CFD ANALYSIS OF TURBOCHARGER TURBINE

Naveen. B
Assistant Professor, Department of Mechanical Engineering
Kuppam Engineering College, Kuppam, Chittoor (dt.), A.P-517425, India.

Kallu. Raja Sekhar
Associate Professor, Department of Mechanical Engineering
Kuppam Engineering College, Kuppam, Chittoor (dt.), A.P-517425, India.

ABSTRACT

Downsizing is a trend in engine development that allows better efficiency and ower emissions based on the increase of power output in reduced displacement engines. A natural gas engine for producer gas operation was adopted. The Producer gas fuelled Engines are the upcoming Technology and more friendly to the environment compared to diesel and petrol Engines. There are some issues related to power de-rating from the engine related to the fuel properties. The main cause for the power de-rating is due to the relatively lower heating value of stoichiometric mixture of producer gas and air. This loss in power can be recovered to a much large proportion by Turbocharging. Matching of the correct turbocharger to an engine is of great importance and is vital for successful operation of a Turbocharged engine. It is important to have a Turbine Map for matching the Turbocharger with an engine. The characteristics of the Turbocharger’s Turbine from the original manufacturers were not available. In this work an attempt is made to establish the Performance Characteristics and hence the turbine map for a stripped out Holset Turbocharger Turbine to match with a producer gas fuelled engine.

KEYWORDS

PG Producer Gas, Tin Inlet Temperature, Tout Outlet Temperature, Pin Inlet Pressure, Pout Outlet Pressure, Pr Pressure Ratio.

INTRODUCTION

Now a days the greater importance is being placed on improving the efficiency and fuel consumption of automobiles due to the global move to reduce Co_2 emissions. An engine ideally should deliver simultaneously a high power density at low fuel consumption. High pressure Turbocharging is very much essential to improve the fuel consumption of an engine by enabling downsizing. Therefore, it necessitates developing an optimum turbo charging system to suit the engine specification. Computational fluid dynamics (CFD) is the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of computer-based simulations. The advent of high speed digital computers combined with the development of accurate numerical algorithms for solving physical problems on these computers has made it possible to use CFD as a research tool and design tool. In the present study CFD simulation on a Turbocharger Radial Turbine was conducted to obtain the characteristics of flow through Turbine using commercially available CFD tool ANSYS 14.5 and Turbine map is generated for different pressure ratios at varying rotational speeds (N) and mass flow rate ($\dot{m}$).

Compressed air (or air/gas mixture in case of gas engine) flow results in a larger quantity of mixture being forced into the engine, creating more power. The energy used to drive the turbo compressor is extracted from waste exhaust gases. As exhaust gasses leave the engine they are directed through a turbine wheel placed in the exhaust flow. The gases drive the turbine wheel around, which is directly connected via a shaft, to the compressor wheel.

Increased exhaust gas flow increases the inlet pressure which thereby drives the turbine wheel faster, this provides the engine more air or air/gas mixture, producing more power. A limit is reached once a pre-determined boost pressure is achieved. At this point the exhaust gas is redirected away from the turbine wheel, thus slowing it down and limiting the maximum boost pressure. This redirection valve is known as the waste gate. This extraction of energy, from exhaust gas, to improve engine efficiency is the device known as the Turbocharger. The advantages of using a turbo engine include improved fuel efficiency and reduced exhaust emissions.

In order to handle speeds up to 150,000 rpm, the turbine shaft has to be supported very carefully. Most bearings would explode at speeds like this, so most turbochargers use a fluid bearing. This type of bearing supports the shaft on a thin layer of oil that is constantly pumped around the shaft. It cools the shaft and some of the other turbocharger parts, and it allows the shaft to spin without much friction. When intake air (or air/gas mixture) is compressed by a turbocharger it is also heated. Hot intake air (or air/gas mixture) is not good for power and will increase the chance of detonation. A charger cooler (or After-cooler) reduces the intake temperature by pushing the mixture through a heat exchanger (much like a small radiator) that absorbs some of the heat out of the charge.

