International Journal of
Science and Engineering Investigations vol. 1, issue 3, April 2012
ISSN: 2251-8843

Pulsating Flow Analysis in a Small Turbocharger Turbine

Tabatabaei H.R.[1], Boroomand M.[2], Taeibi-Rahi M.[3]
[1]Department of Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran
[2]Department of Aerospace Engineering, Amirkabir University of Technology, Tehran, Iran
[3]Department of Aerospace Engineering, Sharif University of Technology, Tehran, Iran
([1] hrtabatabaei@ iaukashan.ac.ir, [2]boromand@aut.ac.ir, [3] taeibi@sharif.ir)

Abstract- Possibilities and limitations of 1D unsteady and 3D steady flow simulations in the vane-less turbocharger turbine of a 1.7 liter SI engine is presented experimentally and numerically. To validate the results of simulation by experimental results, a test setup of the turbocharged engine on dynamometer is prepared and various performance parameters are measured at the different engine speeds. The complement form of the volute and rotor vanes are modeled. Mesh studies are done and the best them is selected and qualified by calculation of the dimensionless wall function. Two ways of modeling the rotating wheel, Multiple Frames of Reference (MFR) and Sliding Mesh (SM), are considered. Finally, the variations of performance turbine parameters are studied by pulses frequencies surveying and the present paper is concluded that the 3D unsteady flow is needed to get a good result of the turbine modeling. The results obtained here will be used in simulating three dimensional and unsteady compressible flows in turbocharger turbine which will be reported very soon.

Keywords*-Computational Fluid Dynamics; Unsteady Flow; Compressible flow; Rotating Fluids; SI-Engine; Simulation.*

I. INTRODUCTION

One dimensional modeling often good results and acceptable but unfortunately many of the details cannot be used by a 1D modeling observed. The mean values of one dimensional result can be used for boundary conditions values in the three dimensional flow simulations.

The Multiple Frames of Reference (MFR) and Sliding Mesh (SM) are considered for modeling the rotating wheel. The MFR technique uses a coordinate system that rotates with the wheel turbine and NS equation is modified to take into account the coriolis and centrifugal forces. This is done by adding the appropriate source term in the momentum equation. In the SM technique, one part of the mesh is moving or rotating in relative to the stationary part which will sets demands on both the cell size in sliding region and the time step. Information between two parts is exchanged through mass conserving interpolation. At the sliding interface, the connectivity for cells on either side of the interface change at each time step. The time step must be small enough to ensure that cells on both sides of the sliding interface do not pass each other completely during one time step.

The Investigations of pulsating flow performance of radial turbines were started by Wallace and Blair [1], Dale and Watson [2] and Yeo and Baines [3]. Winterbone, Nikpur and Alexander [4] were modeled a vane-less radial turbine but the results not validate with experiments data. Chen, Hakeem, and Martinez-Botas [5] improved the model and showed that the unsteady model is better capable of predicting the turbine flow behavior than a quasi-study model. Elrich [6] performed extensive measurements on a six cylinder diesel engine to analyze the on engine turbine performance. The experiments showed that the flow velocity, temperature and pressure within the exhaust manifold and turbine propagate with different velocities. King [7] present parameterized model in GT-Power, but it appears to still need significant development before it can be a valuable tool to an engine designer. Lam, Roberts and McDonnell [8] have done a complete 3D CFD calculation of unsteady flow in a radial turbocharger turbine. They showed that the pulse amplitude is damped quite heavily through the volute and nozzle vanes. Costall, Szymko, Martinez-Botas, Filsinger and Ninkovic, [9] and Rajoo and Martinez [10] are studied on the instantaneous efficiency for mixed flow turbine on pulsating flow and present a good correlation between the pressures measured at different locations of the turbine. Winkler and Angstrom [11] have performed engine simulations in the one dimensional fluid dynamics for transient operation ad validated with engine experiments. He showed that it is difficult to predict the performance of the gas exchange system and its components, especially the turbine performance. Elrich [12] performed extensive measurements on a six cylinder diesel engine to analyze the on engine turbine performance. The experiments showed that the flow velocity, temperature and pressure within the exhaust manifold and turbine propagate with different velocities.

This paper is presented the turbine performance under 1D unsteady and 3D compressible steady flow simulation. The results are validation by experiments. The MFR and SM techniques are used to modeling of the rotating wheel. Results show that the affect of pulsating flow are indispensable. The pulse frequency, especially in low speed of engine, can be impeded a good prediction of the turbine efficiency.

