



Steady Simulation of the Flow inside the Internal Combustion Engine Turbocharger's Turbine

M. Farajpoor¹, S. M. Agha Mirsalim^{1*}, S. S. Alavioun²

¹Department of Mechanical Engineering, Amirkabir University of Technology, Tehran, Iran

²Department of Mechanical Engineering, Khaje Nasir University of Technology, Tehran, Iran

ABSTRACT: Turbocharger systems, can increase volumetric efficiency and decrease fuel consumption and emissions of an engine due to compressing the entering air to the engine. Flow characteristic inside turbine is sophisticated and several phenomena like flow separation and high turbulent flow can occur inside turbine. Determining exact performance behavior of turbine can alter the matching process of the turbocharger with the engine. The main goal of this research is 3D and steady simulation of the flow inside turbocharger's turbine and analysis of the performance behavior of turbine under different working conditions. To this end, 3D flow inside turbine including volute, rotor, diffuser and wastegate passage is investigated steadily. Validating simulation results by experimental results shows that there exist 3 to 9 percent of error between these. In order to control the rotational speed of turbine, turbocharger is equipped with wastegate. By measuring the exact amount of wastegate opening in different working conditions of turbine on a test cell, flow simulation inside turbine is accomplished in different wastegate openings and the effect of wastegate opening on the turbine performance and isentropic efficiency is investigated. The results shows that opening of wastegate can reduce isentropic efficiency and power produced by the turbine significantly.

Review History:

Received: 3 Mar. 2018

Revised: 13 Jul. 2018

Accepted: 17 Jul. 2018

Available Online: 24 Jul. 2018

Keywords:

Turbocharger

Turbine

3D simulation

Wastegate

Isentropic efficiency

1- Introduction

The primary intentions of turbocharging an engine are increasing the power output and efficiency of the engine and hence reducing engine's fuel consumption from one point of view and decreasing pollutant emissions of the engine from other hand by compressing the entering air to the engine. In a typical turbocharger system which is consisted of a turbine and a compressor, the hot gases exiting the engine from exhaust manifold, hit the turbine's blades and rotate the impeller of the turbine. These gases exit the turbine outlet after transferring a major part of their energy to the impeller. The turbine and the compressor are connected via a common shaft, so rotation of the turbine's wheel caused by passing exhaust gases, rotate the compressor blades which causes the inlet air to the engine to be compressed.

Generally the boost pressure of the engine should be controlled in some way to avoid damages to the engine and the turbocharger system itself. One of the most common way to control the boost pressure is controlling the turbine's rotational speed with the aid of a valve called wastegate. The wastegate controls the amount of opening a bypass port in the turbine. By opening the wastegate some of the entering hot gas to the turbine, passes through this passage and bypasses the turbine's blades and hence opening of the wastegate causes the rotational speed of the turbine to decrease.

Extensive researches have been devoted to the field of turbocharger performance analysis [1-4]. Galindo [5] investigated the 3D flow inside a variable geometry radial

turbocharger turbine. It was shown in this research that the turbine's mass flow calculated using sliding mesh method is around 5 percent more accurate than the Moving Reference Frame (MRF) method at high pressure ratios. Tabatabaie [6] performed a research on comparing the 1D simulation and 3D simulation of the flow inside a turbocharger turbine and validated the simulation results with experimental engine test data. It was shown that 3D simulation results are more reliable than 1D ones.

In the present research, the 3D steady flow inside the turbocharger's turbine is simulated using Computational Fluid Dynamics (CFD) at different working points. For this purpose the whole turbine including turbine's housing, impeller, exit nozzle and wastegate passage is modelled and introduced to the solver as the computational domain. Also one of the distinguished aspect of this paper is the investigation the effect of wastegate opening on the turbine's behavior in different working conditions. Also in order to validate the simulation results, the performance of the turbocharger's turbine is examined experimentally on an engine test cell.