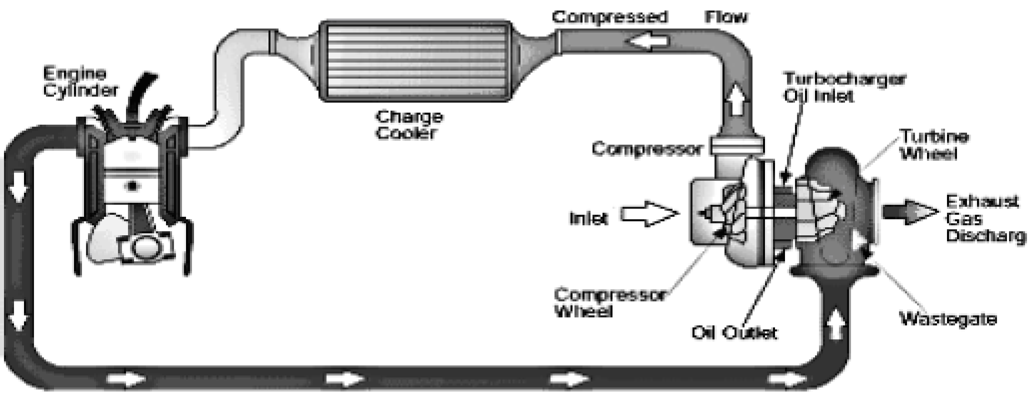


Fig. 1 Working of Turbocharger

Chemical reaction equation of stoichiometric mixture of producer gas-air is given by Equation, 1 mole of Producer gas (PG) = 0.1961 H_2 + 0.1968 CO + 0.0252 CH_4 + 0.1255 CO_2 + 0.4564 N_2

Individual chemical reaction equations of stoichiometric fuel-oxygen mixtures of H_2, CO and CH_4, assuming complete combustion, are given by Equation,

$H_2 + 0.5\ O_2 \rightarrow H_2O$

$CO + 0.5\ O_2 \rightarrow CO_2$

$CH_4 + 2\ O_2 \rightarrow CO_2 + 2\ H_2O$

1 mole of (PG) + 0.2469 (O_2 + 79/21 N_2) → 0.3475CO_2 + 0.2465 H_2O + 1.3850 N_2

Here the Products of the reaction are the main fluid for Turbine.

OBJECTIVE

The main objective of the work is to study the performance characteristics of the adial Turbine by plotting the Turbine map by modelling the impeller and casing separately and then interfacing them in ANSYS CFX and to simulate it for various conditions of flow rate and speeds and to analyse the results.

METHODOLOGY

The present work is to obtain the characteristics of turbocharger's compressor by using commercially available CFD code ANSYS 14.5. Three different packages of ANSYS 14.5 reused in this study. Namely,

1. ANSYS BladeGen
2. ANSYS TurboGrid
3. ANSYS CFX

The impeller geometry was modelled in BladeGen and structured mesh is generated using TurboGrid. The volute of turbine was modelled using CATIA V5R16 and unstructured mesh was generated in ICEM CFD. The CFD simulations are carried out in two different steps first is with only impeller and the second is with impeller and volute. For both the cases the simulations are conducted for varying speeds and mass flow rates and the performance is studied. Analysis was carried out for Ptotal inlet and mass flow rate outlet Boundary Condition. This boundary condition was defined for various rotational speeds.

