International Journal of Ignited Minds (IJIMIINDS)

Performance Investigation of High Pressure Ratio Centrifugal Compressor using CFD

Manjunath DC[a], Rajesh[b], Dr.V.M.Kulkarni[c]

[a]*PG student, Department of Mechanical Engineering, TOCE, Bangalore, Karnataka, India*
[b]*Scientist, NAL, Bangalore, Karnataka, India*
[c]*Professor, Department of Mechanical Engineering, TOCE, Bangalore, Karnataka, India*

ABSTRACT

The overall efficiency and the operating range of the compressor are dependent on the design of both impeller and the diffuser. Computational studies were carried out using the available commercial CFD code FLUENT to study the flow behavior in a centrifugal impeller and conventional vane diffuser for the different blade setting angles with reference to the fixed impeller blade outlet angle. The results obtained by the CFD analysis were compared with the experimental data available in test lab, which gave a total –total pressure ratio of 2.2 with an adiabatic total-total efficiency of 70%. The experiment was carried out for impeller of diameter 211mm, consisting of 38 blades and conventional diffuser of 33 blades. The experiments were conducted up to 30000rpm. The impeller is designed for pressure ratio of 5 at 47413rpm, to get the operating line, stall line and other characteristics. The compressor stage was analyzed using CFD at 35000rpm, 40000rpm, and 47413rpm.

Keywords: *Centrifugal compressor, Impeller, Diffuser, Splitter blade, Tip Clearance, CFD.*

1. INTRODUCTION

Centrifugal compressors play an important role in small gas turbine engines, turbochargers, refrigeration plants, chemical and process industry, use of centrifugal compressor stage in the development of small engines have proven to be advantageous in achieving better specific weight compare to axial stages. There is a growing tendency for design, modification and replacement of axial stage with a single stage centrifugal compressor.

This paper describes an effort to investigate and better define the capabilities of a modern CFD solver in predicting the design performance of centrifugal compressors. The results will validate the use of commercially available CFD software tools for these problems and lead to a better understanding of the flow phenomenon within the centrifugal compressor. Furthermore, better compressor designs that enhance Off-design operation and improve performance could result once confidence is gained in applying CFD. Analysis tools to the centrifugal compressor design process.

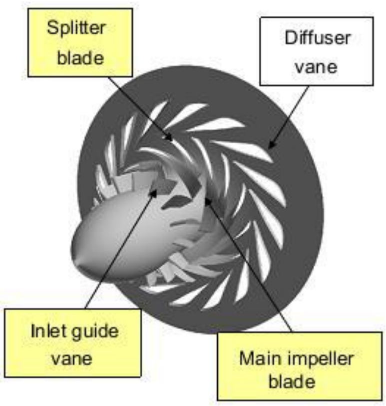


Fig1: Schematic of compressor

The compressor model geometry designed using CATIA, ANSYS GAMBIT was used to generate the appropriate mesh, and ANSYS CFD was used to solve the design compressor flow problem. The entire range of design conditions were investigated and compared to experimental data. Furthermore the effects of different mesh densities and turbulence models were also examined.

2. MODEL CONSTRUCTION AND FORMULATION

The CAD model of impeller is generated as per the coordinates provided by experimental setup. The coordinate's points are imported in catia-V5 and curves are generated. These curves are used to build blade surfaces. Fig 1 shows the CAD model of the fluid flow compressor impeller. Uniform tip clearance of 0.5mm is considered from inlet to outlet. Single flow channel is considered for analysis out of 19 Channel passages.

