

# Centrifugal Compressor Analysis by CFD

Shalini Bhardwaj<sup>1</sup>, Dr. K. K.Gupta<sup>2</sup>

Mechanical Department, MITM, Indore (M.P.), India

HOD, Mechanical Department, MITM, Indore, India

**Abstract:** In order to obtain more power from the engine, a new and larger turbocharger is being used. This paper is the culmination of the complete fundamental study of air flow physics. The purpose of this project is to analyze a centrifugal compressor in a turbocharger system of a diesel engine. Turbochargers are extensively used throughout the automobile industries as they can enhance the output of an internal combustion (IC) engine without the need to increase its cylinder capacity. This paper deals with the computational fluids dynamics analysis of flow in high speed turbocharger.

**Keywords:** CFD, Turbocharger, Centrifugal Compressor

## 1. Introduction

Computational Fluid Dynamics constitutes a new “third approach” in the philosophical study and development of the whole discipline of fluid dynamics. In the 17th century, the foundations for experimental fluid dynamics were laid. The 18th and 19th centuries saw the gradual development of theoretical fluid dynamics. As a result, throughout most of the 20th century, the study and practice of fluid dynamics (indeed, all of physical science and engineering) involved the use of pure theory on the one hand and pure experiment on the other. The learning of fluid dynamics as recently as, say, 1960, involved operating in the “two-approach world” of theory and experiment. However, the advent of the high speed digital computer combined with the development of accurate numerical algorithms for solving physical problems on these computers has revolutionized the way we study and practice fluid dynamics today. It has introduced a fundamentally important new third approach in fluid dynamics – the approach of Computational Fluid Dynamics.

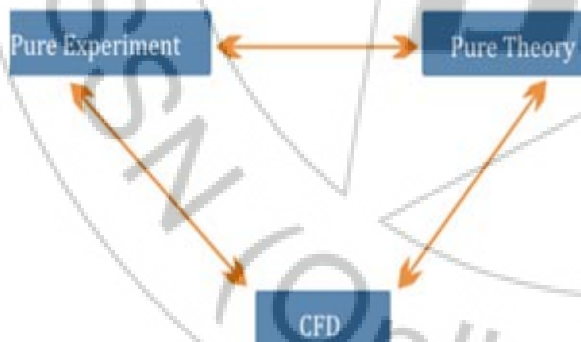


Figure 1.1: Approach of CFD

Computational Fluid Dynamics (CFD) is fluid analysis software predicts the interaction of a working fluid with its geometrical surroundings and operational environment. Accurately predicting these interactions is highly dependent on understanding the energy loss models embedded within the design code. These loss models dictate how severely performance diminishes due to inherent or sometimes improper geometrical and operational constraints. Such energy losses include skin friction, excessive pressure recovery, airfoil incidence, flow recirculation, and blade tip leakage to name a few.

## 2. Turbocharger Operation

The turbocharger compresses intake air to a density up to four times that of atmospheric pressure. This greater amount of dense air allows more fuel to be burned, thereby doubling the engine’s power output. The turbocharger also reduces exhaust emissions and exhaust noise. It compensates for the less dense air encountered in higher altitude operation. A naturally aspirated engine has a limited supply of air for combustion. The air has only atmospheric pressure pushing it into the cylinders. A turbocharger provides pressurized air, which allows for more air to be packed into a cylinder for each firing. This provides more power and much better combustion efficiency. Thus, more power is realized from a given engine size, fuel economies improve, and emissions are reduced. A centrifugal compressor pulls air through a rotating wheel at its center, accelerating the air to a high velocity, which flows radially outward through a shell-shaped housing. The air velocity is slowed after leaving the wheel, which converts velocity energy into pressure. This type of compressor is a high speed device. Current turbochargers run at 80,000 to 130,000 rpm. A normal engine may run at only 2500 to 4000 rpm. Gear- or belt-driven compressors require engine power to drive them. Of the fuel energy available for an engine, about 40% is wasted in the exhaust. A turbocharger uses some of this waste energy to drive its 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. The centrifugal compressor consists essentially having a stationary casing containing a rotating impeller which imparts a high velocity to the air. Air is sucked into the impeller eye and whirled round at high speed by the vanes on the centripetal disc. At any point in the flow of air through the impeller, the centripetal acceleration obtained by a pressure head, so that the static pressure of the air increases from the eye to the tip of the impeller. The remainder of the static pressure rise is obtained in the diffuser, where the high velocity of the air leaving the impeller tip is reduced to somewhere in the region of the velocity with which the air will cause the impeller eye; it should be appreciated that the friction in the diffuser will cause some loss in stagnation pressure. The normal practice is to be design the compressor so that about half the pressure rise

