

Investigation of Flow Structure in a Turbocharger Turbine under Pulsating Flow Conditions

Li Chen, Weilin Zhuge, Yangjun Zhang and Yazhuo Li
Tsinghua University

Shuyong Zhang
National Key Laboratory of Diesel Engine Turbocharging Technology

Copyright © 2008 SAE International

ABSTRACT

A three-dimensional numerical investigation into aerodynamic feature of the turbocharger turbine under pulsating flow conditions is conducted in this paper. Dual time stepping approach is applied to solve the unsteady Navier-Stokes equations, while the Jameson central scheme is brought in for spatial discretization, and Spalart-Allmaras turbulence model is employed in order to get good viscous resolution, accuracy and computing efficiency. The quasi-steady and unsteady performance of the turbine is given and compared. Five blade passage cross sections are chosen to analyze the structure of the secondary flow at 4 key instants. The developments of different vortex, especially the tip leakage vortex, passage vortex are discussed. The results have shown that the unsteady performance deviates substantially from quasi-steady performance, and the secondary flow structure varies tremendously under the pulsating flow conditions.

INTRODUCTION

Turbochargers have been widely applied to vehicles in order to improve the overall performance. There is a continuing upward trend in boost pressure ratio in order to achieve higher power density and better fuel consumption of an internal combustion engine (ICE). Meanwhile, emission control technologies like EGR applied to ICE to satisfy the increasingly more stringent emission standards bring a decrease in the intake air mass flow rate, thus a higher turbocharger pressure ratio is needed to compensate such a disadvantage. All these require that turbines used in turbochargers have a higher efficiency to extract more energy from the exhaust gases.

On the other hand, turbines used in turbochargers matched to reciprocating engines are under natural pulsating flow conditions, and the inlet parameters, such as pressure and mass flow rate, are continuously varying in time. This fact leads to a more complex and

unsteady flow in a turbocharger turbine. Until now, there have been some effective experimental and numerical studies of the performance of turbocharger turbines under pulsating conditions. In early days, even nowadays, restricted by test instrumentation and experimental difficulty, the performance of turbine under pulsating inlet conditions is usually evaluated based on quasi-steady assumption, which considers any instant time of the pulsating flow as steady flow [1-5]. Such a simplification will lead to an incomplete investigation into the real flow phenomenon and the actual performance of the turbine under pulsating flow condition. Therefore, it is important and necessary to further the study on the unsteady behaviors of the turbocharger turbine under actual conditions. Thanks to the development of testing technology, Dale et al presented the instantaneous testing results of a radial inflow turbine under pulsating conditions, of which rotor was accommodated in a twin-entry nozzleless volute. The work undertaken then showed occurrence of significant deviation of quasi-steady and actual performance of the turbine, and the instantaneous efficiency formed a loop around the quasi-steady efficiency [6]. After that, Winterbone et al [7], Baines et al [8], Arcoumanis et al [9] and Karamanis et al [10] also carried out experimental investigations of turbocharger turbines under pulsating flow conditions and observed distinct difference between steady and instantaneous performance as well as appearance of instantaneous efficiency loop.

As the development of CFD technology, numerical investigations are conducted more and more to derive detailed flow field inside a turbine. Palfreyman et al [11] presented one of the first successful three-dimensional numerical simulation of the flow field within the turbine stage of a pulse system turbocharger under realistic pulse conditions. Reasonable agreement has been observed between predicted and experimental results. Time accurate spectra of turbine performance and flow properties at various locations in the turbine stage were presented, as well as contour plots of velocity within a turbine passage at two critical positions during the pulse period. It was shown that the flow field within the turbine

stage is highly disturbed and influenced primarily by the pulsating inlet conditions. Blade torque, work output and loading fluctuate substantially at the frequency of the pulse. The understanding gained through this work was extended by Costall et al [12]. The performance, mass-averaged data at inlet and exit of the turbine wheel were presented. Velocity and vorticity contours in one turbine passage, as well as blade loadings at 4 key instants were analyzed.

So far the approach of evaluating the behaviors within a turbocharger turbine under steady or quasi-steady flow conditions may be handy and simple. Several significant studies, such as shown above, have been performed to determine the characteristics of turbocharger turbines under pulsating flow conditions considering the realistic operating situation in an ICE, nevertheless knowledge of flow field and mechanisms inside the turbine still remains insufficient, and demands a great deal of further research efforts in this area.

Here in this paper, an unsteady three-dimensional numerical investigation into a turbocharger turbine is carried out with a view to obtain more information of the aerodynamic feature of turbines under pulsating conditions.

