

# The CFD Application for Efficient Designing in the Automotive Engineering

Norihiko Watanabe, Shinya Miyamoto, Masayuki Kuba, Junichi Nakanishi

Software CRADLE Co., Ltd.

Copyright © 2003 Society of Automotive Engineers, Inc.

## ABSTRACT

Recently, many CFD analyses which assist mechanical designing have been carried out in every aspects of automotive engineering. While many CFD tools have been developed for easies of use and advances in CFD analyses tools continue, instances of effective design using these tools are not well discussed in the literature. This paper discusses a CFD analysis system, SC/Tetra, is equipped with an automatic hybrid mesh generator, a high-speed flow solver and a state of the art postprocessor. In SC/Tetra, the flow solver is built on a node-based finite-volume discretization method to achieve both high-speed computation and small memory consumption. The advancing-front method is applied to the automatic mesh generator. These features of efficient computation are desired to achieve shorter design times. Discussions are made after performing sample computations of a torque converter for an automobile, which presents difficulties for efficient CFD analysis. The difficulties are based on the numerical modeling, the mesh generation and retaining the accuracy of the computational result. SC/Tetra is optimized for effective computation and copes with these difficulties with adequate strategies. CFD tools will be able to give more parametric data to accelerate product design processes and to improve products.

## INTRODUCTION

A large number of recent industrial design works have been supported by CAE (Computer Aided Engineering) which includes wide variety of computer aided tools such as computer aided design (CAD), computational fluid dynamics (CFD) and finite element analysis (FEA). Commercial CFD codes as well as CAD, FEA codes have been rapidly advancing due to market demands. CFD codes are used in many design phases to reduce the time and cost of product development. A lot of effort has been devoted on the development of CFD tools for automotive engineering[1,2,3,4,5]. In previous paper,

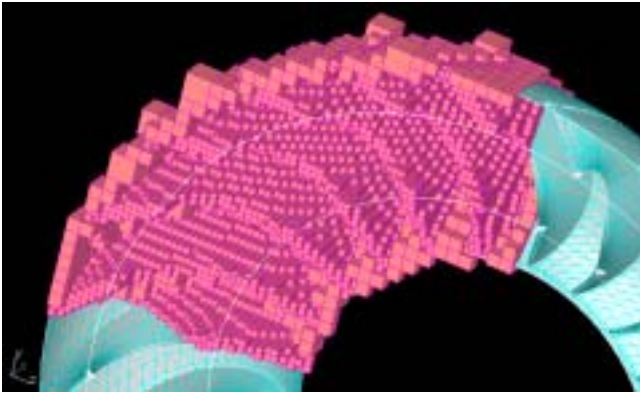
CFD software was evaluated and always recognized as a helpful tool to many industrial fields. However, some difficult problems which have to be solved still exist hindering the further advancement of CFD. They are, for example, correcting an imported model before creating a mesh, choosing a suitable turbulence model and calculating a complex flow field with moving bodies. Despite these restrictions, many automotive makers are progressively using CFD tools in their designs to achieve a highly efficient product development process. In fact, some trials of the completely systematic designing process including CFD have been performed[2,4].

CFD codes for application to practical design works have been evaluated in detail before. One might expect that there would be some universal ways for the evaluation of CFD codes. Unfortunately, a lot of detailed results of evaluations have not been published except only for common benchmark tests, even if they were expected to be of great help for many designers of different disciplines.

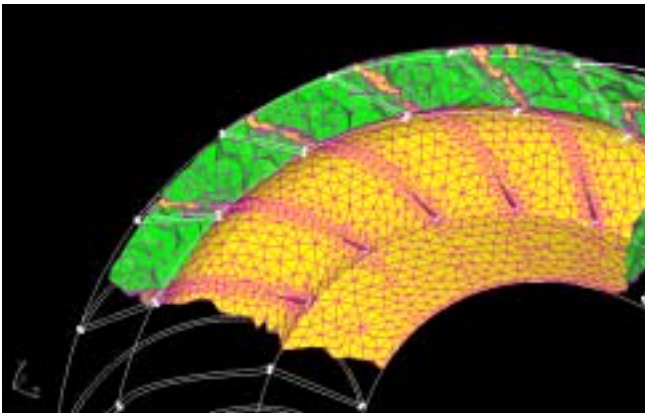
An analysis of the flow field of a torque converter is conducted to propose a way of effective engineering design using CFD applications. A torque converter is selected for the comparison of conventional design flow and the proposed design flow since it usually requires skilled human inputs to solve the internal flow and interpret the result. The torque converter used in this paper is currently being design in an automobile company in Japan. The numerical evaluation of a CFD code for the development of a torque converter is also performed.