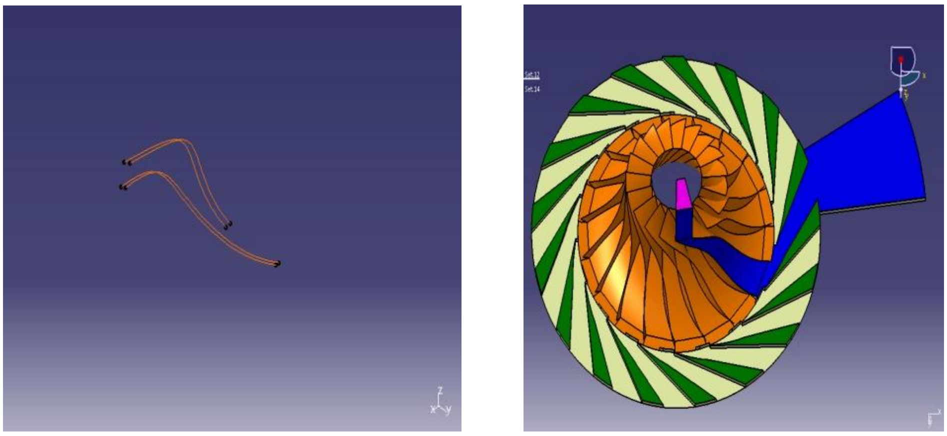


Fig 2: CATIA Model of Centrifugal Compressor

Since the compressor impeller blades are axis-symmetric and the steady state analysis has been carried, single flow channel has been considered for the analysis. Fig 2 shows the CAD model of single flow channel. The flow domain is divided into three segments: upstream domain, rotor domain and downstream.

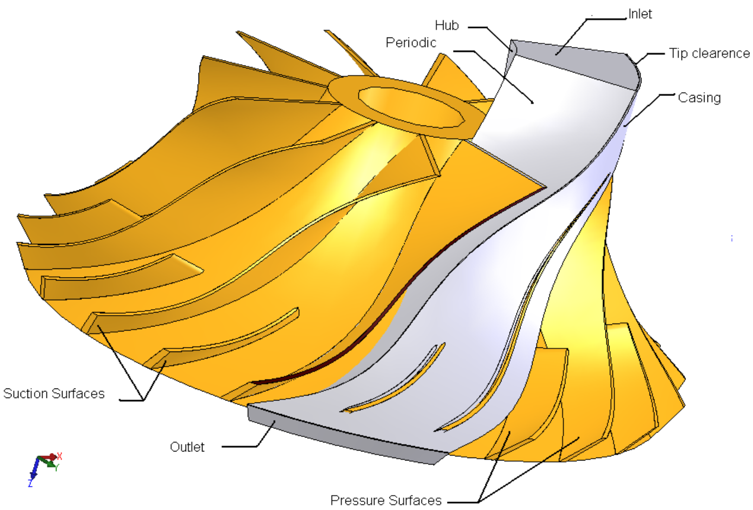


Fig 3: Centrifugal Impeller Model

Vane Type	No of Blades
(Impeller + Splitter) Vanes	19 Impeller + 19 Splitter
Diffuser Vanes	33Diffuser Vanes

Table 1: Blade Details

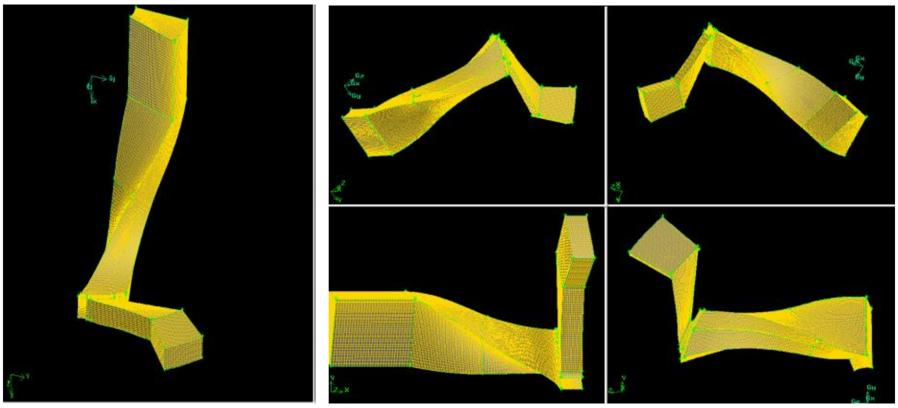


Fig 4: Meshed Centrifugal Impeller using Gambit

Analysis has been carried out for the model with 0.5mm constant tip clearance gap under different shaft rotational speeds namely, the design speed of the compressor is 47413 rpm, and Computations were carried out with six different shaft rotational speeds in order to bring out the influence of the parameters on the flow behavior.

