Computational Fluid Dynamics Analysis of A Turbocharger System

Engineering

KEYWORDS : CFD, Turbocharger, Compressor, Turbine

Shalini Bhardwaj | Assistant Professor, Department of Mechanical, IES, IPS ACADEMY, INDORE (M.P.)

Yashwant Buke | Assistant Professor, Department of Mechanical, IES, IPS ACADEMY, INDORE (M.P.)

ABSTRACT

In recent years, manufacturing industries are employed a new manufacturing methods to increase environmental protection and improved output. CFD is a complimentaryexperimental, theoreticalapproach forsimulating real flow as it is much cheaper than experimental testing.CFDas an analyzing tool for the turbocharger design, enhances off-design operation and improves performance. This paper describes the procedure of analyzing the turbocharger flow passage using CFD.

I. INTRODUCTION

Turbocharger is a method of forced induction. Forced induction is the process of cramming more air into an internal combustion engines for more horsepower that is normally draws by its own vacuums, so that the cylinders contains denser air , which results in a bigger explosion, thus more horsepower. This paper describes the procedure of simulating fluid flow in turbochargers turbines and compressors with the help of CFD. Computational Fluid Dynamics (CFD) is a cost affected tool to provide detailed flow information inside the complete turbocharger. In present work, numerical flow simulation in turbocharger (turbine and compressor) has been carried out using commercial code ANSYS CFX. The investigated turbocharger was composed by centrifugal compressor and radial- axial turbine with diffuser, casing, blades, impellers and return channel. CFD results were validated with experimental results for certain chosen performance parameters such as pressure ratio, vector plots and polytrophic efficiencies. The variation of efficiency and discharge with speed factor are presented in graphical form. The pressure and velocity distribution at runner blade for both compressor and turbine are also presented.

II. GEOMETRY

1. Turbocharger

Turbocharger is extensively used throughout the automobile industries as they can enhance the output of an internal combustion engine without the need to increase its cylinder capacity. A turbocharger is a device that adds to the engine in order to increase power and efficiency. Some like to say that a turbocharger is a centrifugal supercharger driven by exhaust gas instead of mechanically, the only differences are the way of the system is powered.

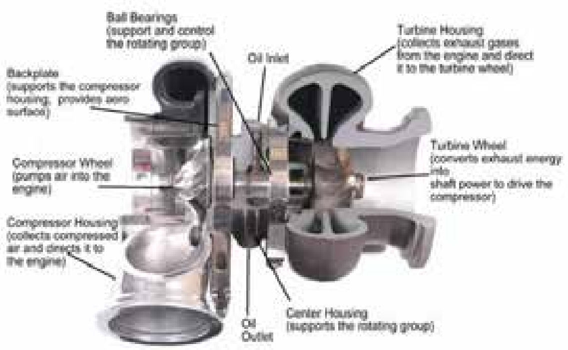


Figure1. Turbocharger cross section

Turbocharger components are classified as turbine housing (volute), turbine (radial and axial flow type), compressor, compressor wheel (impeller), diffuser, bearing system ,bearing housing , control system ,waste gates, inter cooler. Turbocharger having a single shaft on which both compressor and turbine is takes place. Back plate (support the compressor housing, provides aero surface, Center housing (supports the rotating group).

Compressor

Turbocharger compressors are generally centrifugal compressors consisting of three essential components: compressor wheel, diffuser, and housing. With the rotational speed of the wheel, air is drawn in axially, accelerated to high velocity and then expelled in a radial direction.

Compressor Wheel (Impeller)

Compressor impellers are produced using a variant of the aluminum investment casting process.

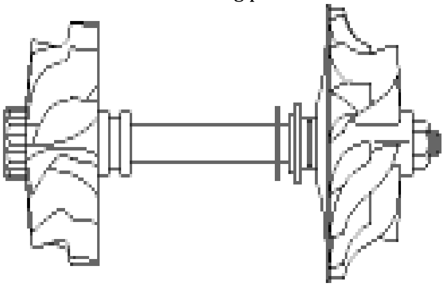


Fig2 - Compressor wheel (impeller)

Compressor Cover

Compressor housings are also made in cast aluminum.