II. Basic Theory

A. Governing equations

The instantaneous equations of mass, momentum and energy conservation can be written as follows in a stationary frame:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \boldsymbol{U}) = 0 \tag{1}$$

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla.(\rho \boldsymbol{U} \otimes \boldsymbol{U}) = -\nabla p + \nabla.\boldsymbol{\tau} + \boldsymbol{S_M} \tag{2}$$

$$\frac{\partial (\rho h_0)}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho \boldsymbol{U} h_0) = -\nabla.(\lambda \nabla \mathrm{T}) + \nabla.(\boldsymbol{U}.\boldsymbol{\tau}) + \boldsymbol{U}.\boldsymbol{S_M} \tag{3}$$

Where the stress tensor, $\boldsymbol{\tau}$, is related to the strain rate by:

$$\boldsymbol{\tau} = \mu \left[\nabla \boldsymbol{U} + (\nabla \boldsymbol{U})^{\mathrm{T}} - \frac{2}{3} \delta \, \nabla.\boldsymbol{U} \right] \tag{4}$$

Where, ρ is density, t is time, $\boldsymbol{U}$ is velocity vector, p is pressure, T is temperature, h_0 is the total enthalpy, δ is Kronecker Delta function, λ is thermal conductivity, μ is viscosity, $\boldsymbol{\tau}$ is stress vector, ∇ is gradient operator and ⊗ dyadic operator. The term $\boldsymbol{S_M}$ represents momentum sources.

B. Real Gas Properties

Cubic equations of state are a convenient means for predicting real fluid behavior. They are highly useful from an engineering stand point because they generally only require that the user know the fluid critical point properties. They are called cubic equations of state because, when rearranged as a function volume they are cubic in volume. Three versions of cubic state equations are available. The Aungier form of Redlich-Kwong formulation is used in this paper. It can be written as:

$$p = \frac{RT}{v - \frac{0.0866 RT_c}{p_c} + a} - \frac{0.4272 R^2 T_c^2}{p_c} \left(\frac{T}{T_c} \right)^{-n} \frac{1}{v(v+b)} \tag{5}$$

$$a = \frac{RT_c}{p_c + \frac{0.42747 R^2 T_c^2}{p_c v_c (v_c + b)}} + \frac{0.08664 RT_c}{p_c} - v_c \tag{6}$$

$$n = 0.4986 - 1.2735 \left[log_{10} \left(\frac{p_v}{p_c} \right) - 1 \right] - 0.4754 \left[log_{10} \left(\frac{p_v}{p_c} \right) - 1 \right]^2 \tag{7}$$

Where v is the specific volume, the vapor pressure, p_v is calculated at $T=0.7T_c$. The subscript c is mentioned to critical point.

C. Wall Function

The wall-function approach in this paper is an extension of the method of Launder and Spalding.
The log-law region, the near wall tangential velocity is related The logarithmic relation for the near wall velocity is given by:

$$u^+ = \frac{\mathrm{U}}{u_\tau} = \frac{1}{\kappa} \ln y^+ + C \tag{11}$$

$$y^+ = \frac{\rho \, \Delta y \, u_\tau}{\mu} \tag{12}$$

$$u_\tau = \left(\frac{\tau_w}{\rho} \right) \tag{13}$$

$where, u^+$ is the near wall velocity. the friction velocity, u_τ, is the known velocity tangent to the wall at a distance of Δy from the wall, $y^+$is the dimensionless distance from the wall, τ_w is the wall shear stress, κ is the Von Karman constant and C is a log-layer constant depending on wall roughness (natural logarithms are used). The flow for y^+ less than 11.36 is laminar and the y^+ greater than 200 is not recommended because it shows that the grids are coarse.

D. Turbulence Modeling

One of the most prominent turbulence models, the k-ε model, has been implemented in most general purpose CFD codes and is considered as a standard model in many flow simulation cases due to its stability, numerically robustness and its well established regime of predictive capability. Previous investigations have shown that his model with all its capabilities may not be suitable for applications such as rotating fluid flow in a turbine where sudden change in strain may occur. In this regard, the RNG k-ε model recommended by some researchers has been used.

E. Solution Strategy

Coupled solver solves the hydrodynamic equations (for u, v, w and p) as a single system. This solution approach uses a fully implicit discretization of the equations at any given time step. For steady state problems, the time-step behaves like an 'acceleration parameter', to guide the approximate solutions in a physically based manner to a steady-state solution. This reduces the number of iterations required for convergence to a steady state, or to calculate the solution for each time step in a time-dependent analysis.

III. Experimental Setup

The tests are performed on a 1.7 liter in-line four cylinder turbocharged SI engine and various performance parameters are measured at 12 different engine speed. Main specifications of engine are summarized in Table I.
A vane-less turbocharger turbine is completely modeled. The air gap between turbine blades and shroud are neglected. Table II show the main geometry and acceptable model mesh size of turbocharger turbine.

