CFD Simulation of Centrifugal Blower Using ANSYS CFX

Girish A R¹, Prajwal Sandyal², Lokesh K S³, Varun N⁴
1,2,4Assistant Professor, Department of Automobile Engineering, Srinivas Institute of Technology, Mangalore
Visvesvaraya Technological University, Belgaum
3Assistant Professor, Department of Mechanical Engineering, Srinivas Institute of Technology, Mangalore
Visvesvaraya Technological University, Belgaum

Abstract: Centrifugal blower is designed to handle compressible fluids like air and gases. Centrifugal blowers are used in ventilation, power station, workshops etc. Computational Fluid Dynamics Simulation is being increasingly applied in the analysis of fluid flow in the centrifugal blower. By CFD Simulation approach makes it possible to visualize the flow condition inside centrifugal blower and provides the variable design information of the centrifugal blower. CFD largely reduces lead times and costs in designs and production compared to experimental-based approach and offers the ability to solve a range of complicated flow problems where the analytical approach is lacking. The present paper describes the total pressure ratio and to evaluate the Centrifugal Blower performance using the ANSYS CFX-13, a Computational Fluid Dynamics simulation tool. Simulation results of centrifugal blower are very much related to flow choking phenomena, which may occur anywhere within the machine such as at the inlet in the impeller or in the diffuser section. Choking is defined as the phenomenon in which the mass flow rate reaches the fixed value irrespective of any of the pressure ratios. CFD simulation can be used to detect the flow choking in centrifugal blower and to get safe range of operating at different speed.

Keywords — Centrifugal blower impeller, CFD, Total Pressure ratio, Flow Choking.

I. INTRODUCTION

Turbo machines which absorb power used to handle compressible fluids like air, gases etc can be broadly classified into fans, blowers and compressors. A fan moves large amount of air with low increase in pressure. Compressor is a machine for raising gas-compressible fluid to a higher level of pressure. Blower is a machine for moving volumes of gas with moderate increase of pressure. Blowers may consist of one or more stages of compression with the rotors mounted on a common shaft. The air is compressed in a series of successive stages and is passed through a diffuser located near the exit to recover the pressure energy from the large kinetic energy. Blowers are used in ventilation, power station, workshops etc. In general, the centrifugal compressor may be known as a fan, blower etc depending on the need to be served. Broadly speaking, fans are the low pressure compressors, blowers are the medium pressure compressors. It is therefore the analyses of one, say centrifugal compressor, will also holds good to the other machines like blower, fan. So an analysis of the blower is similar to the analyses of compressor and fan. Many scientists and researchers carried out CFD simulation of centrifugal blower, fan, compressor. Ref [1] carried out “Numerical simulation of centrifugal blower using CFX. Ref [2] has carried out an “CFD analysis for a centrifugal benchmark blower with flat hub: flow structures and noise source generation. Ref [4] has contributed to design methodology for aerodynamic design of centrifugal compressor. Ref [6] has carried out parametric study of centrifugal fan performance: experiments and numerical simulation. Ref [7] has contributed to performance improvement of centrifugal fan by using CFD. Ref [5] has carried out study of Secondary Flow Modifications at Impeller Exit of a Centrifugal Compressor. Ref [8] has carried out aerodynamic optimization in the rotor of Centrifugal Fan Using Combined Laser Doppler Anemometry and CFD modelling. Ref [9] has carried out Centrifugal Fan Impeller design with Optimization of blade. Ref [10] has carried out numerical design and parametric optimization of Centrifugal Fans with Airfoil blade impellers.

II. CFD METHODOLOGY

The methodologies considered in CFD are Pre-process, Solver, Post-process.

A. Problem set up - Pre-process

Creation of geometry- Step 1

The first step in this CFD analyses is an extraction of the flow region (i.e., the computational domain) for the CFD calculations.

Mesh Generation- Step 2

During the pre-process stage mesh generation constitutes one of the most important steps after the definition of the domain geometry. CFD requires the subdivision of the domain into a number of smaller, no overlapping sub domains in order to solve the
flow physics within the domain geometry that has been created; this results in the generation of a mesh (or grid) of cells (elements or control volumes) covering the whole domain geometry. The essential fluid flows that are described in each of these cells are usually solved numerically so that the discrete values of the flow properties such as the velocity, pressure, temperature, and other transport parameters of interest are determined. This yields the CFD solution to the flow problem that is being solved. The accuracy of a CFD solution is governed by the number of cells in the mesh within the computational domain. In general, the provision of a large number of cells leads to the attainment of an accurate solution.

Simulation Set up- Flow Physics and Fluid Properties- Step-3

CFD simulation is carried out by considering some assumptions namely compressible fluid flow, constant fluid properties, steady state conditions. The most important part in CFD analysis is to define the fluid properties and boundary condition. The fluid is defined and the boundary condition is defined to every flow region or part of the Centrifugal Blower model. Turbulence model is the most common model used in CFD to simulate turbulent conditions. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy ($k$). The second transported variable is the turbulent dissipation ($\varepsilon$) which determines the rate of dissipation of the turbulent kinetic energy.

Specification of boundary conditions- Step-4

In the pre-process stage the fourth step deals with the specification of allowed boundary conditions that are available for impending simulations. Evidently, where there exist inflow and outflow boundaries within the flow domain, suitable fluid flow boundary conditions are required to suit the fluid behaviour entering and leaving the flow domain. Appropriate boundary conditions are also required to be assigned for external stationary solid wall boundaries that bound the flow geometry and the surrounding walls of possible internal obstacles within the flow domain.