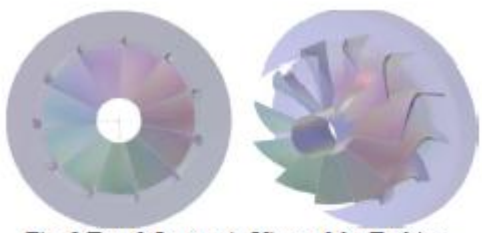


Fig. 2 Top & Isometric View of the Turbine

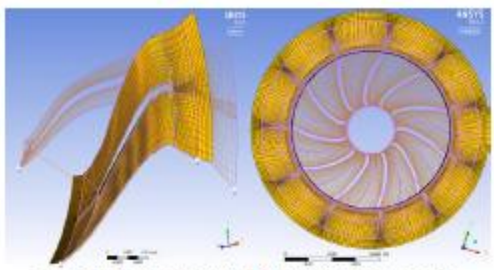
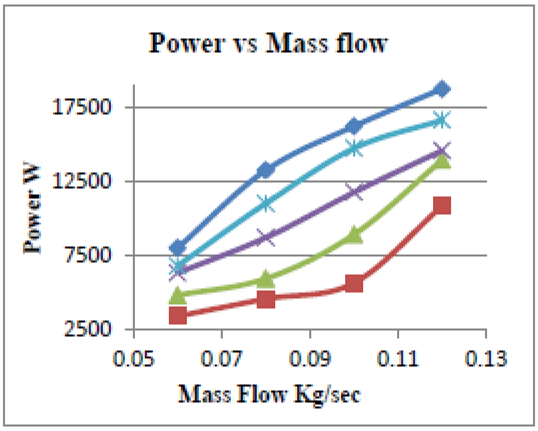
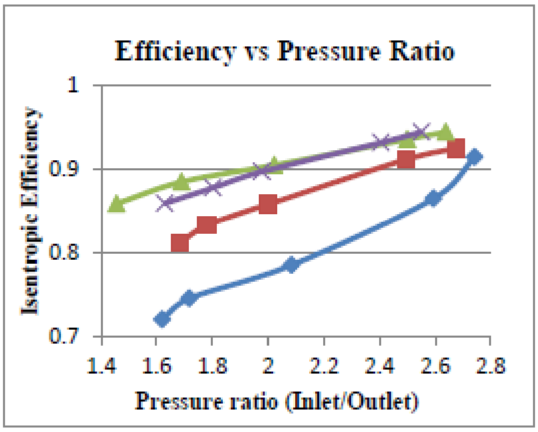
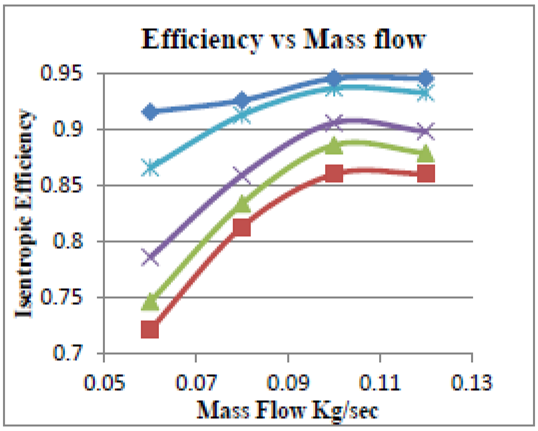


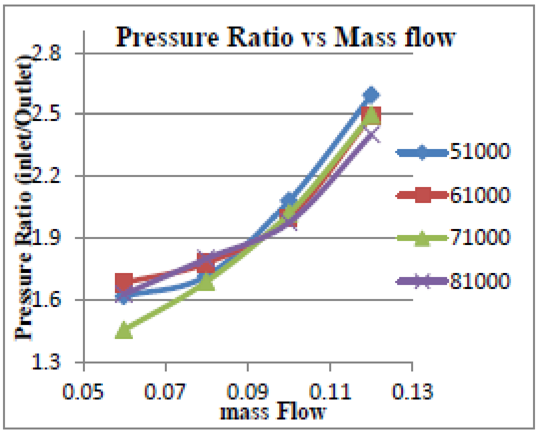
Fig. 3 Mesh generated for entire passage & Impeller