## CFD CODE

SC/Tetra has been developed as a pure design tool from the start. The development of SC/Tetra was proposed in details in the previous paper [6]. The preprocessor, solver and postprocessor of SC/Tetra were designed to perform all the required steps for the evaluation of the



**Fig. 1 The octree of torque converter (part).**



**Fig. 2 The surface mesh derived from the octree system in fig. 1 (part).**

flow field of complex models as if they are on one assembly line. It imports a designed model from CAD, solves systems of equations and visualizes the results. In addition, the results can be exported to FEA software for further stress analyses. It is suitable for design work because it is not only high-speed but user friendly and low-cost at the same time. The high compatibility to the available CAD and FEA software is also the strength. The features of each program code are described in the following.

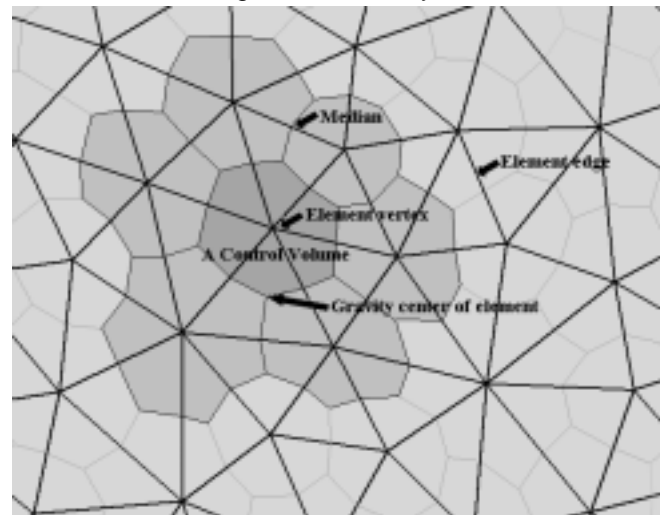
### THE PREPROCESSOR

The preprocessor can generate tetrahedral and prismatic elements. Such elements make it feasible to represent complex analysis regions such as an engine air intake port and an engine room. Most CFD analysis tools have been used with a tetrahedral based hybrid mesh system. The computational models what become the computational domain are directly imported from familiar CAD system as polygon data files. The imported model usually does not require any time consuming corrections since most of the corrections are done automatically. An octree is used as the intermediate interface to decide the concentration of mesh. Part of the octree of the torque converter is shown in Fig. 1. An octree is the bulk of cuboids which covers the whole computational model. A designer distributes the mesh

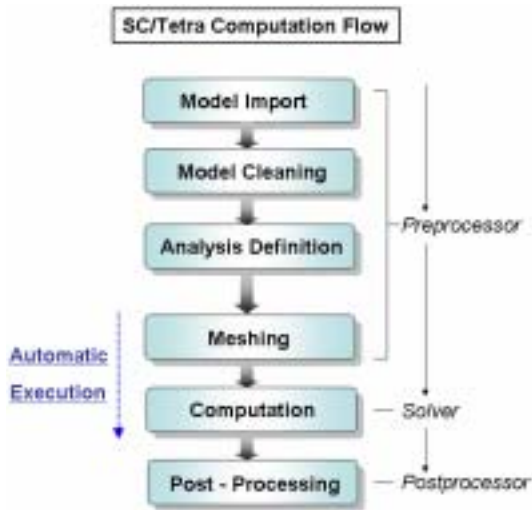
concentration by refining and joining the cuboids. After determining the mesh concentration, the preprocessor creates the mesh automatically based on the size of cuboids in the octree. Part of the mesh of torque converter is shown in Fig. 2. The distribution of mesh concentration is comparable to Fig. 1. The method of mesh generation is based on the advancing-front method [7]. The geometrical quality of generated mesh is mostly satisfactory since a high quality mesh smoother is implemented in this mesh generator. The smoother rearranges the size and shape of elements to remove largely distorted elements. After the generation of volume mesh, prismatic elements are placed on wall boundaries to maintain the accuracy of solutions at the vicinity of boundary layers. The notable features of this preprocessor are the high speed process and easy interface.