B. Numerical Solution- CFD Solver

The suitable usage of a commercial CFD code commands the core understanding of the underlying numerical aspects inside the CFD solver. This section focuses on the treatment of the solver element. The prerequisite processes in the solution procedure that have implications in the computational solutions are initialization, solution control, monitoring solution, CFD calculation, and checking for convergence.

C. Result Report and Visualization- Post-Process

CFD has a reputation of generating vivid graphic images and while some of them are promotional and are usually displayed in stunning and superb colorful outputs, the ability to present the computational results effectively is an invaluable design tool. In this section, we concentrate on some essential computer graphic techniques frequently encountered in the presentation of CFD data. The majority of ways that the CFD results are emphasized graphically can be classified under different categories.

III. MODELING AND ANALYSIS

The geometry of design needs to be created from the initial design. Having arrived at the overall geometry, other features of the design such as blade contouring, etc has to be carried out. Any modelling software can be used for modelling. The design can then be shifted to other simulation software for analysis purposes.

A. Generation of mesh

![Fig. 1 Meshed Model](image)

CFD simulation is carried out by considering some assumptions namely compressible fluid flow, constant fluid properties, steady state conditions. The most important part in CFD analysis is to define the fluid properties and boundary condition. The fluid is defined and the boundary condition is defined to every flow region or part of the Centrifugal Blower model.

IV. SIMULATION RESULTS

The mass flow rates considered are 0.033kg/sec, 0.066kg/sec, 0.1kg/sec, 0.133kg/sec, 0.166kg/sec and 0.2kg/sec with each different speed. Rotating speeds are 24500RPM, 26950RPM, 29400RPM and 31850RPM.

Case 1

The Fig. 2 represents, Total pressure ratio is plotted against the mass flow rate at a speed of 24500RPM. As expected, Total pressure ratio decreases as the mass flow rate increases at 24500RPM. At 0.166kg/sec and 0.2kg/sec the total...
pressure ratio is less than one which means the particular design is not safe to operate at 0.166kg/sec and 0.2kg/sec for 24500RPM.

The Fig. 2 represents Total pressure ratio with different mass flow rates at 24500RPM.

The Fig. 3 represents the pressure distribution on the blades at 0.133kg/sec for 29400RPM.

The Fig. 4 shows velocity vectors plotted at 0.133kg/sec for 31850RPM.

Case 2

The Fig. 5 represents, Total pressure ratio is plotted against the mass flow rate at a speed of 26950RPM. As expected, Total pressure ratio decreases as the mass flow rate increases at 26950RPM. At 0.166kg/sec and 0.2kg/sec the total pressure ratio is less than one which means the particular design is not safe to operate at 0.166kg/sec and 0.2kg/sec for 26950RPM.

The Fig. 6 represents the pressure distribution on the blades at 0.133kg/sec for 29400RPM.

The Fig. 7 shows velocity vectors plotted at 0.133kg/sec for 31850RPM.

Case 3

The Fig. 5 represents, Total pressure ratio is plotted against the mass flow rate at a speed of 26950RPM. As expected, Total pressure ratio decreases as the mass flow rate increases at 26950RPM. At 0.166kg/sec and 0.2kg/sec the total pressure ratio is less than one which means the particular design is not safe to operate at 0.166kg/sec and 0.2kg/sec for 26950RPM.
The Fig. 8 represents, Total pressure ratio is plotted against the mass flow rate at a speed of 29400RPM. As expected, Total pressure ratio decreases as the mass flow rate increases at 29400RPM. At 0.2kg/sec the total pressure ratio is less than one which means the particular design is not safe to operate at 0.2kg/sec for 29400RPM.

The Fig. 9 represents the pressure distribution on the blades at 0.133kg/sec for 29400RPM.

The Fig. 10 shows velocity vectors plotted at 0.133kg/sec for 29400RPM.

Fig 8: Total pressure ratio with different mass flow rates at 29400RPM

Fig 9: Pressure contour plots at 29400RPM

The Fig. 11 represents, Total pressure ratio is plotted against the mass flow rate at an angular velocity of 31850RPM. As expected, Total pressure ratio decreases as the mass flow rate increases at 31850RPM. At 0.2kg/sec the total pressure ratio is less than one which means this design is not safe to operate at 0.2kg/sec for 31850RPM.

The Fig. 12 represents the pressure distribution on the blades at 0.133kg/sec for 29400RPM.

The Fig. 13 shows velocity vectors plotted at 0.133kg/sec for 31850RPM.

Fig 11: Total pressure ratio with different mass flow rates at 31850RPM

Fig 12: Pressure contour plots at 31850RPM

Fig 13: Velocity vectors plotted on the plane at 0.133kg/sec for 31850RPM

Case 4

The Fig. 14 represents, total pressure ratio is plotted against the mass flow rate at different speeds.
In the figure, total pressure ratio decreases as the mass flow rate increases with different speeds. At higher mass flow rate, the total pressure ratio is less than one which means this Centrifugal Blower is not safe to operate at particular mass flow rate because flow choking takes place.

![Fig 14](image)

**V. CONCLUSIONS**

The designed centrifugal blower model is free from flow choking at design point. At higher mass flow rate with different rotating speed the total pressure ratio is less than one, it means this centrifugal blower design is not safe to operate at that range. As per Mach number values there is no flow choking. The flow is not choking at inlet Mach number values. Mach number is based on flow velocity.

**REFERENCES**

2. Robert Sandboge “CFD analysis for a centrifugal benchmark blower with flat hub: Flow structures and noise source generation”.