

Effect of Actual Gaps and Leakage Flow of Variable Geometry Turbine on its Performance

Natsuko Motoda and Akihiro Yamagata
IHI Corporation

ABSTRACT

Recently, a variable flow capacity turbine has become important for automotive turbochargers to satisfy the demands of engine performance and emissions. Especially, a variable nozzle turbine is an indispensable component for diesel engines of both passenger and commercial vehicles to realize a high boosting from the low end to the rated engine condition.

On the other hand, since a variable nozzle turbine has several gaps and clearances around movable nozzle vanes to ensure the smooth operation under the hot gas condition, therefore there is a possibility that these complicated geometries may cause the additional loss and the turbine efficiency decrement.

The authors conducted detailed CFD calculations of a variable nozzle turbine taking account of nozzle vane clearance, strut pins, spindles. As most realized calculation, a full-domain analysis which includes a flow leakage through a nozzle link assembly, and these complicated geometries' effects on a turbine performance are revealed when nozzle opening is varied from the closed to the fully-opened condition.

CFD results show that the nozzle vane clearance dominates the turbine performance at the nozzle closed condition and strut pins have a relatively large influence at the fully-opened condition. In a full-domain analysis, it is confirmed that the predicted turbine efficiency becomes close to the experimental result and CFD modeling considering actual gaps and leakage is necessary for the aerodynamic design of a variable nozzle turbine and its improvement.

CITATION: Motoda, N. and Yamagata, A., "Effect of Actual Gaps and Leakage Flow of Variable Geometry Turbine on its Performance," *SAE Int. J. Engines* 5(2):2012, doi:10.4271/2012-01-0708.

INTRODUCTION

In recent years, due to the demand for CO₂ reduction, further improvement of fuel consumption is required of automotive engines. In order to reduce the fuel consumption, engine displacement becomes smaller and turbo-charging is widely applied for both diesel and gasoline engines to improve the decrement of torque and power caused by the downsizing of engines.

To satisfy the requirement of such engine performance for the whole operating range, a variable flow capacity turbine which has movable nozzle vanes is mainly used in turbochargers for automotive diesel engines. In order to ensure the smooth operation of nozzle vanes, there are many gaps and clearances around nozzle vanes and these links in the actual turbine, which will cause additional pressure loss and turbine efficiency decrement. However, it is difficult to clarify the flow mechanism of turbine performance deterioration only by the cascade flow analysis.

On the other hand, the ability of a numerical simulation is growing up rapidly due to the progress of computational resource and parallel computing technique. Therefore, CFD

analysis for the whole domain of a radial turbine has been realistic and applied to the practical design of a variable geometry turbine.

In this paper, the authors conducted several CFD calculations of a variable geometry nozzle turbine to reveal the effects of actual turbine's gaps and clearances on turbine efficiency and flow capacity. Each calculation has been conducted for several nozzle openings which are from the nozzle closed to the fully-opened condition.

CFD results are also compared with the actual turbine performance measurement. Prediction accuracy of a turbine performance due to the above realistic and complicated CFD modeling is investigated and further problems to improve the design procedure of a variable nozzle turbine for automotive turbocharger are discussed.

TURBINE CONFIGURATION

A cut-way view of the RHV4 turbocharger with a variable nozzle turbine is shown in [Figure 1](#). This turbocharger has been developed in IHI Corporation for passenger vehicles' diesel engines. The turbine stage is constructed of a volute to

lead a gas flow into a nozzle, a variable nozzle and its link system and a radial inflow impeller. The nozzle vanes can change a flow velocity at the impeller inlet and optimize a turbine flow capacity for the various engine operating conditions from the idled to the max-power condition. The variable nozzle system is constructed of nozzle vanes, shafts, levers to rotate vanes and a ring to drive levers. Each vane can rotate around its pivot axis and vary its throat area to change a turbine flow capacity.