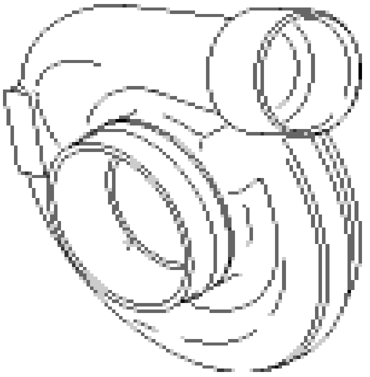


Fig3- Compressor cover

Turbine

The turbocharger turbine, which consists of a turbine wheel and turbine housing, converts the engine exhaust gas into mechanical energy to drive the compressor. The gas, which is restricted by the turbine's flow cross-sectional area, results in a pressure and temperature drop between the inlet and outlet. This pressure drop is converted by the turbine into kinetic energy to drive the turbine wheel.

There are two main turbine types: axial and radial flow. In the axial-flow type, flow through the wheel is only in the axial direction. In radial-flow turbines, gas inflow is centripetal, i.e. in a radial direction from the outside in, and gas outflow in an axial direction.

Up to a wheel diameter of about 160 mm, only radial-flow turbines are used. This corresponds to an engine power of approximately 1000 kW per turbocharger. From 300 mm onwards, only axial-flow turbines are used. Between these two values, both variants are possible.

Wheel
The Turbine Wheel is housed in the turbine casing and is connected to a shaft that in turn rotates the compressor wheel.

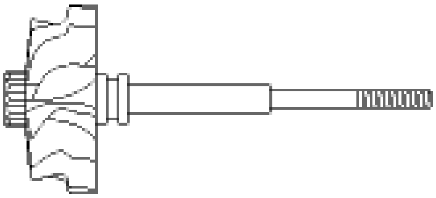


Fig4 –Turbine wheel

Turbine Housing
Turbine housings are manufactured in various grades of spheroid graphite iron to deal with thermal fatigue and wheel burst containment.

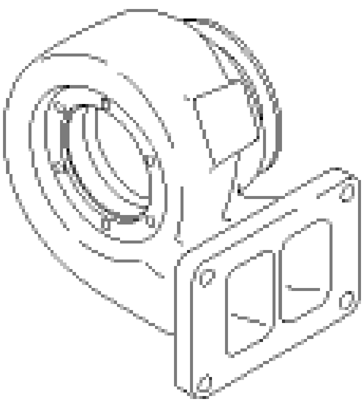


Fig5-: Turbine housing

Bearing Housing
A grey cast iron bearing housing provides locations for a fully floating bearing system for the shaft, turbine and compressor which can rotate at speeds up to 170,000 rev/min. Shell molding is used to provide positional accuracy of critical features of the housing such as the shaft bearing and seal locations.

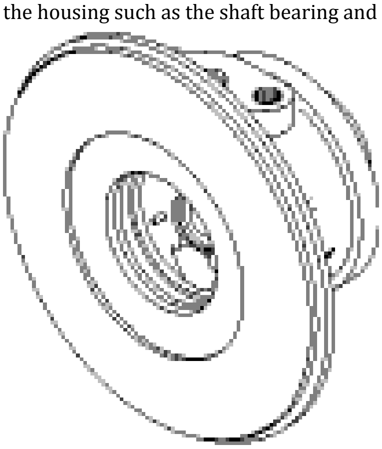


Fig6-: Bearing housing

CFD Analysis
Computational Fluid Dynamics (CFD) is the present day state-of-art technique in fluid engineering flow analysis. It has wide range of applications-like pumps, fans, compressors, turbines, automobiles, process industries, aerospace, in fact in any areas of study, where there is fluid in motion, air, water, steam etc. CFD analysis is very useful for predicting pump performance at various mass-flow rates. For designers, prediction of operating characteristics curve is most important. All theoretical methods for prediction of efficiency merely give a value; but one is unable to determine the root cause for the poor performance. Due to the development of CFD code, one can get the efficiency value as well as observe actual intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages; The CFD occupies today a very significant place in the disciplines of fluid mechanics and turbo machinery due to the great progress in the development of numerical methods and computing power. However, the initially use of CFD tools to design a new machine represents a non realistic procedure (Arnone, 1999). The design of a new machine (or upgrading an existing machine) would require a great investment of time without guarantee of success. Along with the introduction of CFD tools, the incorporation of computer aided design (CAD) codes has speeded up the design process because of a faster geometry and grid generation.