### THE SOLVER

The solver is a computer program that numerically solves the systems of equations. Most CFD codes use the Navier-Stokes equation. The Navier-Stokes equation is formulated with three conservation equations of mass, momentum and energy. The thermo-fluid properties such as density, velocity, pressure and temperature are calculated from these conservation equations. Many numerical algorithms to solve these equations have been proposed such as FDM (Finite Difference Method), FEM (Finite Element Method) etc. SC/Tetra is implemented with the FVM (Finite Volume Method) strategy to keep the physical conservational properties constant. This solver program uses the 'cell vertex' discretization scheme to reduce the amount of computer memory consumption. A schematic two-dimensional drawing of the 'cell vertex' scheme is shown in Fig. 3. The numerical results are held at nodes of elements. A control volume does not lie on each volume element but a volume region which is defined around a node of elements. To solve the equations more effectively, the AMG(Algebraic Multi Grid) method[8] is applied to the numerical time integration. It is very effective in reducing



**Fig. 3 Schematic drawing of the cell-vertex strategy.**



**Fig. 4 Execution flow of SC/Tetra.**

the computational time for steady state analysis. Concerning the numerical accuracy, the second-order upwind differential scheme based on the MUSCL approach[8] was implemented for spatial discretization. This feature is suitable to derive highly accurate numerical solutions without instability which comes with higher-order differencing schemes. In this solver, seven numerical turbulence models for the turbulent flow computations are available including three low-Reynolds type models.

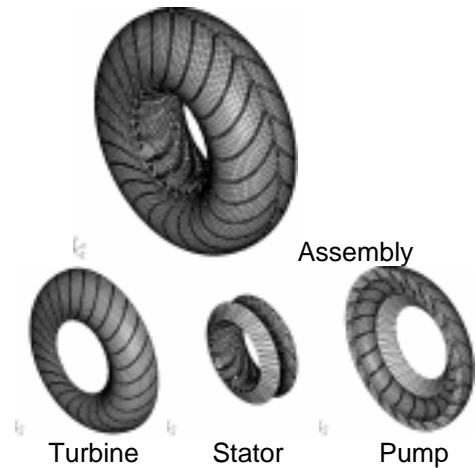
The solver has many computational options. More specifically, the solver can adopt blocking mesh strategy and the ALE (Arbitrary Lagrangean Eulerian) method supports moving mesh analysis. It is feasible for the numerical computations of models which include rotating objects.

#### THE POSTPROCESSOR

The postprocessor allows scientific visualization from the computational results of the solver. The visualization technique includes contour lines and vector animation of an arbitrary cross section, stream lines and particle tracers in a field, oil flow on a surface and many others. Combining the above technique, a design engineer can visualize the results in every aspect of variables and spaces. It is possible to make a video clip such as a car going through foggy road using the function to change the view point automatically, just like a movie scene. The output is also effective for presentations since it appeals to human intuition via visible sense. All the figures of flow visualization shown in this paper were made only by the postprocessor included in the SC/Tetra package.

#### WORKFLOW OF A COMPUTATION

The workflow of a computation is summarized in Fig.4. After the definition of the mesh arrangement and numerical conditions, the rest of the analysis can be automatically executed. For more user independent analysis, the AMR (Adaptive Mesh Refinement)



**Fig. 5 Three piece of CAD model used for computations; left: turbine, center: stator, right: pump.**

technique is also implemented in the SC/Tetra. Using AMR, designers can obtain computational results by simple preparing a CAD model and defining the numerical conditions. Parametric computations can be performed sequentially.