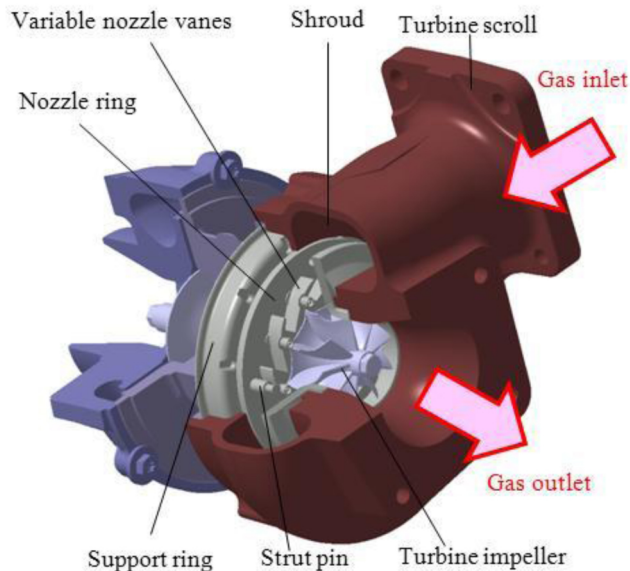


Figure1. Cut-way view of RHV4 turbocharger

Figure 2 shows axial views of nozzle vanes and a radial impeller at typical nozzle openings which are closed, medium and fully-opened conditions. The numbers of nozzle vanes and impeller blades are 11 and 10 respectively, and the impeller tip diameter is about 45mm. At the low engine speed condition, nozzle vanes are closed to increase the impeller inlet velocity to rotate turbine wheel with a relatively smaller gas flow rate. Nozzle vanes are being opened when the engine speed becomes higher, i.e. a gas flow rate increasing. Finally, at the engine rated condition, nozzle vanes are fully-opened with keeping a minimum radial gap with an impeller tip to prevent an excessive flow acceleration in the nozzle and avoid the over boosting and the increase of engine pumping loss.

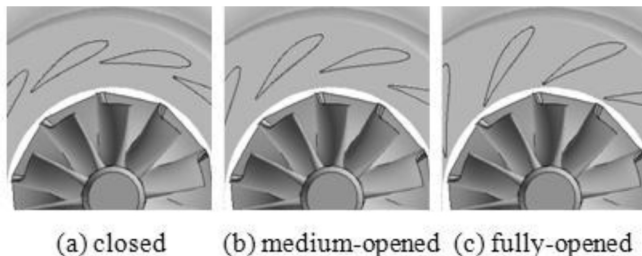


Figure2. Axial view of variable nozzle vanes and impeller blades at three typical nozzle settings

COMPUTATIONAL MODEL

Figure 3 shows a schematic view of a computational model used in this study. CFD calculations are conducted for the entire region of a turbine stage, which contains a turbine scroll, nozzle vanes, an impeller, an exit pipe and a nozzle cavity.