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor. The CFD analysis was solved by means of ANSYS CFX software. The fully 3D, Compressible, viscous, turbulent analysis of the fluid (air) flow was solved. The Computational domain was periodically repeating segment of impeller, diffuser and return channel. The solution was so made on the segment around one impeller blade and around two return channel vanes.

III. METHODOLOGY
The complete analysis and design of a turbocharger is presented in this report. The turbocharger analysis was performed utilizing Finite Element Analysis, also known as FEA, which permits the user to analyze the model by using of software according required environmental condition and useful parameters that are necessary to converge on a solution. The analysis for the turbocharger is constrained by the following two main items:

- Compressor impeller and turbine should have similar outside diameters
- Turbine and impeller rotate at same rotational velocity

The reason for maintaining similar overall diameters for the turbine and compressor impeller is based on ease of manufacturability and placement in the engine compartment. Having two very different diameters would prove cumbersome when arranging other components that go in the engine compartment. This is an example of a design consideration that is dependent on application. The second constraint is due to the fact that the impeller and turbine sit on the same shaft. This constraint is found in most turbo generators.

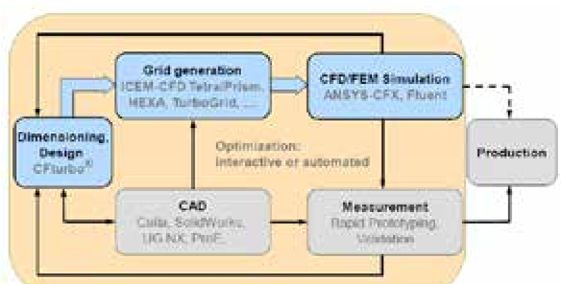


Figure7 -: Flow chart for Design and Simulations of Turbocharger Components by using of design software

IV. ANALYSIS
The work presented in this paper is carried out on and around a turbocharged; 4-cylinder SI-engine, more detailed data is pre-

Volume : 3 | Issue : 5 | May 2014 • ISSN No 2277 - 8179

sented in Table1. For the main part of the work the engine maintained its production version of turbocharger and manifold. The turbocharger was a waste gated, single entry, vane less turbine, as is standard for SI-engines. The manifold was a cast-iron, compact 4-2-1 design.

1. Table. Engine data.

No. cylinders	4
Valves per cylinder	4 (pent-roof)
Bore x stroke [mm]	90 x 78
Max power [kW]	151
Max torque [Nm]	280
Max inlet pressure, controlled [Bar]	1.8

The initial parameters are given in Table I. With these initial parameters it is possible to start the analysis employing the correct equations in FEA.

2. Table Parameter list for compressor and turbine

Parameter	Compressor	Turbine
Volumetric flow rate V [m^3/s]	0.33	NA
Mass flow rate m[kg=s]	NA	0.55
Inlet pressure	$P_{01} = P_{02} = 1.8$bar	$P_{in} = P_{ce} - \Delta P_{loss}$
Design rpm	60,000	60,000
Isentropic efficiency	87%	83%
Hub Diameter station 2:	D_{h2} = 24mm	NA
Tip diameter station 2:	D_{t2} = 67:8mm	NA
Tip diameter station 3:	D_{t3} = 102mm	NA
Relative inlet angle	β2 = 143°	NA

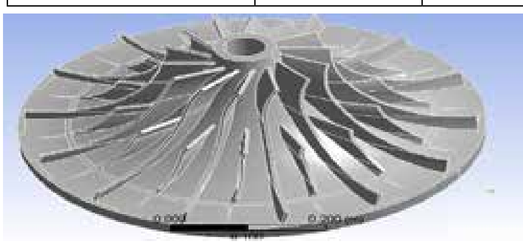
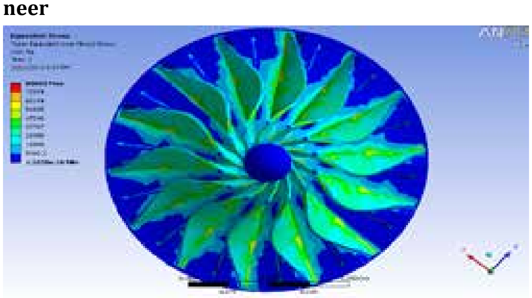


Figure 8: Centrifugal compressor designed with Pro-Engineer



Figur.9: Impeller safety factor of 0.5 from stress analysis

An additional consideration in the design is the impeller's material specification. The selected material is dependent on manufacturability, cost, and stress analysis on the impeller. The stress analysis is performed on ANSYS. ANSYS is capable of performing various stress analysis, one of which is applied centrifugal forces. This analysis was performed on both the impeller and turbine rotor. For this part of the design, a stainless alloy was chosen as the base material; however, after a performed stress analysis using centrifugal loads, a safety factor of 0.5 was attained. A safety factor of 0.5 is insufficient, and therefore other materials were selected and retested. A strong alloy was selected in the end after stress analysis results proved a safety factor of 2. Figure 9 shows the safety factor results and Figure 10 shows deformation from the stress analysis on stainless steel impeller.