occurs in the impeller and half in the diffuser. The centrifugal compressor consists essentially of a stationary casing containing a rotating impeller which imparts a high velocity to the air, and a number of fixed diverging passages in which the air is decelerated with consequent rise in static pressure.

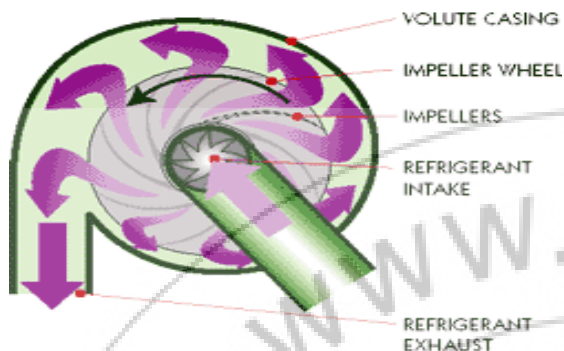


Figure 1.2: Flow Diagram for Compressor

Air is sucked into the impeller eye and whirled round at high speed by the vanes on the centripetal disc. At any point in the flow of air through the impeller, the centripetal acceleration obtained by a pressure head, so that the static pressure of the air increase from the eye to the tip of the impeller. The remainder of the static pressure rise is obtained in the diffuser, where the high velocity of the air leaving the impeller tip is reduce to somewhere in the region of the velocity with which the air will cause the impeller eye; it should be appreciated that friction in the diffuser will cause some loss in stagnation pressure. The normal practice is to design the compressor so that about half the pressure rise occurs in the impeller and half in the diffuser.

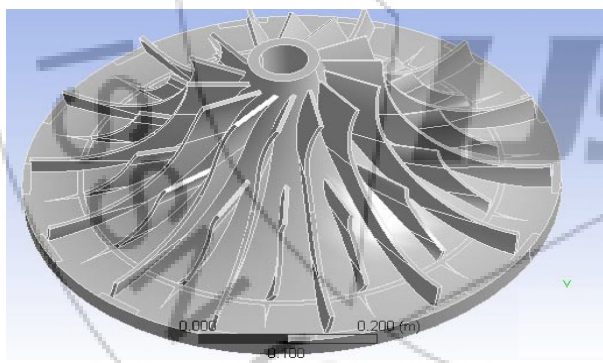


Figure 1.3: Centrifugal compressor designed with Pro-Engineer

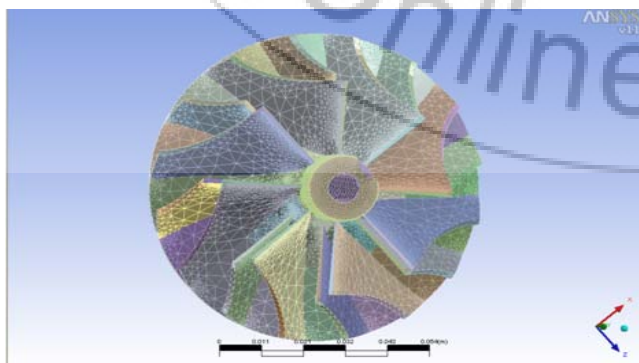


Figure 1.4: Meshing of a centrifugal compressor

As mentioned above, the stage simulation is composed of two sections - impeller and diffuser – connected by a "stage type" mixing plane where the flow properties are averaged circumferentially. The exit of the impeller and the inlet of the diffuser make up this interface. At the inlet, the boundary was defined as a subsonic inlet, with measured total temperature, total pressure and flow direction profiles. The turbulence level was defined to be medium intensity of about 5%. Periodic boundary conditions were applied to the impeller periodic plane at the middle of the impeller flow passage. The blades, hub and shroud were defined as adiabatic walls with the appropriate rotational velocity. Figure 1.5 provides a summary of the boundary conditions used in the centrifugal compressor stage simulations.