TableI. Main Geometry and performance parameters of SI engine

Displaced Volume	1650 cm^3
Stroke length	85 mm
Bore Diameter	78.6 mm
Compression Ratio	10.5:1
Number of Cylinders-Valves	4-16
Maximum Power	110 kw
Maximum torque	215 N.m
Maximum outlet of gas temperature	1250 K

Table II. Main geometry parameters and the acceptable model mesh size

Rotor tip mean diameter	45.3 mm
Number of rotor blades	11
Number of nozzle vanes	Vaneless
Blade width at inlet	6.86 mm
Mesh size (tetrahedral cell)	1.8×10^6

Fig. 1 shows the scheme of the test cell configuration. A PC based system is used for control and measurement. The measurement method system takes both analogue and digital input. Analogue input is measured by using either a fast system for crank angle resolved data (1MHz) or by using a slower time averaged system (1 Hz). Pressure transmitters are used to measure the relative (gauge) and absolute pressures in gases.

The measuring device for the transmitter is a piezo-resistive element, range 0-600 kpa. Temperatures are measured by thermocouple type k, range 200-1300 ˚C. The turbo speed is measured by a speed sensor that worked with the eddy current measurement technique, range 0-400000 rpm.
The Fig. 2 and Fig. 3 show the turbocharger engine test cell.

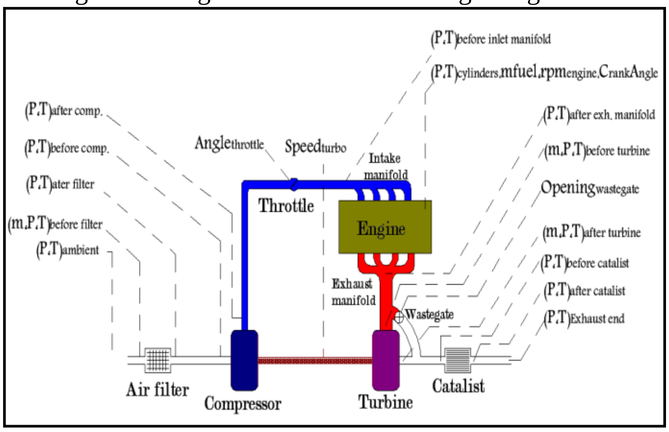

Figure 1. The scheme of test cell configuration


Figure 2. The turbocharger on the engine

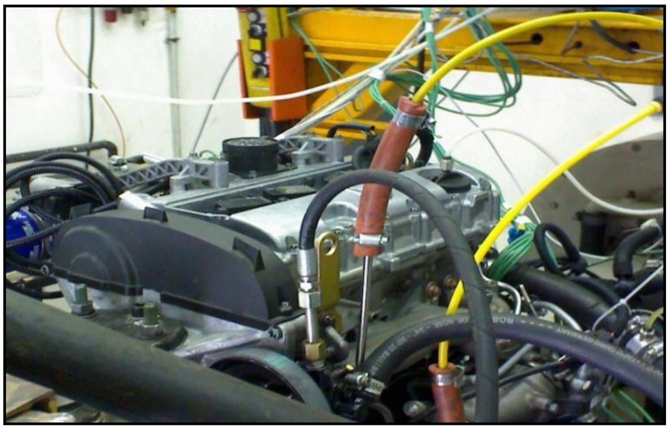
Figure 3. The test bench of turbocharger engine

IV. MODEL SETUP

A. Modeling

Two different simulations, the 1D unsteady flow simulation for turbocharged SI engine and the 3D steady compressible flow modeling for turbocharger turbine are done. Fig. 4 shows the scheme of 1D simulation.

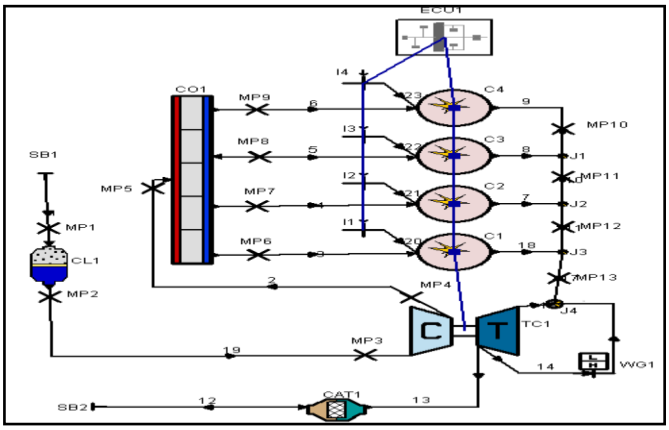

Figure 4. The scheme of 1D simulation

The turbine is comprised both solid (turbine blades) and fluid regions. The gap between the turbine vanes and the shroud is neglected. The three dimensional model of turbine casing and its vanes are shown in Fig. 5.