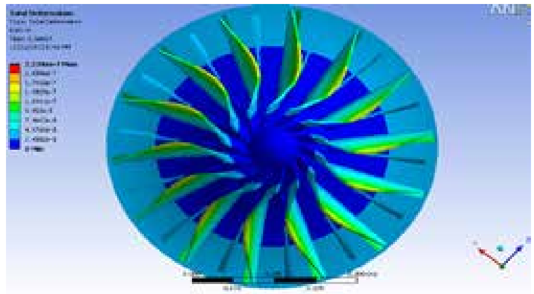


Figure 10: Impeller deformation results from stress analysis

Compressor Solution

Before the starting of any solution in ANSYS the meshing of model is necessary for refinement of solution under the whole body. The meshing of compressor part is shown in figure 11.

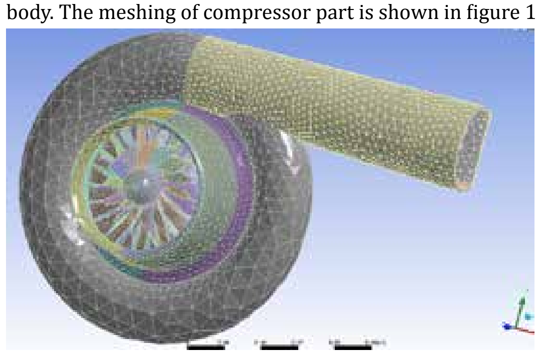


Figure 11. The meshing picture of compressor part

The second step in the process for finding a viable compressor solution is to solve for the thermodynamic properties of the impeller. Beginning with the inlet temperature and using build in thermodynamic properties in FEA by using of CFD analysis, enthalpy and density at the inlet are found. Next the geometry is defined for the inlet and outlet area of blade; however, since no information is given for the outlet area, a guess value must be used. In particular, a blade height of 3mm, 4mm, 5mm, 7.5mm, 10mm, and 12.5mm were used. The picture of CFD solution with assembly of compressor impeller and housing is shown in figure.12.

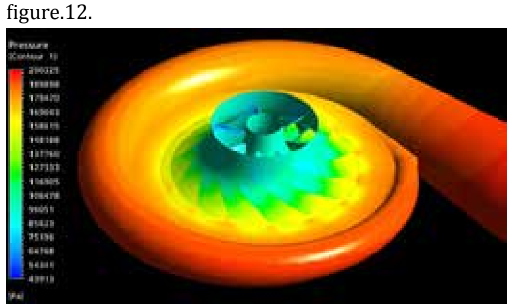


Figure 12.Housing of compressor with result of pressure effect analyze by using of CFD Analyze

The pressure increase is attained in the compressor due to the diverging radial vanes. The vanes extend outward in the radial direction causing the physical area between the vanes to increase. Due to the continuity equation, this increase in area forces the velocity of the air to slow, and therefore pressure to increase. The figure 8 and 9 is shown the velocity effect in compressor housing. Immediately following the radial impeller of the compressor, the air enters the diffuser component of the turbocharger where the area is increased further between the diffuser vanes causing a higher pressure increase.