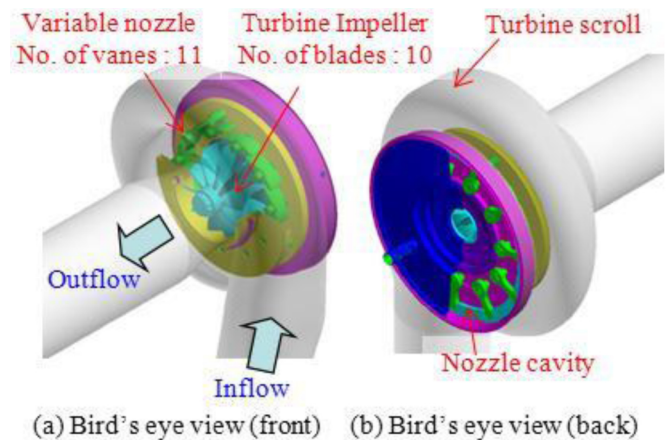


Figure3. Schematic view of a computational model of a whole turbine stage

As a solver, a commercial CFD code Fluent 6.3 was employed in this study. All calculations are compressible viscous flow analyses at a steady state condition. Using a function of multi-rotating reference frames to rotor-stator boundaries, rotor-stator interaction between nozzle vanes and an impeller is expressed by frozen-rotor assumption in which a computational mesh of a rotating part does not move, i.e. circumferential relative location between a rotating part and a stationary part is not changed during the calculation, while a centrifugal force and a Coriolis force occurred in the rotational regions are considered. As a turbulence model, k-epsilon Realizable two equation model within Fluent is used to estimate turbulent viscosity. Also the advanced wall function, which is called Enhanced Wall Treatment, is applied to estimate the shear stress in a viscous sub-layer near the wall.

Within a variable geometry turbine, there are several gaps and clearances around variable nozzle vanes. In this paper, the authors focused on the effects of nozzle side clearance, strut pins and a flow leakage around a nozzle link system especially. These gaps and clearances are needed to ensure nozzle vane's smooth operation at the hot gas condition exceeds 800degC. However, these may also cause additional pressure loss and turbine efficiency decrement which is not allowable to satisfy the engine requirement of the fuel consumption and emission.

In Table 1, calculation cases investigated in this study are summarized. In order to clarify the influence of gaps and clearances on turbine performance quantitatively, which are

turbine efficiency and flow capacity, following four calculations are conducted.

Table1. Computational cases for variant nozzle configurations

	Shroud / Hub clearance	Strut pins and Spindles	Full domain
Case1	NO	NO	NO
Case2	YES	NO	NO
Case3	YES	YES	NO
Case4	YES	YES	YES

In Case1, no-clearance calculation for both shroud and hub side of a nozzle vane is conducted as an ideal condition.

In Case2, in order to investigate the influence of nozzle vane's clearance only, the calculation considering nozzle clearance is conducted. At the actual turbine operating condition, it is hard to detect the nozzle axial position between the hub and shroud surface. Thus, in this calculation, nozzle clearances for shroud and hub sides are set equally, i.e. nozzle vanes are located at the center of nozzle passage as a nominal condition, and its amount is set to 2% of nozzle vane's height.

In Case3, strut pins to support the shroud piece and nozzle spindles to avoid nozzle vane's sticking and a flow leakage through vane's shaft holes are added to Case2. RHV4 turbocharger used in this study has 11 nozzle vanes and 3 strut pins, therefore the circumferential spacing of strut pins becomes non-uniform. Nozzle vanes are supported for both shroud and hub sides, thus nozzle vanes have spindles for both sides and a large pressure loss due to spindles' wake is expected.

In Case4, a full domain analysis including a nozzle link cavity and a seal structure between the turbine housing and the shroud piece are considered. Figure5 shows a detail of computational domain of Case 4. Flow leakage through the cavity between turbine housing and a shroud piece to impeller exit is sealed by the mechanical seal ring. In this region, a large pressure difference is implied, therefore it is expected to cause some flow leakage at the seal ring and we need to know its amount to estimate turbine performance precisely. In this calculation, the axial position and the radial clearance of the seal ring is determined to match the leakage flow rate with the actual measurement result obtained by the leakage test with seal ring only, and the effect of the leakage through the shroud seal on the turbine performance is investigated directly.

Furthermore, not only the nozzle link cavity but also the nozzle vanes' shafts and holes are modeled in Case4. The clearance between the nozzle shaft and its hole is set to the nominal value because the actual turbine performance test which will be compared with the computational results at the later section is conducted at the cold condition, i.e. the

turbine is driven by a compressed air, not a combustion gas, and it is expected that a heat deformation is relatively small.

