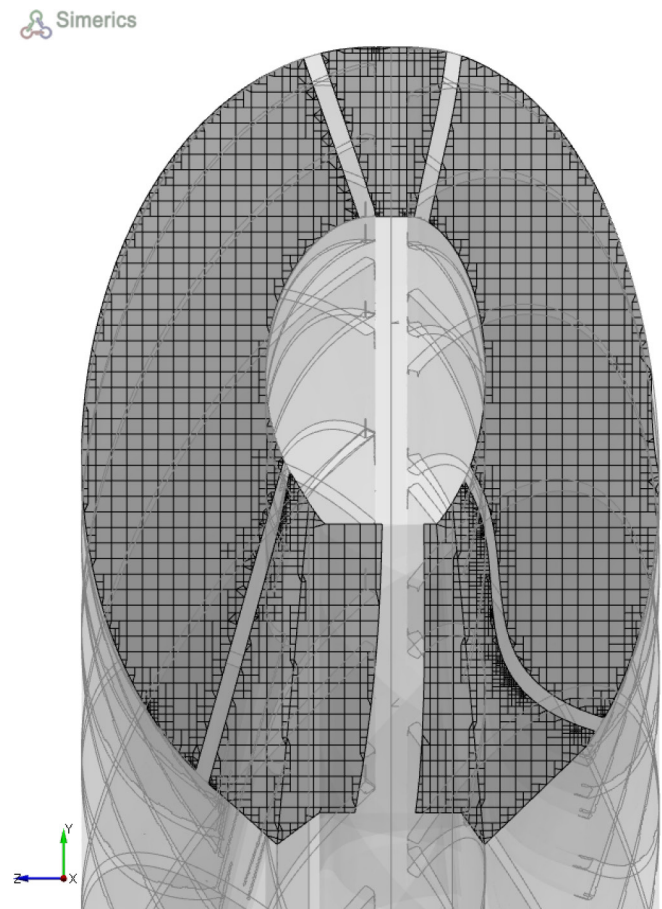


Figure 4. -x cross-section cut view of the binary tree mesh

The impeller, turbine and reactor domains are separated in a CAD tool while preprocessing for mesh generation in such a way that the meshes can be deformed independently. The meshes are generated separately for each of the domains. They are then connected together