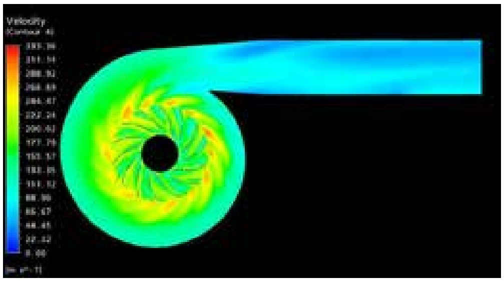


Figure 13. The CFD solution for velocity effect inside the compressor housing

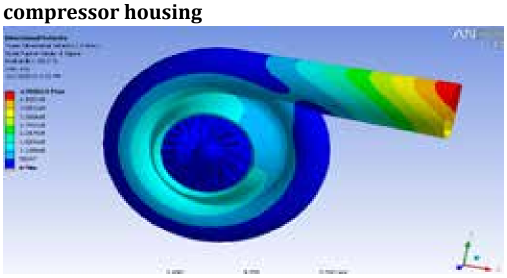
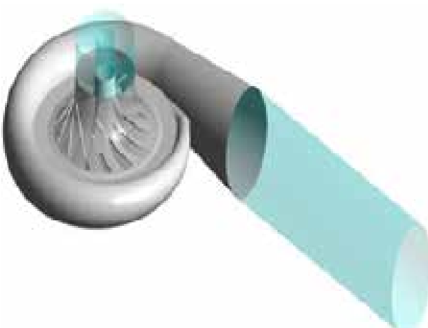


Figure 14. The CFD solution for velocity effect inside the compressor housing

V. RESULTS



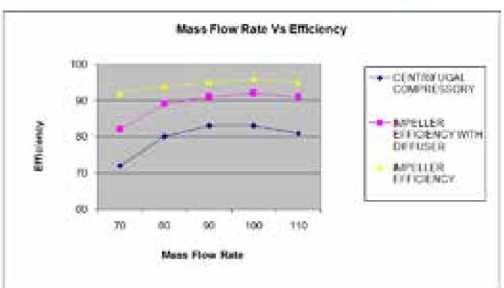


Figure 15: Efficiency of different stages at different flow rates

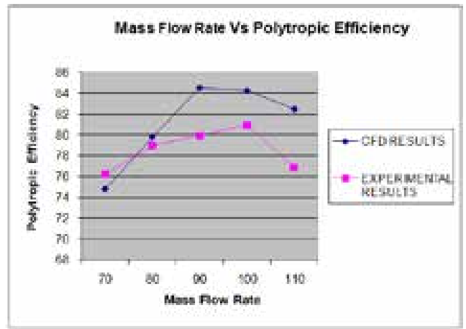


Figure 16: Mass flow rate Vs Polytropic Efficiency

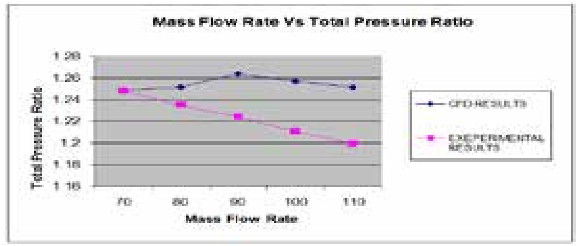


Figure 17: Mass flow rate Vs Total pressure ratio

V. CONCLUSION

The CFD simulation offers a virtual image of the internal flow in the machine allowing the analysis and comprehension of more complex phenomena. An effort was made to model the flow from inlet to the exit of a centrifugal compressor stage consisting of all the components in place using CFD tools. The vector plots, contour plots and Stream line plots are generated for better understanding of fluid flow through centrifugal compressor stage. The results obtained from CFD analysis were validated with the experimental results for performance parameters such as polytrophic efficiency, power input, and total pressure ratio. CFD results on the polytrophic efficiency of a centrifugal compressor predicted the experimental results closely with a variation of 2%. Similarly power input to a centrifugal compressor stage is predicted by CFD which compares closely with experimental results with a variation of 2.9%. Total pressure ratio of a centrifugal compressor stage as estimated by CFD tools almost complies, with negligible variation of 0.03%, with experimental results.

REFERENCE

[1] Dipl.-Ing. Jonas Belz and Dipl.-Ing. Ralph-Peter Müller "rapid Design and Flow Simulations for Tubocharger Components" EASC ANSYS Conference 2009 RAPID, CFDnetwork® Engineering, CFturbo® Software & Engineering GmbH | | [2] MeinhardSchobeiri. Turbomachinery Flow Physics and Dynamic Performance. Springer, 2005 | | [3] L.M. Larosiliere, J.R. Wood, M.D. Hathaway, A.J. Med, and T.Q. Dang. Aerodynamic design study of advanced multistage axial compressor. Paper TP2002-211568, NASA, December 2002 | [4] D. G. Shepherd. Principles of Turbo machinery. The Macmillan Company, New York, 1956. |