Table 2 shows turbine operating conditions applied in this study. The nozzle opening is varied from the closed condition to fully-opened condition keeping the turbine pressure ratio as a constant value. The nozzle throat area at minimum flow condition is set to 20% of the fully-opened condition, and the turbine rotating speed is also changed with nozzle opening to compare the actual turbocharger performance test result. For the nozzle closed condition, rotation speed is set to 67% of the rated condition, and 83% rotation speed for the medium-opened, and the rated speed is applied to the large opened condition.

Table2. Summary of turbine operating conditions

Total to static pressure ratio	1.8 (Constant)
Nozzle opening	20~100% (Throat area ratio to max opened)
Rotational speed (Impeller tip Mach number)	67% speed (Mach 0.48) at 20~40% opening
	83% speed (Mach 0.59) at 50~70% opening
	100% speed (Mach 0.71) at 80~100% opening

Figure4 shows computational mesh used in this study. Polyhedral mesh which can be treated in Fluent is applied to reduce the number of grids keeping the enough mesh density. It takes about a half or one day calculation time to obtain a converged solution for one operation point using Intel Xeon multi-processor workstation.

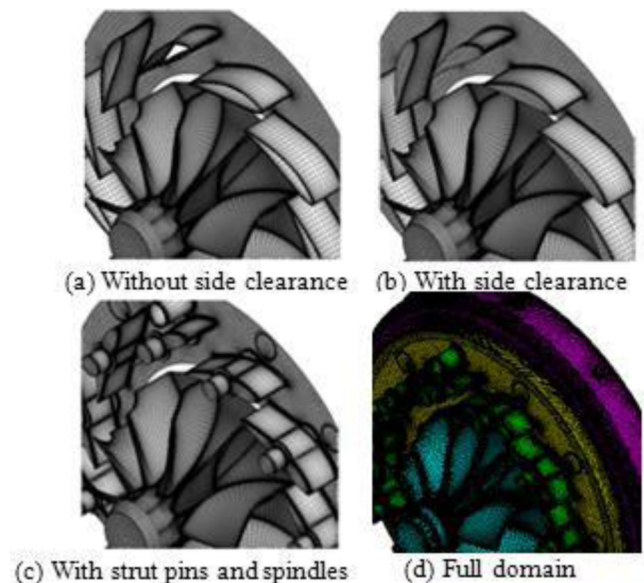


Figure4. Computational mesh of different nozzle configurations

To validate CFD analysis results of Case4 including a shroud seal and a nozzle link cavity, the additional pressure

measurement at the shroud cavity and the nozzle link cavity are conducted for several nozzle opening as the same manner as CFD analyses. Pressure measurement locations are shown in Figure 5.

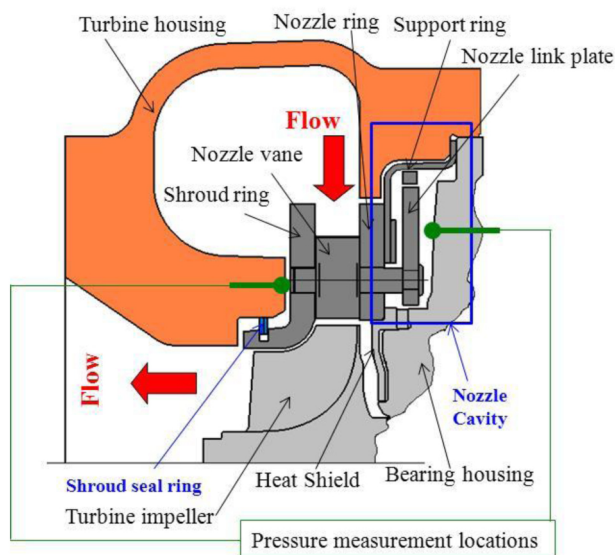


Figure5. Structures of shroud seal and nozzle link cavity considering in Case4

RESULTS AND DISCUSSIONS

CFD calculations have been conducted for four different nozzle configurations mentioned in the previous section in detail, which are with or without nozzle side clearance, strut pins and nozzle spindles, and a shroud seal and a nozzle link cavity. In this section, we would like to describe and focus on how each difference varies the turbine flow field and gives the influence to the turbine performance when the nozzle condition is varied from the closed to the fully-opened condition.