(N/N_d=60%), 25000rpm.
(N/N_d=70%), 30000rpm.
(N/N_d=80%), 35000rpm.
(N/N_d=90%), 40000rpm.
(N/N_d=100%), 47413rpm.

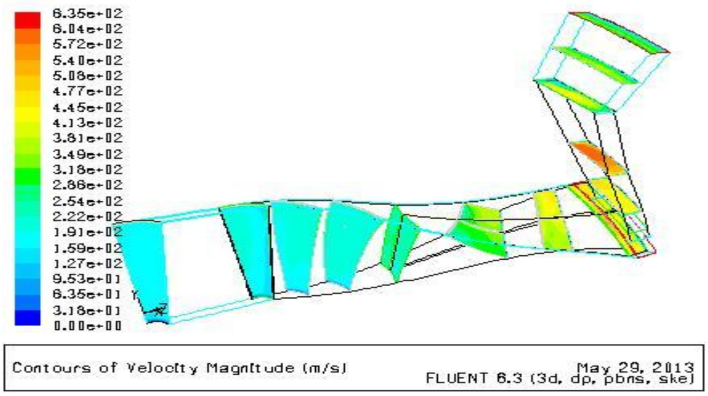


Fig 5: Centrifugal Impeller using Gambit

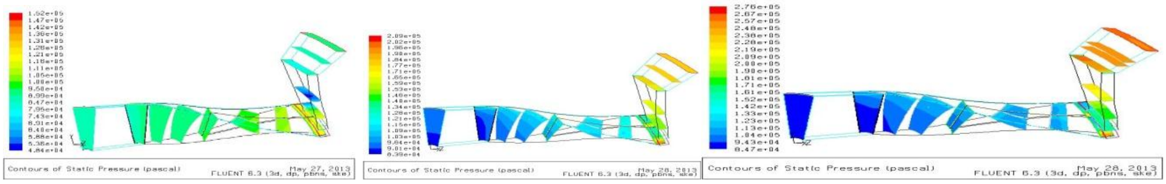
The contours of flow properties in the impeller passages were investigated in this part. The results from two different size models at different rotational speed of identical work coefficients are discussed.

The Mach number distribution on the blade to blade surface presents in the Fig 5.The plots proved that the choke condition did not occur at the design point 1 and 3, and stage can still deliver further mass flow rate. However, at design points 2, and 4 the choke condition occurred, so that the stage would unable to deliver further mass flow unless increasing the inlet back pressure. It can be seen in the plots for design point 3 and 4 that the choke condition occurs at the leading edge of the blade. It proves that there is a performance limit at High speed, due to the choking occurs around the inducer section

The various contours of the flow parameters such as pressure, turbulence, velocity etc. in the flow field for the different models are given below. The red colored regions are the regions where the properties attain their maximum values.