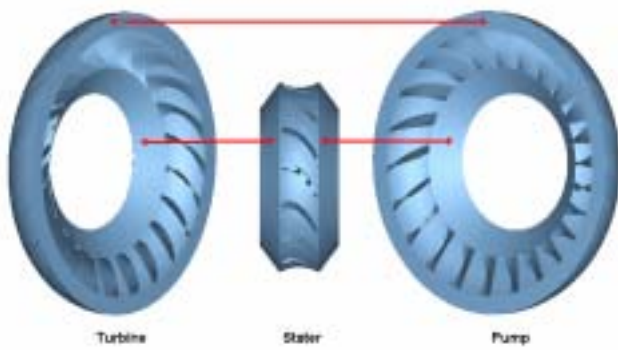
### CFD BASED DESIGN

#### DESIGNING A TORQUE CONVERTER

A torque converter is a fluid coupler used in automobiles for the purpose of multiplying torque and transferring kinetic power from the engine to the transmission system. A typical torque converter consists of three blade-rotors of pump, stator and turbine. The shape of the blades has significant influence on the performance of a torque converter such as torque amplification and controlled slippage. The blade shape is well considered and designed to achieve the target performance. For effective design, a lot of human experience is necessary. In this experience and experiment driven field, CFD tools are used to reduce the time span of the design cycle.

#### THE NUMERICAL MODEL AND ASSUMPTIONS

The computational model for analysis is shown in Fig.5. It is also shown in Fig.1 and Fig.2 of the previous section. The rotational axis is fixed in the X-direction of the Cartesian coordinate. The mesh is generated for each model and then all three meshes are merged with a blocking-mesh strategy. In Fig.6, the schematic illustration of the mesh blocking strategy is shown. The flow is assumed to be three-dimensional incompressible viscous turbulent flow. The standard  $k-\epsilon$  turbulent model[10] is used for computations of turbulent features. The second-order accuracy discretization scheme is applied for spatial discretization and a first-order accuracy discretization scheme is used for implicit time integration. The analyses are performed for both steady and unsteady analysis. It is well known that the steady

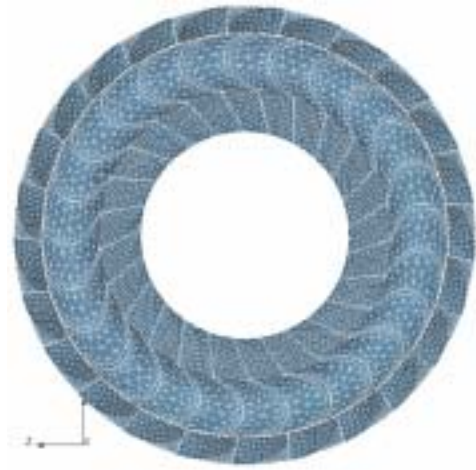


**Fig. 6 Three elements of ALE computational model of the fully geometrical torque converter.**

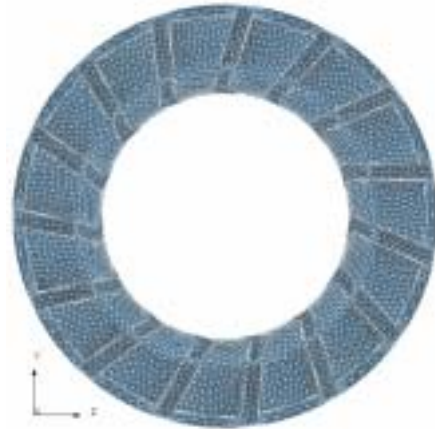
state computation is typically faster than the unsteady computation because of the difference of numerical approach. Typical time step in the unsteady computation is set shorter than the time step in the steady computation. Moreover, the computation cycle for unsteady computation is much more than that of steady computation. Therefore, the accumulated effect due to numerical viscosity is considered larger in unsteady computations. However, the unsteady computation is useful to investigate time-dependent phenomena. If it is significant to describe time-dependent flow dynamics in the torque converter, unsteady approach must be applied. To investigate such difference, two types of computation were performed. For the unsteady computations, time step was set as 0.3[ms]. This time step coincide the 5.0 Couran number. Each unsteady computation was performed with 360 time steps to reach the time of three rounds of pump. Both results will be compared and discussed in the next section. The fluid in the torque converter is assumed to be automatic transmission fluid whose density and viscosity are  $868.0[\text{kg}/\text{m}^3]$  and  $3.877 \times 10^{-2}[\text{Pa}\cdot\text{s}]$  respectively. This automatic transmission fluid has viscosity index of 176. The viscosity becomes  $3.877 \times 10^{-2}[\text{Pa}\cdot\text{s}]$  at the point of the temperature of 313[K]. However, in this computation, the variation of viscosity with temperature was ignored because the temperature variation is negligible after reaching to the constant rotational speed.