AT THE NOZZLE CLOSED CONDITION

Figure 6 shows total pressure contours of the nozzle vane-to-vane plane at 10%, 50% and 90% span-wise fraction from hub for nozzle closed condition, 25% opening. At 50% span, there is no significant difference between with and without nozzle side clearance, Case1 and the others. Regarding the strut pins, the nozzle inlet velocity becomes relatively small because of a small turbine mass flow rate, thus it cannot be found an obvious growth of strut pin's wake in Case3 which includes strut pins. Case3 also includes the nozzle spindle geometry, and the additional pressure loss due to the spindle's wake is observed because of the rapid acceleration within the nozzle passage and the further turbine efficiency deterioration is expected. This pressure gradient also makes a very strong clearance flow near the shroud and hub wall. Therefore, a large pressure loss is observed at 10% and 90% spans at nozzle closed condition. Comparing the flow fields of 10%

and 90% span of Case4, it is found that the pressure loss of 90% span is larger than that of 10% span. This indicates that the leakage from the shroud cavity to the nozzle passage occurs and the additional pressure loss due to the mixing of the main flow and the leakage flow from the shroud cavity. And also, due to the increase of the leakage flow, a turbine main passage flow rate becomes smaller compared with Case3 in which such leakage flow is not considered and a further turbine efficiency decrement is expected in Case4.

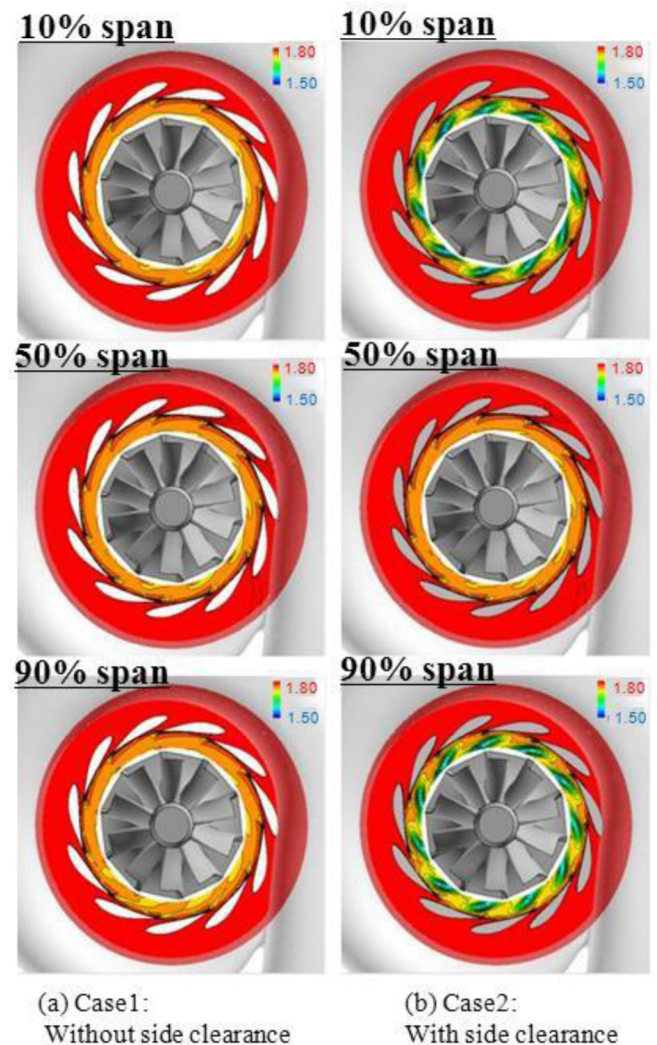


Figure 6. Comparison of total pressure distributions at the nozzle closed condition