Comparison of Ansys CFX and Ansys Fluent solver for compressor assembly

Gauri V. Thorat and Laxmikant D. Mangate
Mechanical Engineering Department, Vishwakarma Institute of Technology, Pune, India - 411037

*Abstract :*The aim of the work is to compare the numerical results for axis symmetric model of Compressor using different solvers such as Ansys FLUENT and Ansys CFX using the two different mesh tools such as ICEM CFD and Ansys Mesher. Performance of a compressor in terms of pressure ratio and total to total isentropic efficiency is evaluated. The paper also aims to examine the mesh quality to get the accurate results.

***IndexTerms* - Ansys Fluent, Ansys CFX, ICEM CFD, Ansys Mesher, Compressor**

1. INTRODUCTION

Fluid Dynamics is the study of the fluid flow while Computational Fluid dynamics (CFD) deals with solving the complex fluid problem with the help of numerical methods. Computational Fluid Dynamics is the science of predicting the fluid flow, Heat Transfer, mass transfer and chemical reaction by solving the numerically set of governing mathematical equations by the means of Computer. CFD codes are structured around the numerical algorithm that can tackle fluid flow problem. CFD has gained its popularity with years as it can be employed in solving all kinds of complex problems such as designing the vehicles and improving there aerodynamic characteristics to medical applications like curing arterial diseases to weather forecasting. From the literature it has been observed that the Ansys CFX and Ansys Fluent CFD solvers are reliable solvers and therefore selected for performing the CFD analysis on compressor. Apart from that one of the issue facing today is the differences observed in both solvers. The objective of the paper is to understand the differences in both the solver and also study the physics behind those solvers as well as study the effect of mesh in both the solvers.

Ansys Fluent and Ansys CFX both uses Finite Volume method which discretize the spatial domain using a mesh. Variables such as mass, energy and momentum are stored in this control volumes constructed with the help of mesh. When it comes to finite volume method used for the discretization in both the solvers, CFX uses the vertex centered method, more precisely the dual median method while fluent uses the cell centered method. The basic differences between the two solvers are the location of unknowns. For the present work, computation was performed using Ansys Fluent and Ansys CFX. In case of Fluent analysis, the numerical fluxes were estimated with second order upwind scheme and the turbulence is modeled using k- ω Shear Stress Transport (SST) model while in CFX, fluxes were estimated with High Resolution Scheme method.

2. LITERATURE SURVEY

Daniel Lorstad [1] documented the investigations of differences in Ansys Solvers CFX and Fluent. The paper deals with the Computational Fluid Dynamic study on Siemens gas Turbine (SGT) 800 burner. This is the largest industrial gas turbine. Paper further presents the mesh study and turbulence model study for compressible flow. Two mesh types such as coarse mesh or optical mesh were used and the results were compared by using Ansys CFX and Ansys Fluent CFD solvers. In addition to the above research paper, a Computational Fluid dynamics reference book by John Anderson [2] is also used. This book is regarding the basic concepts of Computational fluid dynamics. It is found to be very helpful to understand the governing equation such as conservation of mass, conservation of energy and conservation of momentum and also focus on solving the CFD related problem with the help of numerical analysis.

Furthermore, a text book mentioning the solution methods of Computational Fluid Dynamics by Andre Bakker [3] is also reviewed. The text book pertains to the methodology of finite volume method and also the discretization method which is used in both Ansys CFX and fluent solver. The book also explained different upwind scheme method such as first order, second order used in Fluent as well as high resolution method used in CFX.

Further the turbulence model is validated using the validation method explained by Brdina et al [4].

3. COMPARISON OF ANSYS CFX AND ANSYS FLUENT

In this work we compared the Ansys CFX and Ansys fluent solver results for compressor assembly. We have divided the compressor assembly into three domain such as impeller domain, cover and inlet domain and did the meshing in ICEM CFD meshing tool and run the mesh in both Fluent and CFX solver.

The results are compared in terms of Compressor overall isentropic efficiency, impeller efficiency, mass flow rate, quality of mesh etc. and observed the result by plotting compressor map in terms of mass flow rate vs Pressure ratio and mass flow rate vs isentropic efficiency.