The meshes of the turbine, stator and pump are shown in Fig.7. The torque converter mesh system consists of three parts of independent meshes. Each mesh is based on hybrid elements which include tetrahedral and prismatic elements. Prismatic elements are inserted on the wall boundaries to better simulate the frictional force to the fluids. A close up view of mesh of the pump is shown in Fig.8. Prismatic elements on the pump-blades are detailed in the figure. The numbers of elements are 154,692, 155,615 and 148,744 for the pump, stator and turbine respectively and the total number of elements is 459,051.

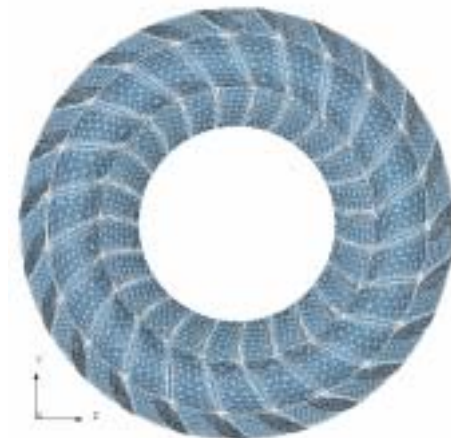
For the comparison of numerical results with experimental results, the torque ratio and K-factor are



**Turbine**



**Stator**



**Pump**

**Fig. 7 Mesh system used for computations; upper: turbine, middle: stator, lower: pump.**

calculated from the results of computation. The torque ratio represents the performance of torque amplification from an engine to a transmission. K-factor indicates pump torque which has unit of  $[\text{rpm}/\text{Nm}^{0.5}]$ . It influences the comfortableness of start of a vehicle. These performances are important characteristics for torque converter design. They vary with speed ratio (ratio of turbine and pump rotation rate), so the computations were performed at speed ratios from 0.0 to 0.9 in every

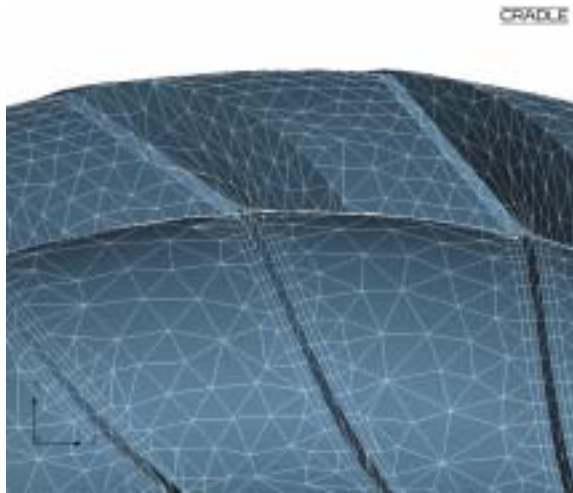


Fig. 8 Close view of the pump blades mesh.

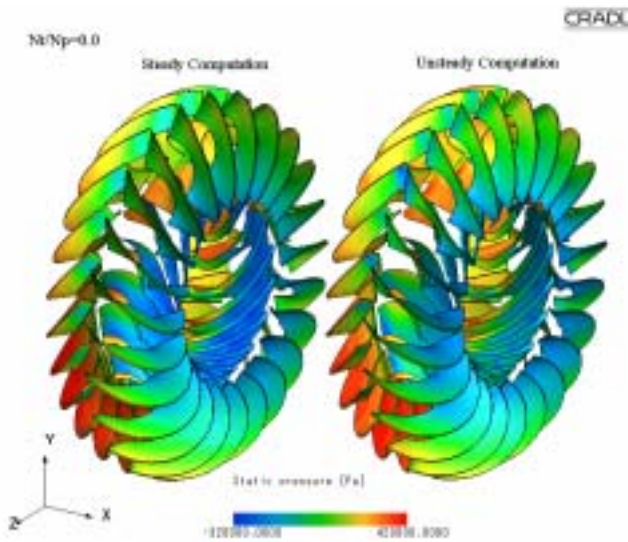


Fig. 9 Static pressure distribution on the blades at speed ratio of 0.0 (stalled); left: steady computation, right: unsteady computation.

0.1 steps. Accordingly, 10 computations were performed. And in these computations, pump rotation speed is assumed as 2,200rpm. The torque on each blade rotor was estimated by the force on the blades which is calculated by integrating the pressure distribution.