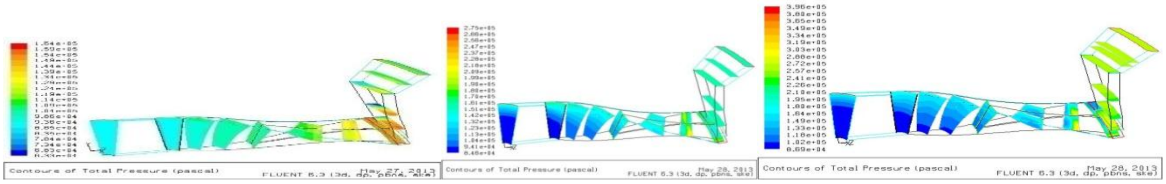
The blue colored regions indicate the regions where the properties are at their minimum. The properties that were analyzed for the various models are as follows:-

1. Static Pressure
2. Dynamic Pressure
3. Total Pressure
4. Velocity Magnitude
5. Velocity Angle
6. Turbulence Kinetic Energy
7. Effective thermal Conductivity



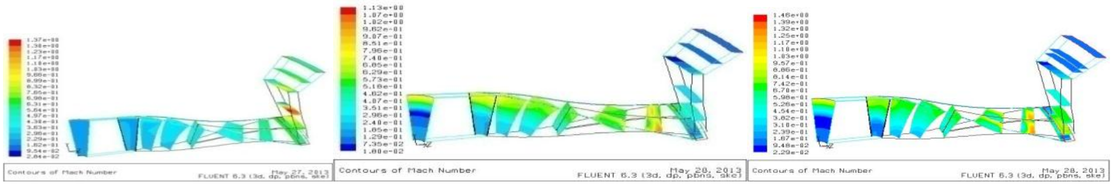
Pressure 0.9 bar 1.6bar 2.6bar

Fig 6: Distribution of static pressure on the planes along the flow path



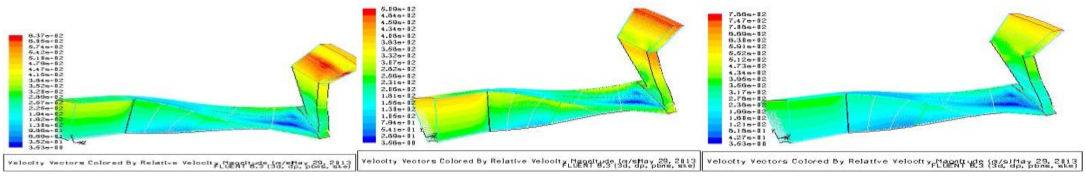
Pressure 0.9 bar 1.6bar 2.6bar

Fig 7: Distribution of total pressure on the planes along the flow path



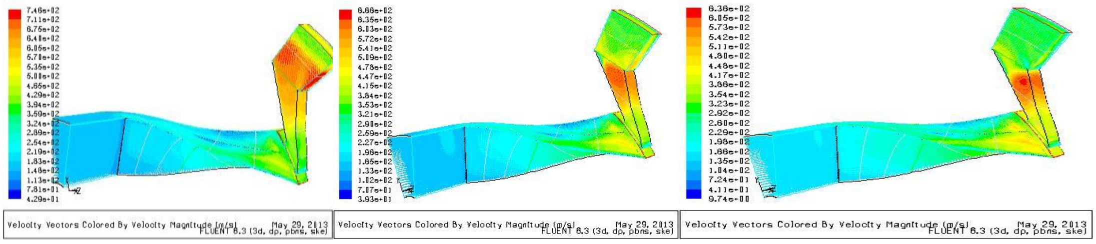
Pressure 0.9 bar 1.6bar 2.6bar

Fig 8: Distribution of Mach no on the planes along the flow



Pressure 0.9 bar 1.6bar 2.6bar

Fig 9: Distribution of velocity vectors along Relative Velocity



Pressure 0.9 bar 1.6bar 2.6bar

Fig 10: Distribution of velocity vectors along Velocity magnitude

It is observed from these contours that the static pressure for a compressor increases along the cascade flow field, while the dynamic pressure and velocity decreases. The total pressure shows a layer-by-layer pattern over the blade profile. The velocity angle at any point in the flow field indicates the direction in which the flow is flowing at that point. The turbulence kinetic energy shows a decreasing value from the inlet. It can be noted that the effective viscosity and effective thermal conductivity is higher at regions of higher turbulence. This can be attributed to the fact that due to turbulence, a greater inter-mixing of the fluid molecules occurs leading to increased resistance to flow, increasing the heat conductivity at the same time.