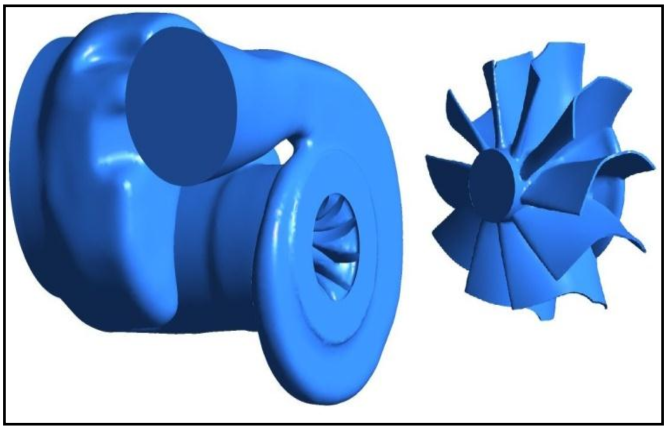

Figure 5. Three dimensional modeling of turbine

B. Grid Study

In the proposed simulation, the unstructured tetrahedral grids due to its more adaptation to the geometry are used. Fig. 6 shows the unstructured tetrahedral meshes on the turbine. For different number of cells, the amount of the mass flow rate from the turbine is calculated and is shown that the mass flow is becoming constant as the number of cells is increased to more than 1.5×10^6 so the specified 1.8×10^6 tetrahedral meshes are used to discrete modeling the complete vane-less turbocharger turbine. In this way, the Mesh size is varied between 0.1 mm to 1 mm.

C. Near wall

In near-wall regions, boundary layer effects give rise to velocity gradients which are greatest normal to the face. Computationally-efficient meshes in these regions require that the elements have high aspect ratios. If tetrahedral are used, then a prohibitively fine surface mesh may be required to avoid generating highly distorted tetrahedral elements at the face. In this regard, five inflations layers near the walls are defined.

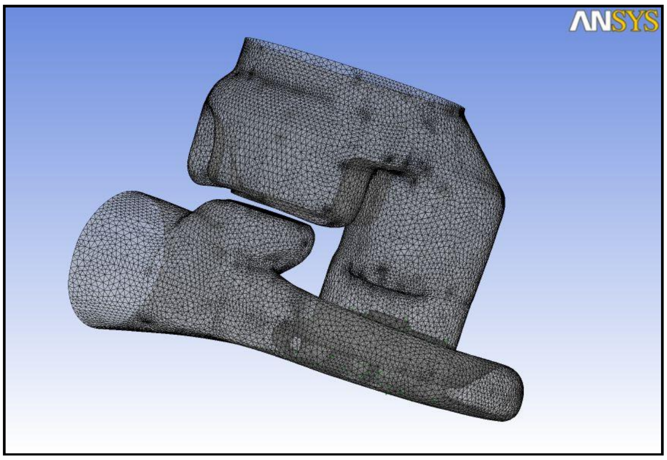

Figure 6. The turbine Unstructured tetrahedral meshes

D. Boundary conditions

In the case of steady three-dimensional flow simulation of the turbine, a mean value of mass flow and a mean value of stagnation temperature are used as inlet boundary conditions. Besides, mean value of static pressure is used as outlet boundary condition. The inlet flow is taken to be turbulent and RNG k-ε turbulence modeling is used. For the turbine wheel, adiabatic wall condition is applied.

V. RESULTS

First, the time-cost reduction is considered by the parallel processing study by a Dual cores CPU. Two ways of modeling the rotating wheel, the Multiple Frames of Reference (MFR) and the Sliding Mesh (SM), are used and compared. It shows that the SM technique is required more time than the MFR technique about 25 percent. The advantages are detailed in Table III. As will be shown later, the SM technique is more accurate than the MFR technique.

Table III. Comparing between the MFR and SM techniques

Type of Processor	**No. of active cores**	**Ave. time for one step (sec)**		**Ave. value for one engine speed (hr)**	
Dual Cores 2 GHz 4 MB Ram		***MFR***	***SM***	***MFR***	***SM***
	1	631	760	80	99
	2	380	480	48	60

The turbine characteristic map supplied by the manufacturer is contained only discrete points which usually measured under steady flow conditions. It often covers only a small range of operation. By using three-dimensional simulation, without the need for time-consuming and expensive tests, the turbine characteristic curve can be plotted. Fig. 7 shows the turbine characteristic map obtained by three-dimensional steady flow simulation. By using simulation, the characteristic curve parameters are calculated at five points. This work is done at seven different turbine reduced speeds. Interpolation and extrapolation techniques are carried out to help to complete curves.