The Pentium III 866MHz CPU class PC with 768Mbytes of memory is used for calculations.

## RESULTS AND DISCUSSIONS

### THE FLOW FIELDS AND ACCURACY

The computational result of the static pressure distribution on the blades is shown in Fig.9. The left and right figures show the results of steady computation and unsteady computation respectively. Fig. 10 shows velocity vector maps on the xy-plane center cross section. The result of steady computation is shown on the left and the result of unsteady computation is shown

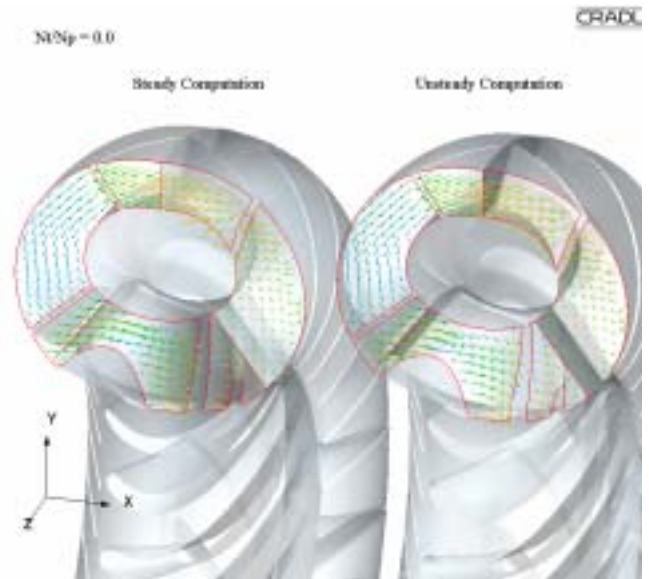


Fig. 10 Velocity distribution on the xy-plane at speed ratio of 0.0 (stalled); left: steady computation, right: unsteady computation.

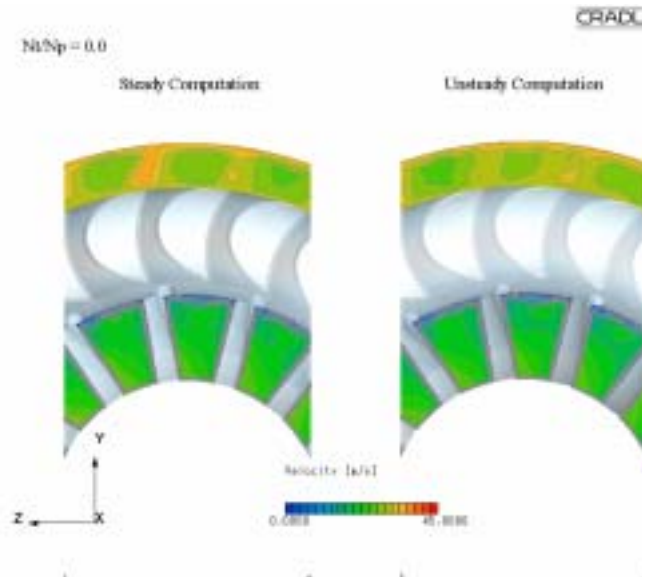
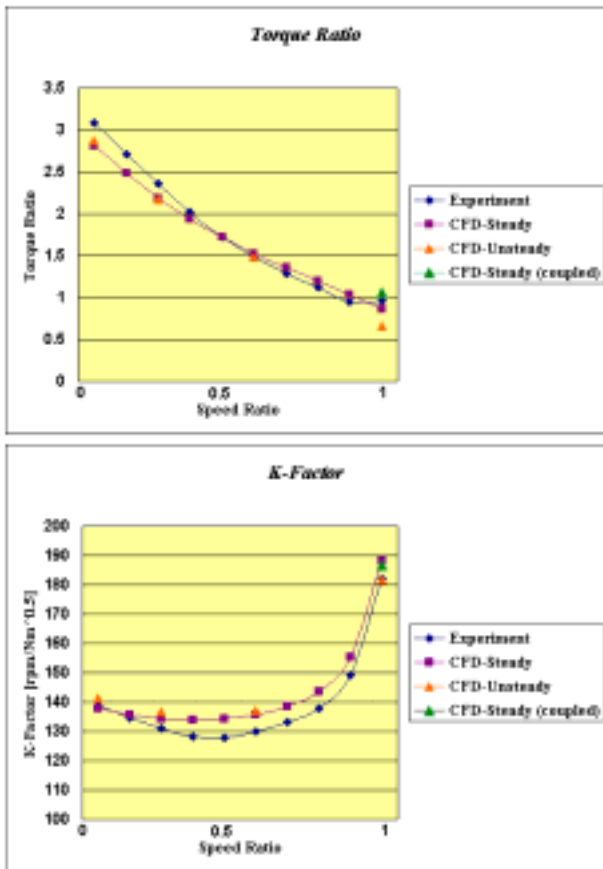