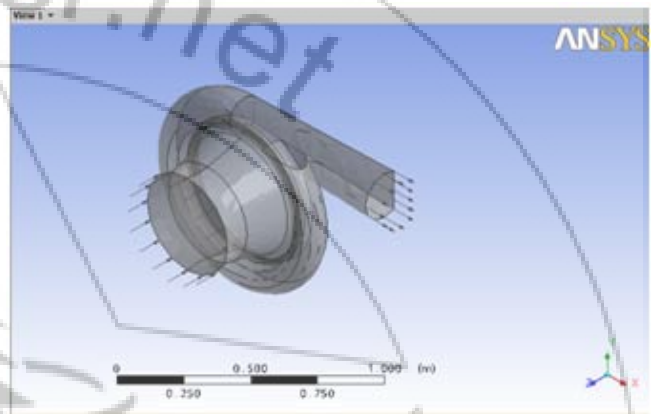


Figure 1.5: Boundary Condition applied at compressor

### 3. Analysis Results & Discussion

Simulation of the off-design performance of the centrifugal compressor stage was considered for six off design speeds (RPMs): 60%, 70%, 80%, 90%, 100% and 105% of design speed.. These numerical predictions of the speed lines are compared directly to available experimental data. These results will be presented in this section. The first performance measure investigated was the total to static pressure ratio. This is a ratio of the static pressure ratio at the exit divided by the total pressure at the inlet respectively. Figure 1.5 and Figure 1.6 shows the plots of total to static pressure ratio as a function of the inlet corrected and exit corrected mass flow.

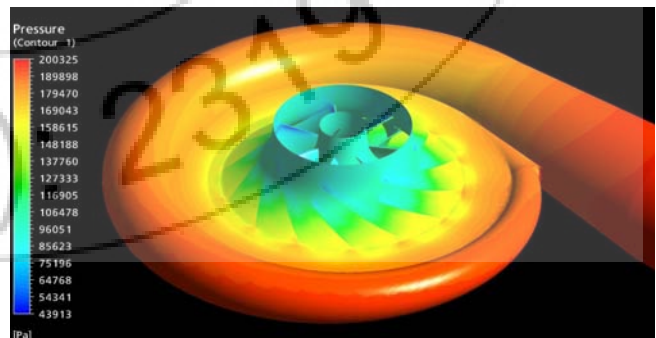


Figure 1.6: Housing of compressor with result of pressure effect analyze by using of CFD

With this basic understanding for the impeller's physics, the CFD analysis can be applied and a solution found. The plots of the pressure ratio versus exit corrected mass flow rate

shown in Figure 8 are useful in understanding compressor off-design operation. The exit corrected mass flow rate graph is useful because for jet engines, the goal of the compressor is to supply a steady flow of air into the combustion section.

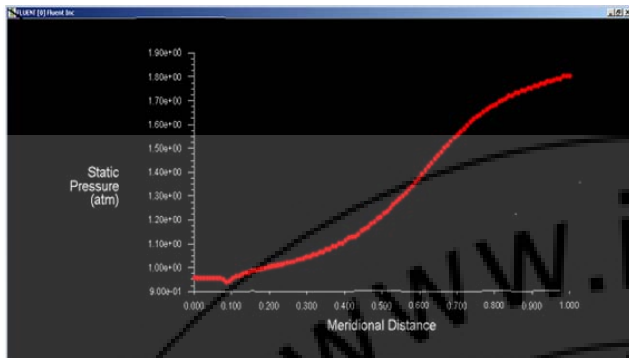


Figure 1.7: Static pressure vrs Meridional distance effect analyze by using of CFD