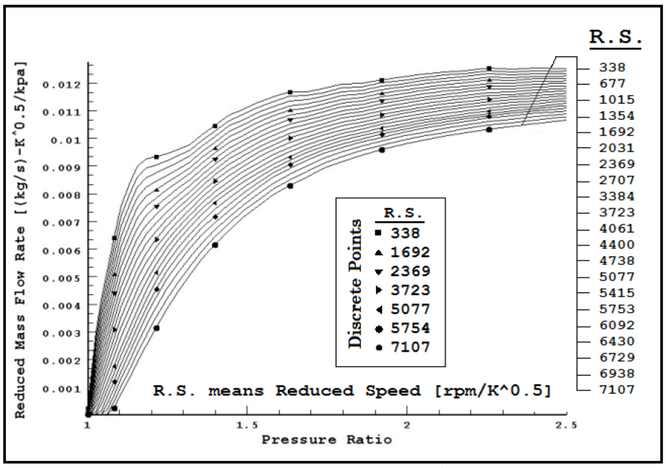

Figure 7. Characteristic curves of turbine

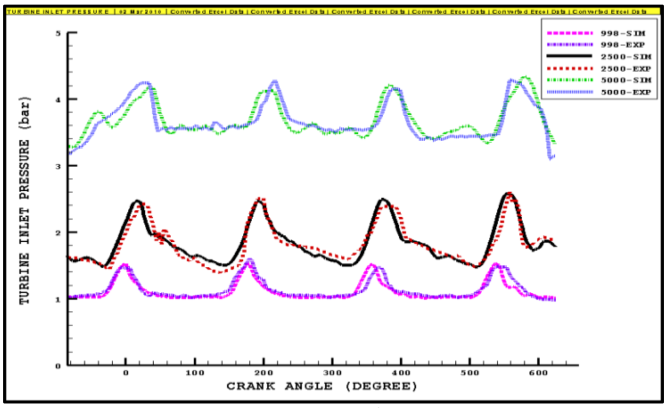

Figure 8. Instantaneous inlet pressure sof turbine vs. crank angle at three different speeds of engine using 1D simulation and test

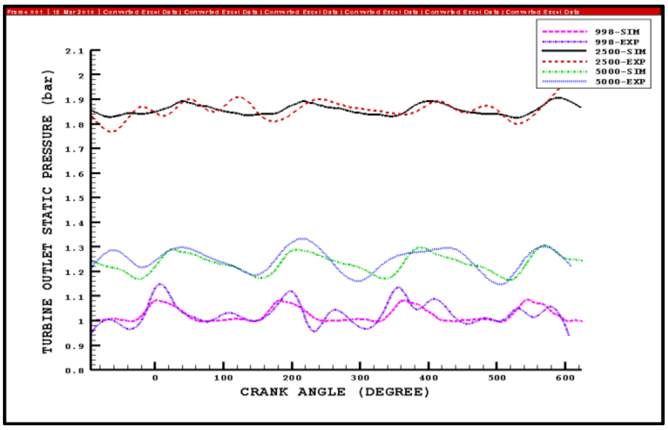

Figure 9. Instantaneous outlet turbine pressures vs. crank angle at three different speeds of engine using 1D simulation and test

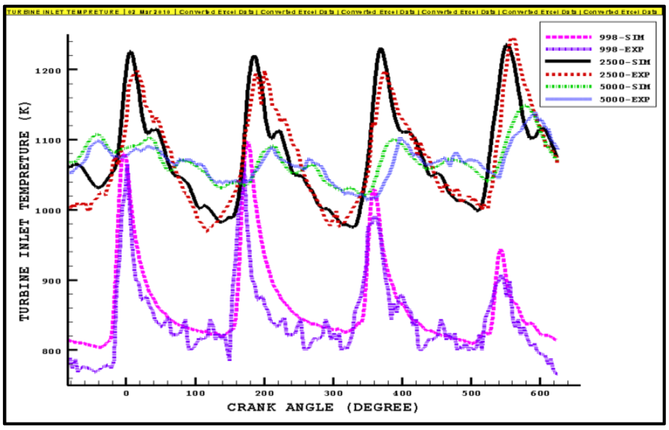

Figure 10. Instantaneous inlet temperatures of turbine vs. crank angle at three different speeds of engine using 1D simulation and test

The instantaneous of turbine inlet and outlet pressures and turbine inlet temperature are shown in Figs. 8-10.

A good agreement is observed between experimental values and simulation results.

The mean values of turbine inlet pressure, turbine outlet temperature and turbine pressure ratio for engine speeds between 1000 to 5500 rpm are shown at Figs. 11-13.

The figures show that the mean values of 1D pulsating flow in both experimental and computational approach are fitted fairly on each other, but the amounts of the mean values of three dimensional simulations are obviously a little bit different. The comparing between instantaneous and mean values of turbine performance parameters shows that the effect of three dimensional flows is less important than the effect of pulsating flow.