Fig. 11 Velocity distribution on the yz-plane at speed ratio of 0.0 (stalled); left: steady computation, right: unsteady computation.

on the right. The turbine is on the left of each model. The flow is from the pump toward to the turbine and the fluid returns to the pump through the stator changing the direction of flow. Fig. 11 shows the velocity distributions on the yz-plane center cross section. A pseudo gray scale color contour map describes the magnitude of velocity. Each result shows good agreement between steady and unsteady computations. It can be concluded that the suspected numerical violations due to numerical viscosity was very small in these unsteady computations.

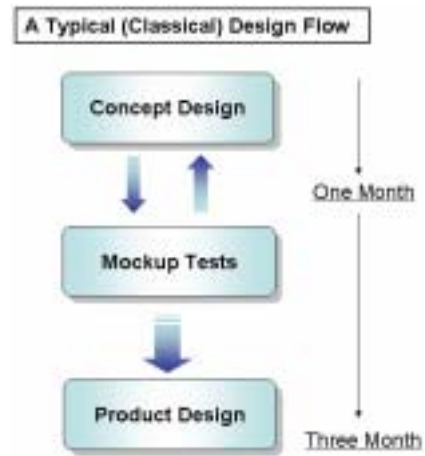
The summary of numerical torque ratio is shown in the upper side graph in Fig. 12. The numerical torque ratio is plotted against speed ratio. The numerical results are in good agreement with experimental observation



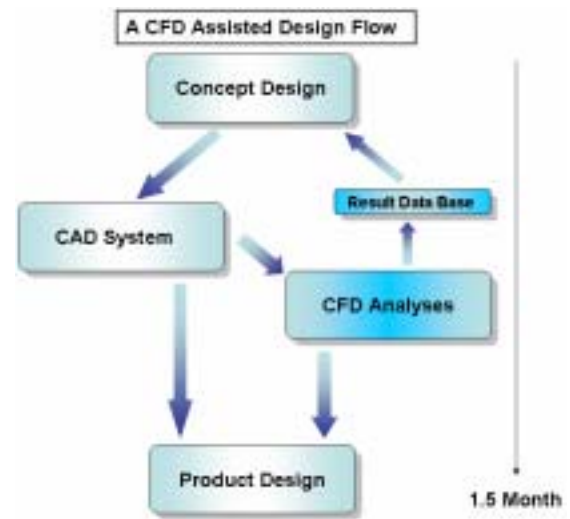
**Fig. 12 Numerical results; upper: torque ratio versus speed ratio, lower: K-factor versus speed ratio.**

qualitatively. In the case of speed ratio is 0.9, computation results have comparable large discrepancy from experiment because of the experimental result was considered with the coupling effect. In high speed ratio, typical torque converter releases the static stator to avoid large flow resistance force of stator. Stator will be rounded freely in high speed ratio. This process is called as coupling. The numerical result considered with coupling for steady computation is also plotted in Fig.12. This result can be comfortably used for quantitative predictions of torque ratio. The summary of numerical results of K-factor is shown in the lower side of Fig. 12. The results have slightly larger error compared to the results of torque ratio. However, it can be sufficiently used as the material for qualitative discussion for the evaluation of K-factor. Results of unsteady computation are slightly larger than steady computation results especially in the region of low speed ratio. The larger K-factor indicates the larger pressure force on the pump blades. Similarly, the larger torque ratio gives the larger pressure force on the turbine blades. Overall, it is comprehended that pressure force is evaluated larger in the unsteady computations.

The time for mesh generation is 15 minutes using only 90Mbytes of memory. And the time of computation is about 9 minutes for all steady computations, 520 minutes for unsteady computation (360 time steps). The



**Fig. 13 Typical design flow of a torque converter.**



**Fig. 14 Design flow of CFD based torque converter design.**

effective design of SC/Tetra allows the quick mesh generation and steady computation. One case of computation can be performed in 30 minutes including time for setting the numerical conditions.