NUMERICAL METHOD

The computational fluid dynamics code NUMECA has been applied to solve the Reynolds-Averaged Navier-Stokes Equations. The Navier-Stokes equations for the relative velocities in a rotating frame can be expressed as:

$$\frac{\partial \vec{U}}{\partial t} + \nabla \vec{F}_I + \nabla \vec{F}_V = \vec{Q} \quad (1)$$

Where $\vec{U}$ is the vector of the conservative variables, $\vec{F}_I$ and $\vec{F}_V$ are respectively the inviscid and viscous flux vectors:

$$U = [\rho \quad \rho W_x \quad \rho W_y \quad \rho W_z \quad \rho E] \quad (2)$$

$$\vec{F}_I = F_{Ix}\vec{i} + F_{Iy}\vec{j} + F_{Iz}\vec{k}, \quad \vec{F}_V = F_{Vx}\vec{i} + F_{Vy}\vec{j} + F_{Vz}\vec{k} \quad (3)$$

$$F_{Ii} = \begin{bmatrix} \rho W_i \\ P\delta_{xi} + \rho W_i W_x \\ P\delta_{yi} + \rho W_i W_y \\ P\delta_{zi} + \rho W_i W_z \\ (\rho E + P)W_i \end{bmatrix}, \quad \delta_{ij} = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases} \quad (4)$$

$$-F_{Vi} = \begin{bmatrix} 0 \\ \tau_{ix} \\ \tau_{iy} \\ \tau_{iz} \\ q_i + W_x \tau_{iz} + W_y \tau_{iy} + W_z \tau_{ix} \end{bmatrix} \quad (5)$$

The stress and the heat flux components are given by:

$$\tau_{ij} = \mu \left(\frac{\partial W_i}{\partial X_j} + \frac{\partial W_j}{\partial X_i} - \frac{2}{3} (\nabla \cdot \vec{W}) \delta_{ij} \right), \quad q_i = \kappa \frac{\partial T}{\partial X_i} \quad (6)$$

The source term vector $\vec{Q}$ containing contributions of Coriolis and centrifugal forces is given by:

$$Q = \begin{bmatrix} 0 \\ -\rho [2\vec{\omega} \times \vec{W} + (\vec{\omega} \times (\vec{\omega} \times \vec{r}))] \\ \rho \vec{W} \cdot \vec{\nabla} \left(\frac{1}{2} \omega^2 r^2 \right) \end{bmatrix} \quad (7)$$

In the equations above, E is total energy per unit volume; $\vec{F}$ is flux vector; P is pressure; T is temperature; $\vec{W}$ is relative velocity; κ is heat conductivity; μ is viscosity; ρ is density τ is stress; ω is rotational speed.

Sparlart-Allmarasone equation turbulence model [13] has been applied after carefully compared to other models in order to get good accuracy and computing efficiency. The S-A model's predictions are far superior to those of algebraic models for separated flows, and the differential equation presents no serious numerical difficulties. Results of experience to date indicate that the S-A model is an excellent engineering tool for predicting properties of turbulent flows [14]. The flow solver is based on a cell centred finite volume approach, associated with Jameson's center scheme as space discretization together with an explicit four stage Runge-Kutta time integration method. A variety of convergence acceleration techniques, such as implicit residual smoothing and full multigrid algorithm, are employed.

The dual time stepping approach is applied for the unsteady calculations, which consists in adding to the time dependent Navier-Stokes equations pseudo-time derivative terms. At each physical time step, a steady state problem is solved in a pseudo time and all available acceleration techniques such as multigrid, local time stepping and implicit residual smoothing can be applied. Then for unsteady flow simulations the Reynolds-Average Navier-Stokes equations can be expressed as:

$$\frac{\partial}{\partial t} \iiint_V U dV + \iiint_V \frac{\partial U}{\partial \tau} dV + \iint_S \vec{F} \cdot d\vec{S} = \iiint_V ST dV \quad (8)$$

Where t is the physical time and τ is the pseudo time, ST are the source terms.