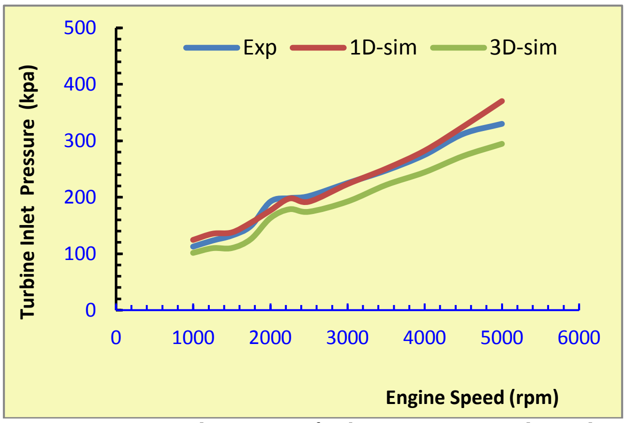

Figure 11. Average inlet pressure of turbine vs. engine speed at twelve different speeds of engine using 1D simulation and test

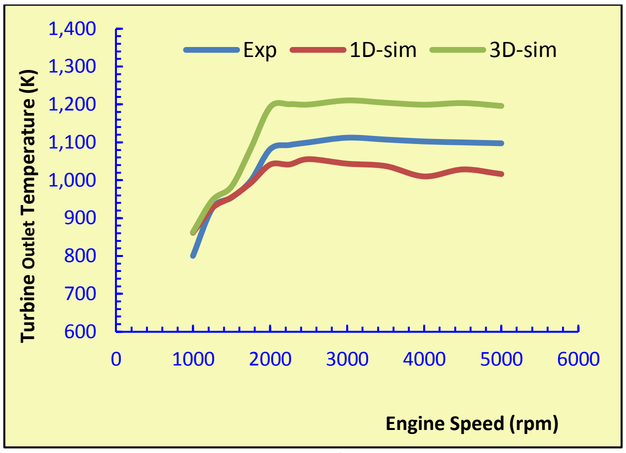

Figure 12. Average outlet temperature of turbine vs. engine speed at twelve different speeds of engine using 1D simulation and test

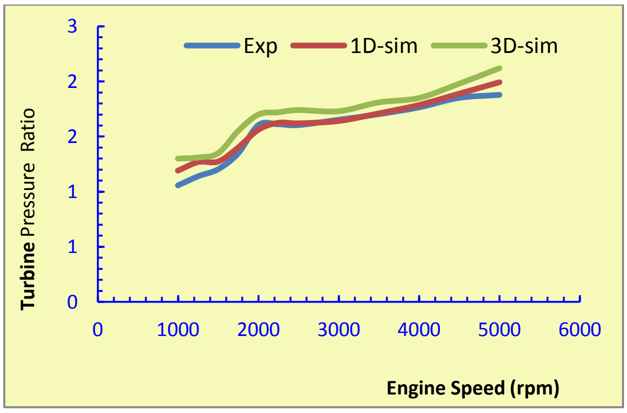

Figure 13. Average pressure ratio of turbine vs. engine speed at twelve different speeds of engine using 1D simulation and test

Fig. 14 shows the turbine characteristic map which is supplied by manufacturer and the results of 1D and 3D simulations and experimental values.

It shows that the 1D simulation and experimental results are matched and show a good agreement between them but the values of 3D steady flow simulation are signally differed. The significant difference between experimental values and simulation results is explained by the effect of pulsating flow.

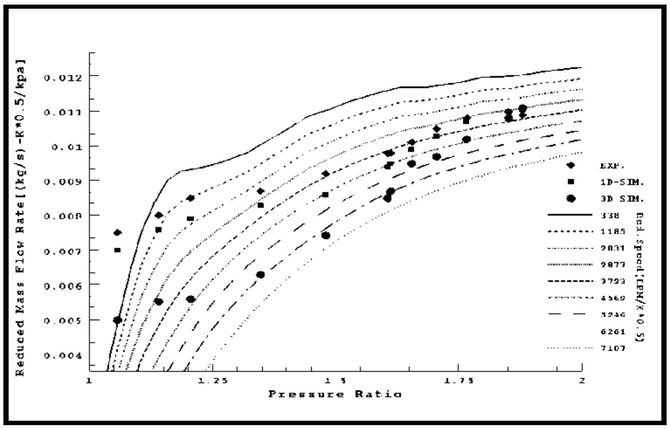

Figure 14. Reduced mass flow rate vs. pressure ratio using 1D and 3D simulation and test

Fig. 15 shows the turbine inlet pressures and their mean values versus time for two different engine speeds. It shows that using of the mean value is not efficient. On the other hand, the averaging method of time variations values is very important.