RESULTS AND DISCUSSION









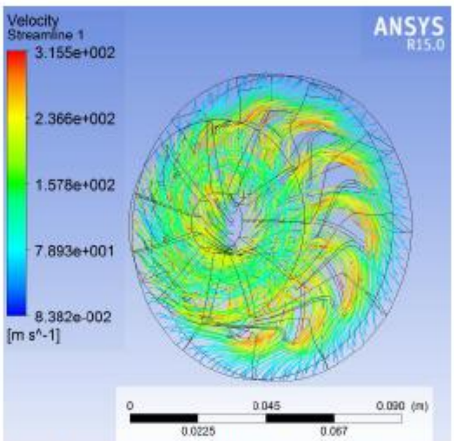


Fig. 5 3D view of Velocity streamlines

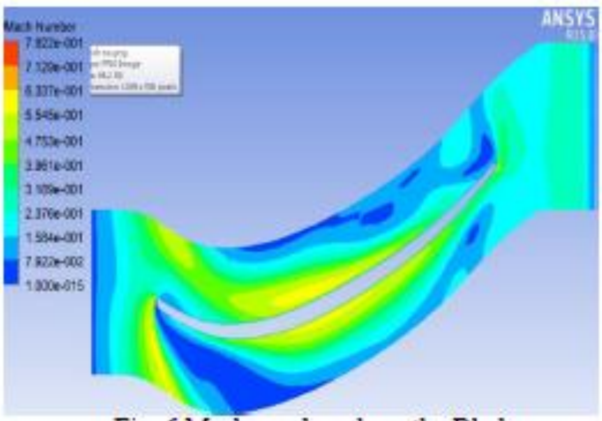


Fig. 6 Mach number along the Blade

CONCLUSIONS

CFD simulations for the Radial Turbine have carried out for different mass flow with respect to the different rotational speeds. From the above Graph it has been confirmed that the turbine is matching with the engine, above the simulation were carried for only for impeller, the further work has to be carried out with assembling the casing with same conditions.

ACKNOWLEDGMENT

I wish to express my deepest gratitude to Dr. N K S Rajan, Chief Research Scientist, CGPL, IISc, Bangalore for giving me an opportunity to carry out the project work in CGPL. It is by the inspiring discussions and close interactions with him that my hope was Strengthened. I am thankful to the Combustion & Gasification Propulsion Laboratory CGPL, IISc) for Providing me the required help and Computer Laboratory facility during my work andeven I am very thankful to my guide Praveen. D, Assistant Professor, Department of Mechanical Engineering, AIT, Chikmagalur. For giving me the support and strength to carry out this project.

REFERENCES

[1] J. Galindo, H. Climent, C. Guardiola, A. Tiseira, J. Portalier, *"Assessment of a sequentially turbocharged diesel engine on real-life driving cycles"*, Int. J. Veh. Des. 49 (1/2/3) (2009).

[2] J. Galindo, H. Climent, C. Guardiola, J. Domenech, —*Strategies for improving the mode transition in a sequential parallel turbocharged automotive diesel engine"*, Int. J. Automot. Technol. 10 (2) (2009) 141–149.

[3] F. Pflüger, —*Regulated two-stage turbocharging – KKK's new charging system for commercial diesel engines"*, in: IMechE, C554/035/98, Sixth International Conference on Turbocharging and Air Management Systems, 1998.

[4]. G Sridhar, H V Sridhar, S Dasappa, P J Paul, N K S Rajan, H S Mukunda, *"Development of producer gas engines", Proceedings of the Institution of Mechanical Engineers",* Part D: Journal of Automobile Engineering Vol. 219, pp. 423-437; 2005

[5]. H V Sridhar, *"Characterization and Matching of Turbochargers for Producer Gas Engine Applications-Experiments and Analysis"*, Ph.D. Thesis, Combustion, Gasification and Propulsion Laboratory, Dept. of Aerospace Engineering, IISc, Bangalore, 2008.

[6]. Ravindra babu yarasu, *"Premixed turbulent combustion of Producer gas in closed vessel and Engine cylinder",* Ph.D. Thesis, Combustion, Gasification and Propulsion Laboratory, Dept. of Aerospace Engineering, IISc, Bangalore, 2009.

[7]. SL Dixon, *"Fluid Mechanics and Thermodynamics of Turbomachinery"*, LinacreHouse, Jordan Hill, Oxford OX2 8DP, UK, 1998.