#### CONTRIBUTION OF CFD TO PRODUCT DESIGN

The biggest advantage of using computational tools in the design process is that it saves time of development and helps to optimize products easily. The design flow based on CFD analysis is discussed in previous sub section. Typical human experience based design flow is shown in Fig.13. For a typical engineering product like a torque converter, about one month is spent to conduct a mockup testing from a concept design and additional one month for production of the mockup. When the mockup test is over, the detail product design is started. At least three months are required from the concept design to the production. On the contrary, in a CFD

based design, only 1.5 months are required for product design as shown in Fig.14. In addition, databases of numerical results are constructed in the repetitive design flow. This database will contribute to improve the performance and reduce the total design time in the development of future products. Because of the reliability and variety of the data obtained from CFD tools, the designer can evaluate the product in many aspects. For example, torque ratio and K-factor are varied with configurations of blades such as number of blades, area, and curvatures. In conventional design, such configurations are determined only by the experiences of designers. Some tunings are done again and again to fix the conceptual design. The experiences based on the entire database of knowledge in the designers' mind were the most useful tools for torque converter design. The CFD applications are strongly expected to do the work of experienced engineers and the followings for the evaluations of torque converters in the near future. 1) Thrust force of center axis. 2) Total internal pressure variation due to the motion of the automatic transmission fluids. 3) Evaluation of the cavitations. 4) Total virtual design of more compact torque converter. Human experiences are irreplaceable to the concept design. However, it is clear that recent CFD tools can help to construct products based on more efficient design flow.

## CONCLUSION

A practical CFD based design work was described and discussed including the evaluation. The applied problem of designing the torque converter was analyzed. The results are summarized as follows.

1. The CFD tool, SC/Tetra, is used and evaluated. It is suitable for the use in design works, considering the quick convergence and adequate accuracy of solutions.
2. The CFD based torque converter design can reduce the design time compared to previous human experience based design.
3. The powerful CAE tools such as CFD tools are expected to reduce time and give more information for quality engineering design works.

## ACKNOWLEDGMENTS

Authors would like to thank the all the engineers who gave us inputs to this paper for their dedicated collaborations.

## REFERENCES

1. Kurosaka K. et. al., 'Development of Air Ventilation Performance Analysis Method Using CFD', *Proc. JSAE Symp.* No.20005454 (2000). (In Japanese)
2. Ikeda K. and Okumura K., 'Development of Fluid Analysis System for Air Condition Using Adaptive Mesh Generation', *Proc. JSAE Symp.* No.20015143 (2002). (In Japanese)
3. Koromilas C. et. al., 'Rapid Aerodynamic Development of Two-Volume Vehicle Shapes', *SAE paper* 2000-01-0488 (2000).
4. Asano H. et. Al, 'Use CAE Technology in DENSO A/C Development', *SAE paper* 2001-01-0033 (2001).
5. Okumura K., 'CFD Simulation by Automatically Generated Tetrahedral and Prismatic Cells for Engine Intake Duct and Coolant Flow in Three Days', *SAE paper* 2000-01-29 (2000).
6. Matsushima T., "An Automatic Mesh Generator Based CFD System to be used as a Design Tool", *SAE paper* 2001-01-0037 (2001).
7. Chan C.T. and Anastasiou K., 'An Automatic Grid Generation Scheme by the Advancing Front Method', *Commun. Numer. Methods Eng.*, 13, pp: 33-46 (1997).
8. Elias S.R., et. al., 'An Adaptive Agglomeration Method for Additive Correction Multigrid', *Intl. J. Numer. Methods in Eng.*, 40, p: 887 (1997).
9. van-Leer B., 'Towards the Ultimate Conservative Difference Scheme V. - A Second-order Sequel to Godunov's Method', *J. Comp. Phys.*, 32, p: 101 (1979).
10. Launder B.E. and Spalding D.B., 'The Numerical Computation of Turbulent Flows', *Comp. Methods in Applied Mech. and Eng.*, 3, pp: 269-289 (1974).

## CONTACT

An author can be contacted at the following e-mail addresses: watanabe@cradle.co.jp (Watanabe N.).