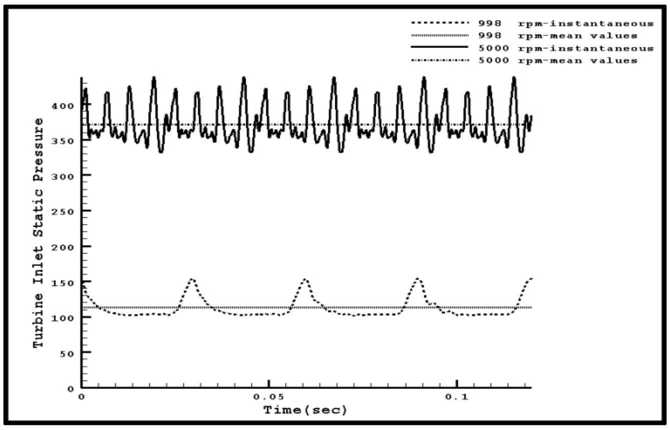

Figure 15. Instantaneous and mean value of turbine inlet pressure vs. time

Turbine efficiency is a basic parameter for turbine performance analysis.

1D and 3D simulations values and experimental results of turbine efficiency at three different engine speeds are summarized in Table IV.

The following results are noticeable:

- At all of cases, the values of turbine efficiency are increased proportional to engine speed. The greater efficiency of turbine can be due to higher turbine inlet temperature of engine at higher speeds.
- The worst results, for turbine efficiency, at all engine speeds, are depended to 3D simulation whereas the results of 1D simulation with comparing of experimental values are a little better. A true guess for the reason of this consequence may be explained by the effect of pulsating flow. The 3D unsteady compressible flow simulation will confirm the above supposition.
- The efficiency values differences obtained between the 1D simulation and the experiments are varied from 16.4 percent at the lowest engine speed to 4.2 percent at the highest engine speed. Similarity, when the 3D simulation and the experimental values are compared, these differences are varied between 19.7 and 6.8 percent for the MRF technique and are varied between 17.7 and 6.1 for the SM technique. The reason can be explained that the frequency of pulsating flow is increased with engine speed and the theory of 1D flow is more reliable as the pulsating flow is act similar to steady flow.
- For 3D steady flow simulation using a sliding mesh technique is more accurate but more time consuming than MRF technique.

Table IV. Comparing between the values of efficiency using the MFR and SM techniques at three different engine speeds

Turbine Efficiencies (%)				
Engine Speed (rpm) / Case		1200	3000	5500
Experimental		31.1	55.5	57.3
1D-Simulation		35.0	61.0	60
3D-Simulation	***MFR***	36.1	61.4	61.5
	SM	35.4	60.8	61.1

One of the major disadvantages of 1D simulation is that cannot provide any information about the flow behavior inside the turbine while three-dimensional approach could well show the behavior of fluid. Figure 16 shows the velocity vectors of the turbine flow. Figs. 17- 18 show the total temperature contours of flow in the turbine casing and the turbine vanes.