2- Methodology

In order to simulate the flow inside the turbine, the fluid domain is separated to two different sub-domains. The stationary domain includes the turbine's housing, outlet nozzle and wastegate passage and the rotating domain includes the volume of fluid around the impeller. These two sub-domains should be connected together via an interface. Sliding mesh and MRF methods are two common type of methods for connecting a rotating domain to a stationary one. In MRF approach, the

* Corresponding author's email: mirsalim@csr.ir



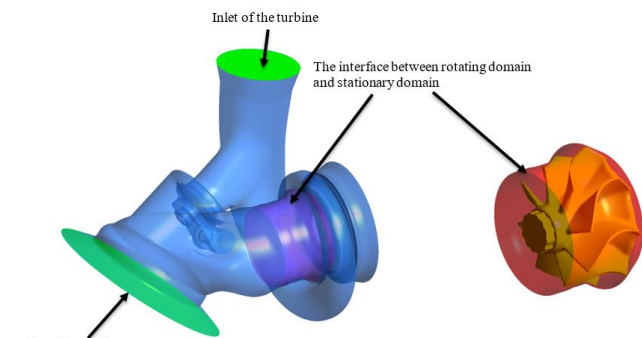


Fig. 1. Domains of fluid in turbine

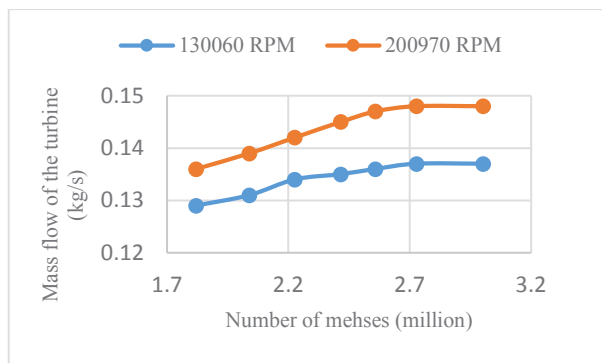


Fig. 2. Mesh independency analysis

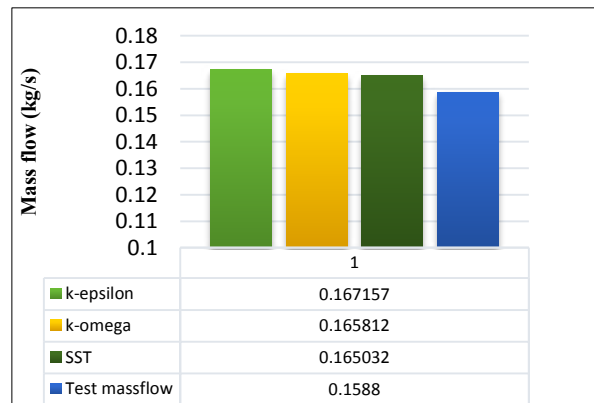


Fig. 3. Mass flow of the turbine calculated with three turbulence models

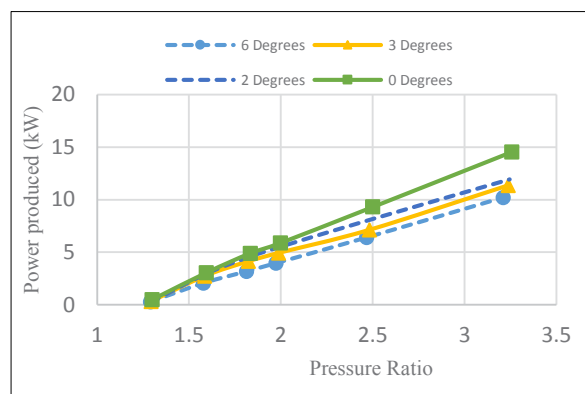


Fig. 4. Power produced by turbine in different wastegate openings

effect of rotation of a domain is considered by introducing two extra source terms into the fluid flow equations [7]. In sliding mesh method, in every time step of solving process, the rotating domain rotates according to its angular velocity and thus a new mesh grid is generated at each time step. In Fig. 1 the geometry of two subdomains of the fluid in the turbine is shown.

The boundary conditions at inlet are adopted as total pressure and total temperature and at outlet, static pressure is selected as boundary condition.