3.1. Boundary conditions

Table 1. The boundary conditions for CFD analysis

Type	Boundary condition
Inlet	Pressure Inlet
Outlet	Pressure Outlet
Wall	Adiabatic, No slip
Rotating Domain	Speed of rotation
Non Rotating Domain	Counter rotating wall

3.2 Compressor Performance Map

Compressor characteristic is the curve to show the behaviour of fluid like change in temperature, pressure, entropy, flow rate etc at different compressor speed. The function of a compressor is to increase the pressure of a fluid passing through it, so that the exit pressure is higher than the inlet pressure. The compressor curves are plotted between various parameters and some are as follows such as mass flow rate versus pressure ratio, efficiency versus mass flow parameter (MFP), Pressure coefficient versus flow coefficient. Here we have plotted the compressor map for three different speed lines such as 45 krpm, 70 Krpm, 100 Krpm at different section of impeller like impeller outlet, diffuser outlet, critical area and domain outlet. We plotted the graph in both Ansys Fluent and Ansys CFX as per the following,

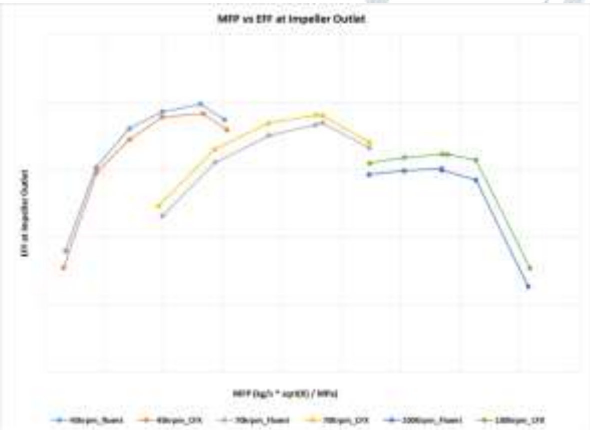

Fig. 1. Compressor map for Efficiency vs. MFP at impeller outlet

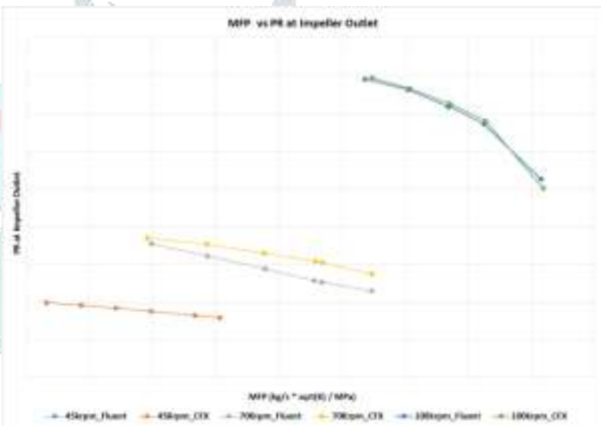