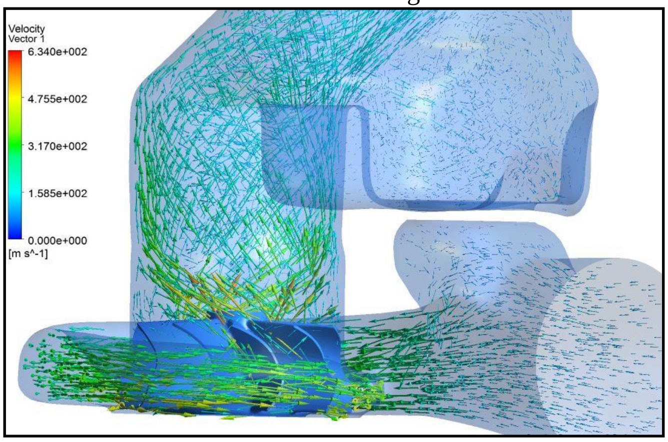

Figure 16. the velocity vector in the turbine flow

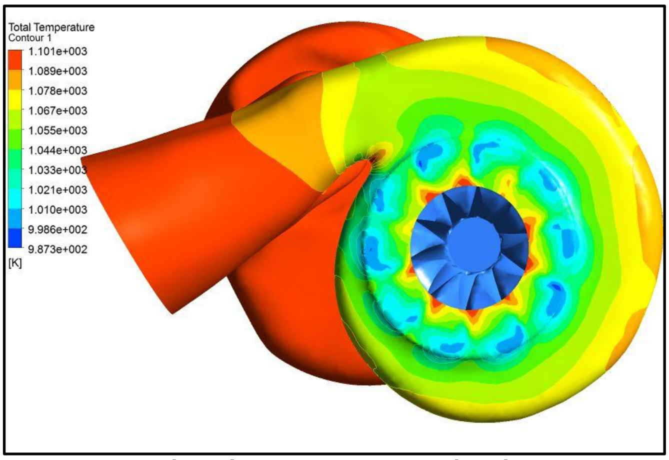

Figure 17. The total temperature contours in the turbine casing

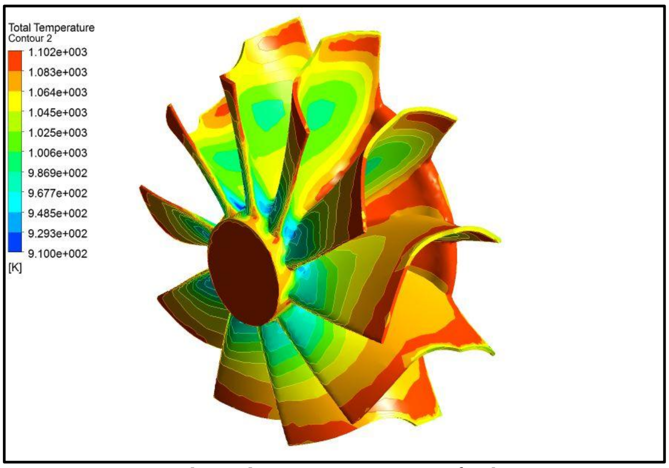

Figure 18. The total temperature contours of turbine vanes

VI. CONCLUSION

The experiments are expensive and nerve-racking. In addition, some of the pulsating flow parameters such as the instantaneous temperature and mass flow rate often could not be directly measured. Although, the one dimensional modeling often good results and acceptable but unfortunately many of the details cannot be used by a 1D modeling observed. The studies are shown, as it reported in this paper, the effect of pulsating flow could not be ignored but isn't reliable alone and also the 3D steady flow simulation are not reliable to get a good visibility of turbocharger turbine performance parameters since the effect of pulsating flow is ignored. Thus both the three dimensional flows and inlet pulsating flow are important and should be analyzed simultaneously to more realistic values of the turbine parameters are calculated so the 3D unsteady flow simulation is necessary.

The results obtained in the present report will be used in simulating three dimensional unsteady compressible flows in turbocharger turbine which will be reported very soon.

REFERENCES

[1] Wallace, F., J., Blair, G, P, "The pulsating-flow performance of inward radial-flow turbines," 65-gtp-21 ASME, 1-19, 1965

[2] Dale, A., Watson, N., "Vaneless diffuser turbocharger turbine performance," Proc. Instn. Mech. Engrs. Part C. C110/86, 65-76, 1986.

[3] Baines, N., C., Yeo, J., H., "Pulsating flow behaviour in a twin-entry vaneless radial-inflow turbine," IMechE Paper C405/004/90, 1990

[4] Winterbone, D.E., Nikpur, B., Alexander, G., I., "Measurement of the performance of a radial inflow turbine in conditional steady and unsteady flow," ImechE Paper C405/015, 1990

[5] Chen, H., Hakeem, I., Martinez-Botas, R., F., "Modelling of a turbocharger turbine under pulsating inlet conditions," IMechE Proc Instn Mech Engrs, 397-408, 1996

[6] Ehrlich, D., A., "Characterization of unsteady on-engine turbocharger turbine performance," Ph.D. Thesis, Purdue University, 1998

[7] King, A., J., " A turbocharger unsteady performance model for the GT-Power internal combustion engine simulation," Ph.D. Thesis, Purdue University, 2002

[8] Lam, J, K-W, Roberts, Q, D, H, McDonnell, G, T, "Flow modeling of a turbocharger turbine under pulsating flow," ImechE Paper C602/025/2002, 2002

[9] Costall, A., Szymko, S., Martinez-Botas, R., F., Filsinger, D., Ninkovic, D., "Assessment of unsteady behavior in turbocharger turbines," ASME Turbo Expo 2006, GT2006-90348, 2006

[10] Rajoo, S., Martinez-Botas, R. "Experimental study on the performance of a variable geometry mixed flow turbine for automotive turbocharger," Proc. of 8th Int. Conf. on Turbochargers and Turbocharging, Instn. Mech. Engrs., 183-192, 2006

[11] Winkler, N., Angstrom, H., E., "Simulations and Measurements of a Two-Stage Turbocharged Heavy-Duty Diesel Engine Including EGR in Transient Operation," SAE Technical Paper 2008-01-0539, 2008

[12] Ehrlich, D. A., "Characterization of unsteady on-engine turbocharger turbine performance", Ph.D. thesis, Purdue University, 1998.