Computational Fluid Dynamics Analysis for Centrifugal Compressor using Advanced Technique

Bharatkumar Manharlal Trivedi¹, Dr Jeetendra A. Vadher²
¹. Research Scholar Rai University, Ahmedabad.
². Professor and head, Mechanical Engineering Department, Government Engineering College, Palanpur

Abstract — To obtain more power from the engine, a new and the larger turbocharger is being used. This paper is the study of the complete fundamental study of air flow physics. Turbochargers are mostly 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. The characteristics derived from Finite Element Analysis and then it will be compared with experimental results.

Keywords— Centrifugal Compressor, Turbochargers, FEA, CFD, IC Engine.

1. INTRODUCTION

Computational Fluid Dynamics constitutes a new “3rd approach” in the philosophical study and development of the whole discipline of fluid dynamics. In the Seventeenth century, the foundations for experimental fluid dynamics were laid. The Eighteenth and Nineteenth centuries saw the gradual development of theoretical fluid dynamics. As a result, throughout most of the Twentieth century, the study and practice of fluid dynamics 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 “2-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 introduced a fundamentally important new 3rd approach in fluid dynamics – the approach of CFD. Computational Fluid Dynamics (CFD) is a latest tool of 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. Description of Test Rig and Instrumentation:

In order to obtain useful results a very simple system was chosen for the experimental study. The test rig is shown in fig.2.1
The compressor was a small (200mm impeller diameter), single stage, centrifugal compressor. The compressor speed selected for the test was 90,000 rpm. The characteristic of the compressor at the chosen speed is shown in fig. 3.9. These characteristics derived from Finite Element Analysis and then it will be compared with experimental results.

The duct work upstream of the compressor consisted of a straight pipe and an inlet bell mouth. The downstream ducting had two right angle bends and a branch to which a fast acting blow-off valve was mounted. (The valve stroke times for the opening and closing of the blow-off valve were 0.25 seconds and 0.5 seconds respectively.) The blow-off valve was a repeatable poppet actuator type, and fast transients were introduced into the system by the operation of this valve. The compressor delivery air exhausted through a throttle valve, and the compressor steady state operating point on its characteristic was determined by the actuation of this throttle valve.

High performance pressure transducers were mounted at the system inlet, impeller inlet, and compressor outlet and blow-off valve stations respectively. Due to the relative small changes in pressure ratios during the transients considered and the slowness of the temperature changes the effect of temperature on the pressure readings were neglected.

The duct work upstream of the turbine section was a straight pipe via a combustion chamber connecting to the high pressure air supply. The exhaust from the turbine was discharged via a straight vertical pipe to the ambient. The combustion chamber becomes operational if insufficient energy is available in the high pressure air (cold) driving the compressor at the required speed and pressure ratio (preheating using the combustion chamber was required for operating conditions above 60,000 rpm).

3. Finite Element Analysis and Characteristic Curve:

Computational Fluid Dynamics (CFD) predicts the interaction of a working fluid with its geometrical surroundings and operational environment. Accuracy of predicting these interactions of fluid in the body is completely dependent on understanding the energy loss models which is co-related within the design code. These loss models decide how severely performance reduces due to inherent or sometimes improper geometrical and operational constraints. Such energy losses include surface friction, excessive pressure drop, airfoil phenomena, recirculation of flow, and blade tip leakage to name a few.

The compressor has 24 blades that revolve about the Z-axis. A clearance gap exists between the blades and the shroud of the compressor. The outer diameter of the blade row is approximately 40 cm.

To run analysis of the aerodynamic performance, a mesh shall be created in both ANSYS Turbo Grid and the Mechanical application using an existing design which is to be reviewed beforehand in ANSYS Blade Gen. Once the meshes have been created, initial parameters defining the aerodynamic simulation will be set in CFX-Pre and then solved in CFX-Solver. The aerodynamic solution from the solver will then be processed and displayed in CFD-Post.

A mesh can be created in both ANSYS Turbo Grid and the Mechanical application. Turbo Grid will create a CFD mesh that will be part of the fluid domain. The Mechanical application will generate the solid blade mesh that is required for solving for the volumetric properties in the blade.

![Figure 3.1 Mesh generation of rotor of the compressor](image_url)
Figure 3.2 Velocity distribution

Figure 3.3 Stream Line view

Figure 3.4 Pressure distribution at 20% span of blade
After performing successive analysis for various interval of flow rate value, the continuous curve of Polytropic head can be found out at specific speed and it is plotted in figure 3.6 which is called Characteristics curve of compressor.

![Pressure distribution at 80% span of blade](image)

**Figure 3.5 Pressure distribution at 80% span of blade**

![Characteristics curve of compressor](image)

**Figure 3.6 Characteristics of compressor by Finite Element Analysis used for validation of mathematical model (Point A and B achieved by experimentally)**

3. Compressor Transients between Stable Points

By actuation of the blow-off valve a fast transient was introduced into the system. The repeated opening and closing of the blow-off valve transients between the corresponding two stable points can be obtained. Sufficient time was given between the opening and closing of the valve so that the post transient steady state operating point can be obtained. During the transient the compressor speed remained nominally constant. The stable points due to the opened and closed position of the blow-off valve are shown by point A and B respectively in above figure.
Surge test will be performed by adjusting the flow through the throttle valve. From the above work it is concluded that the transient due to the closing of the blow-off valve was sufficient to take the compressor into surge and the compressor will be made to operate at point A on the characteristic curve.

REFERENCES

27. Sanford Fleeter, et.al “Gas Turbine Engine Compressor-Combustor Dynamics Simulation Design”, Computer Science Technical Reports, Purdue University, Paper 1437
28. Tiziano Ghisu, “CFD Analysis of Inlet Flow Distortions on an Axial Fan”
33. Donald E. Bently, et.al. “Rotor Dynamics of Centrifugal Compressors in Rotating Stall”, p40, ORBIT, 2Q01