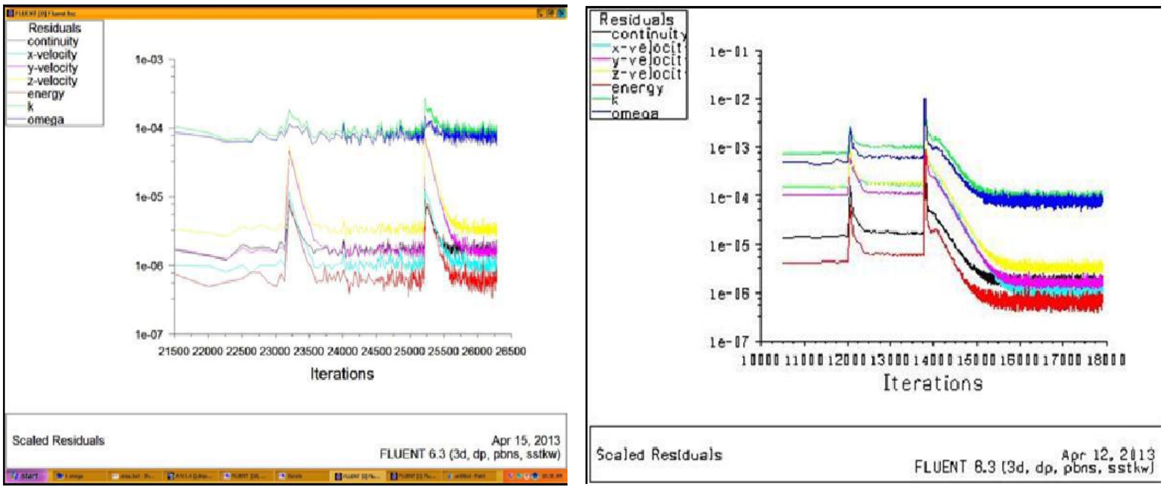


Fig 11: Scaled residuals of K-E model

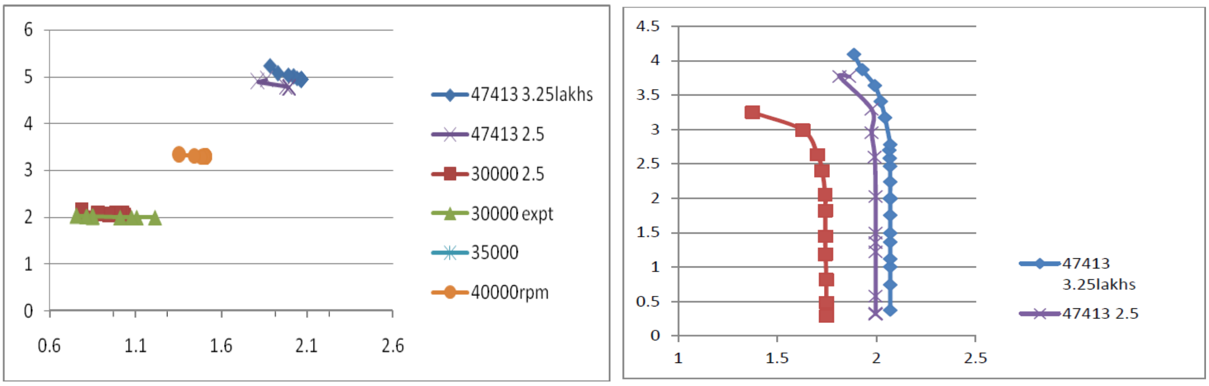


Fig 12: CFD & Experimental comparison between 2.5 lakhs at 47413 /40/30K/35K rpm

3. CONCLUSION

Stage analysis carried out using mixing plane formulation for compressor maximum stage efficiency. At low flow rates the flow reversal has been noticed at the inlet tip. Uniform flow is observed at the impeller exit instead of jet/wake phenomenon satisfying the design approach adopted for generating the impeller vane shape.