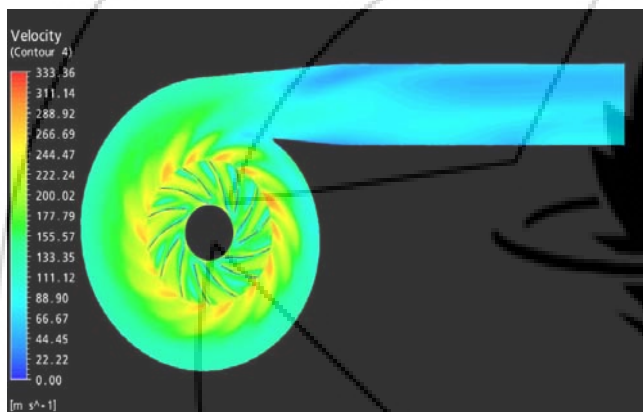


Figure 1.8: The CFD solution for velocity effect inside the compressor housing by using ANSYS

#### 4. Conclusion

The main contribution of this paper was to demonstrate that off-design performance of a centrifugal compressor can be accurately simulated using commercial CFD software, with design software, ANSYS CFD to generate high quality meshes and CFX as the CFD solver. Entire speed maps can be generated with high accuracy, particularly for lower rotational velocities. 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 polytropic efficiency, power input, and total pressure ratio. Surge prediction is also an unsteady phenomenon which means it is an area that can be improved upon. The results of the unsteady centrifugal compressor computations indicate that fully unsteady flow analyses can yield better performance predictions particularly at high rotational speeds.

#### References

- [1] *CFD Model for Turbocharger Journal Bearing Performances*. **M. Deligant, P. Podevin, G. Descombes\*** Conservatoire national des arts et métiers, Laboratoire de génie des procédés pour l'environnement l'énergie et la santé, LGP2ES e EA21 e Case 333, 292, rue Saint-Martin, 75141 Paris Cedex 03, France
- [2] *CFD: AN EFFECTIVE TOOL FOR FLOW SIMULATION IN HYDRAULIC REACTION TURBINES* **Dr. Vishnu Prasad, Dr. Ruchi Khare / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 4, July-August 2012, pp.1029-1035**
- [3] *CENTRIFUGAL COMPRESSOR FLUID FLOW ANALYSIS USING CFD*. **Vemu.Vara Prasad\*1, M. Lava Kumar2, B. Madhusudhan Reddy3** Science Insights: An International Journal Universal Research Publications Science 2011; 1 (1): 6-10
- [4] *AIR MASS FLOW ESTIMATION IN TURBOCHARGED DIESEL ENGINES FROM IN- CYLINDERS PRESSURE MEASUREMENT* **J.M. Desantes et al. / Experimental Thermal and Fluid Science 34 (2010) 37–47.**
- [5] *COMPUTATIONAL ANALYSIS OF INTAKE MANIFOLD DESIGN AND EXPERIMENTAL INVESTIGATION ON DIESEL ENGINE FOR LCV* **.S. KARTHIKEYAN, R. HARIGANESH, M.SATHYANADAN, S. KRISHNAN** International Journal of Engineering Science and Technology (IJEST) ISSN : 0975-5462 Vol. 3 No. 4 Mar 2011
- [6] *THE EFFECTS OF TURBOCHARGER ON THE PERFORMANCE AND EXHAUST EMISSIONS OF A DIESEL ENGINE FULLED WITH BIODIESEL* **Murat Karabektas\*** Renewable Energy 34 (2009) 989–993
- [7] *NUMREICAL SIMULATION OF AIR FLOW THROUGH TURBOCHARGER COMPRESSOR* **Kui Jiao a, Harold Sun b, Xianguo Li a,\* , Hao Wua, Eric Krivitzky c, Tim Schram b, Louis M. Larosiliere c / Applied Energy 86 (2009) 2494–2506**