At the inlet boundary, the magnitude of absolute velocity, static temperature and direction of the absolute velocity vector are imposed. The magnitude of absolute velocity, which is derived from mass flow rate, pulsates at a frequency of 75HZ corresponding to a four-stroke, six cylinder engine running at 1500 rpm, while has a average value of 295 m/s. The magnitude of inlet absolute velocity varies from 206.5 m/s to 383.5 m/s with sinusoidal function in a simplified consideration, which equal 70% and 130% of the average value respectively. The direction of the absolute velocity vector is obtained from the one-dimensional flow analysis of the volute [15]. The simulative pulsation of absolute velocity at the inlet is shown as Figure 1, in which the point on the curve designates each physical time step during the computation. In order to capture enough flow phenomena in one period, the physical time step is specified as 3.333×10^{-4} seconds in this computation, that means compute 40 time steps per period.

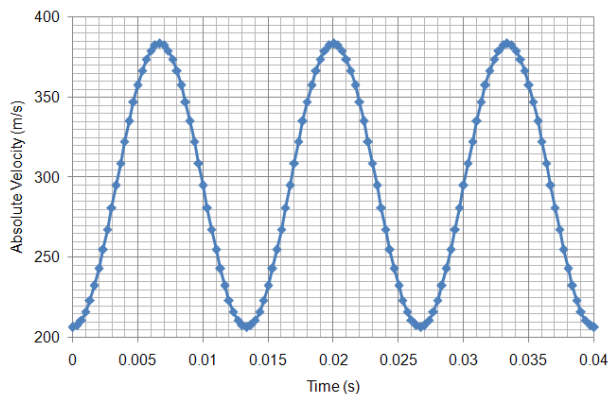


Figure 1. Absolute velocity imposed at the inlet

At the outlet boundary, the static pressure is assumed to be uniform in the tangential direction and the static pressure at a specific radius is given. The pressure distribution in the outlet section is then found by integrating the following equation:

$$\frac{\partial P}{\partial r} = \rho \cdot \frac{V_\theta^2}{r} \quad (9)$$

The remaining dependent variables on the outlet boundary are obtained from the interior field through extrapolation. Adiabatic and no-slip solid wall boundary conditions are imposed.

COMPUTATIONAL GRID

The mesh has a structured hexahedron multi-block topology, and consists of a total of 7 mesh blocks

leading to an overall 397975 grid points. H-topology is used in the inlet, outlet and main flow passage domains while the skin block around the blade uses C-topology. The grid is refined at the near-wall, leading and trailing edge. The tip clearance gap is resolved with 13 radial grid points, and the y^+ value is controlled less than 5 in order to properly capture the high gradients inside the boundary layer.



Figure 2. Geometry of the mixed flow turbine

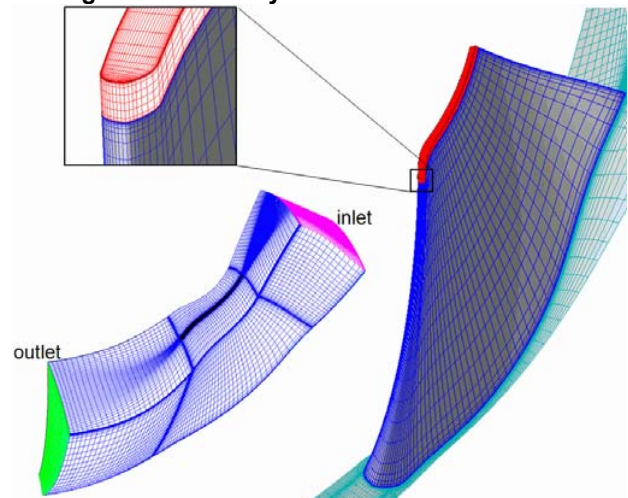


Figure 3.a Blade passage and tip gap

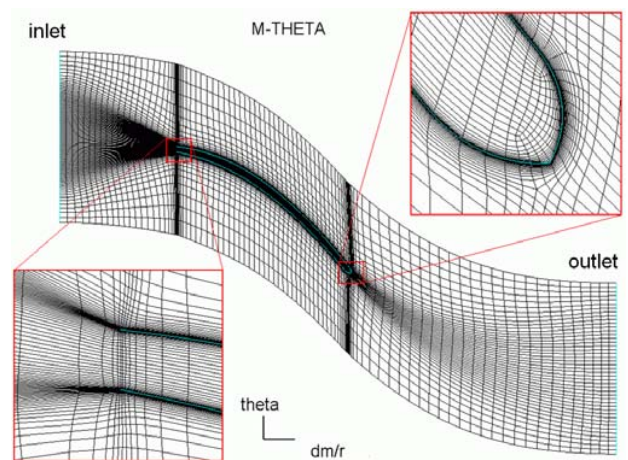


Figure 3.b Shroud in blade to blade view

Figure 3. Computational grids of the mixed flow turbine