3- Mesh Independency Analysis and Choice of Turbulence Models

The mass flow rates of the turbine are calculated at two different rotational speed of the turbine using five different meshes in order to analyze the mesh independency of the turbine.

As it can be seen from Fig. 2 the simulation at high RPMs is more sensitive to grid size than low RPM and according to this chart, a grid with the 2700000 meshes is selected as appropriate grid for the rest of the simulation.

In Figure 3 a comparison between the accuracy of the results obtained from simulation of the flow inside the turbine in one specific working point using three different turbulence model is given. The mass flow rate of the turbine is calculated using three turbulence models and is compared to the experimental test results.

As it can be seen, accuracy of the three mentioned turbulence models differs slightly from each other and because of computational low cost of k-ε model, it's preferred to continue with this model.

4- Results and Discussion

After validating the simulation results in three working point of turbine corresponding to both open and closed wastegate condition, the turbine's performance map and isentropic efficiency chart are obtained in an extensive working range. Also comparing the accuracy of simulation results obtained by implementation of both sliding mesh and MRF approaches, shows that simulations based on sliding mesh approach has better accuracy than MRF in whole working point of the turbine but the former's computational cost is much higher than the latter's one.

It is shown that opening of the wastegate can reduce isentropic efficiency and power output of the turbine significantly which leads to reduction in rotational speed and thus reduction in engine boost pressure. For example in Fig. 4 the power produced by the turbine in four different wastegate opening and in a specific rotational speed is compared to each other over an extensive pressure ratio range.

Based on flow patterns inside the turbine obtained from simulation, two defects of the turbine geometry designs are determined. First one is the bend at outlet of the turbine causes a sudden change of fluid flow path which may result in vortex flow creation and hence the efficiency of the turbine is affected. Second defect is that the main passage of the turbine and the bypass passage are unparallelled to each other and this causes interaction and mixing of the two flows

passing through each passage and hence the performance of the turbine is affected.

5- Conclusion

In this paper a full 3D simulation of the flow inside the turbocharger turbine has been performed under different wastegate openings and the results have been validated with experimental data. The important results are as follow:

- Steady simulation of the turbine has around 6 percent of error in predicting turbine's behavior.
- Wastegate opening causes the power output and efficiency of the turbine to drop significantly at conditions similar to closed wastegate.
- Based on the isentropic efficiency map of the turbine, it is advisable to keep the pressure ratio of the turbine in the range of 1.3 to 2.5.

References

- [1] H. Karrabi, A. Hajilouy-Benisi, M. Nili-Ahmadabadi, Experimental and numerical investigation of a centrifugal compressor, in: ASME 2010 10th Biennial Conference on Engineering Systems Design and Analysis, American Society of Mechanical Engineers, 2010, pp. 447-458.
- [2] Q. Chen, J. Ni, X. Shi, Q. Wang, Q. Chen, S. Liu, Research on Effect of Wastegate Diameter on Turbocharged Gasoline Engine Performance, 0148-7191, SAE Technical Paper, 2016.
- [3] A. Dupuis, Key Features of the Turbocharged Engine System for the New PSA 1.2 L PureTech e-THP 3 Cylinder Gasoline Engine, 19, in: Aufladetechnische Konferenz, Sept, 2014.
- [4] E.M.V. Siggeirsson, S. Gunnarsson, Conceptual design tool for radial turbines, Journal of Applied Mathematics, 2011 (2011) 16.
- [5] J. Galindo, S. Hoyas, P. Fajardo, R. Navarro, Set-up analysis and optimization of CFD simulations for radial turbines, Engineering Applications of Computational Fluid Mechanics, 7(4) (2013) 441-460.
- [6] H. Tabatabaie, M. Broumand, M.T. Rahni, Comparing between 1D and 3D flow analysis in the on-engine small turbocharger experimentally and numerically, Modarres Mechanical Engineering, 11(4) (2012) 75-88. (in Persian).
- [7] L.M.G.-C. González, Experiments and Modelling of Automotive Turbochargers under Unsteady Conditions, 2015.