From the performance characteristics, it is observed that the impeller static pressure ratio is independent of mass flow parameter for a given speed. As the speed increases, the impeller static pressure ratio increases at a given operating point. As the speed increases, the effectiveness of the diffuser to provide increase in static pressure rise increases. At largest mass flow the static pressure peaks in the vane less portion and then rises constantly till diffuser trailing edge. This behavior is observed in all the speeds, whereas flow rates closer to stall there is a sharp increase in static pressure at all speeds. Magnitude of the peak increases with increase in speed. At low flow the incidence to the impeller inlet will be positive and there is a chance for the flow to separate from them pressure surface, whereas at high flow rates the incidence to the diffuser is negative and there is a chance that the flow can separate from the suction surface. In case of vane diffuser, for a given mass flow parameter the blade loading increases with increase in speed and at a given speed the area within the curve decreases with decrease in mass flow parameter, reaches a minimum point and then rises, the minimum point was very close to the optimum incidence to the diffuser.

ACKNOWLEDGMENT

I would like to thank Dr., professor, Dept. of mechanical engineering, TOCE, Bangalore for his valuable support to develop this project.

REFERENCES

1. *Aerodynamic and geometric optimization for the design of centrifugal compressors Original Research Article International Journal of Heat and Fluid Flow, Volume 6, Issue 1, March 1985, Pages 49-56.*
2. *Koji Nakagawa, Sadashi Tanaka, Junichi Kaneko An aerodynamic investigation of a centrifugal compressor for HCFC123 Original Research Article International Journal of Refrigeration, Volume 15, Issue 4, 1992, Pages 199-205*
3. *U.K. Saha, B. Roy Experimental investigations on tandem compressor cascade performance at low speeds Original Research Article Experimental Thermal and Fluid Science, Volume 14, Issue 3, April 1997, Pages 263-276*
4. *Eckardt D, "Detail flow investigations within a high speed centrifugal compressor", Journal ofFluids Engineering, Trans. of ASME, pp 390-402, 1976.*
5. *Dean R.C. and Senoo Y., "Rotating wakes in vane-less diffusers" Journal of Basic Engg.,*
6. *Trans. ASME, Sept. 1960, PP. 563-574*
7. *Krain H,"Secondary flow measurements with L2F Technique in Centrifugal Compressors" Agard-68 Symposium, Advanced Technology for Aero gas turbine components. May 1987.*
8. *Krain H, "A study on centrifugal impeller and diffuser flow", Journal of Engineering for power, Trans of ASME, Vol. 103, pp 688 to 697,1981.*
9. *Krain H, "Swirling impeller flow", Journal of Turbomachinary,*
10. *performance Analysis on Centrifugal Compressors of Different Dimensions and Variable Operation Speed Yu Wang School of Electrical, Mechanical and Mechatronic Systems A stage calculation in centrifugal compressor C.V. Wallis Z.M. Moussa, GE Aircraft Engines, Lynn, MA 01910B .N. Srivastava,*
11. *Experimental and numerical investigation of the performance of a 240 kW centrifugal compressor with different diffusers Abraham Engeda **
12. *Balje, O. E., "Turbo machines, A guide to design, selection and theory "John Wiley and Sons, Inc., 1981ed*
13. *Barina, F. J., "Comparative performance of vane less diffusers designed with different rates of passage curvature for mixed flow impeller", NACA-TN-1490, November 1947*
14. *Boffo, M., "Gas Turbine Engine having a Mixed flow compressor", US Patent Number: 5094069, March 1992.*
15. *Aungier, R. H., "Centrifugal Compressors – A strategy for aerodynamic design and analysis", ASME publ., New York, 2000*