Fig. 2. Compressor map for pressure ratio vs mass flow rate at Impeller Outlet

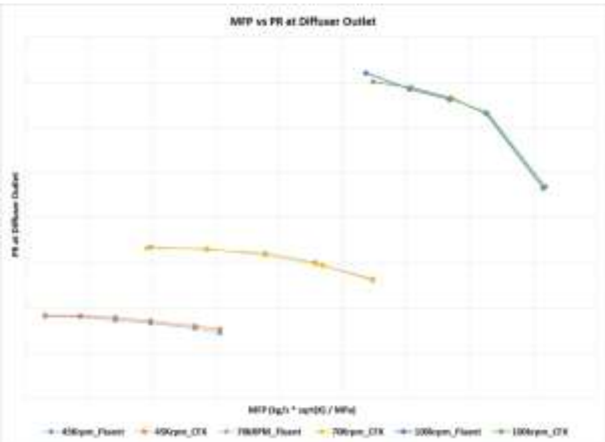

Fig. 3.Compressor map for PR vs MFP at Diffuser Outlet

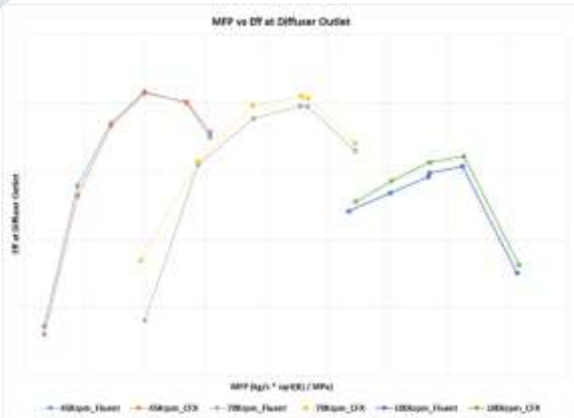

Fig. 4. Compressor map for Efficiency vs MFP at Diffuser Outlet

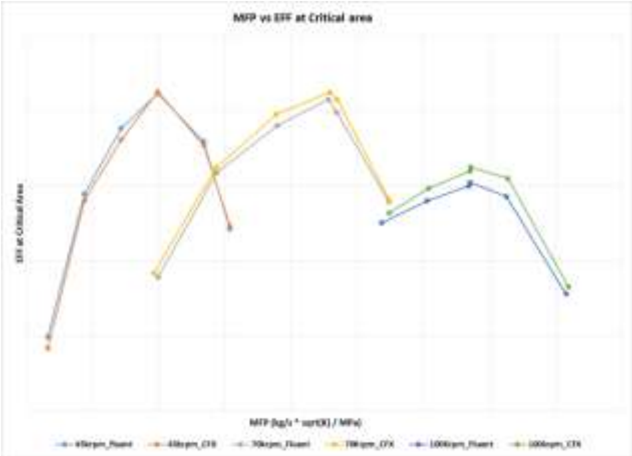

Fig. 5. Compressor map for Efficiency vs MFP at critical area

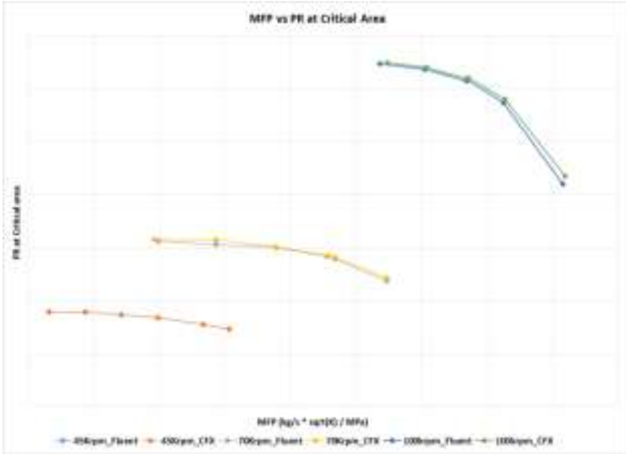

Fig. 6. Compressor map for PR vs MFP at critical area

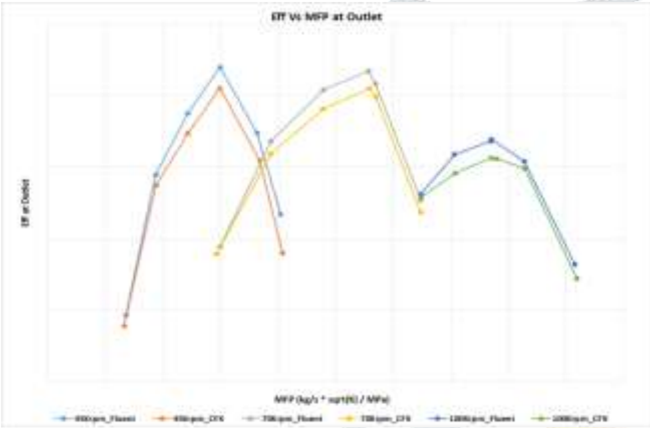

Fig. 7. Compressor map for Pressure ratio vs MFP at domain outlet

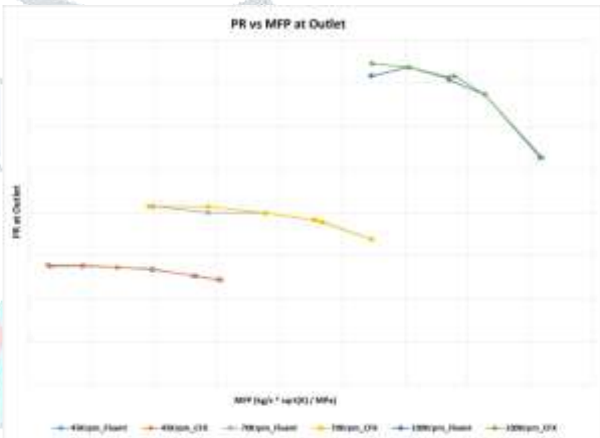

Fig. 8. Compressor map for Efficiency vs MFP at domain outlet

4. DISCUSSION

In this work we have used the two CFD solvers such as Ansys CFX and Ansys Fluent for the compressor stage assembly. In this case we have plotted the graph for three different speed lines such as 45 Krpm, 70 Krpm and 100 Krpm at four different sections such as impeller outlet, diffuser outlet, critical area and domain outlet. The meshing is done using ICEM CFD tool and plotted the compressor map in terms of mass flow parameter vs. pressure ratio and mass flow parameter vs. efficiency. There are point differences observed in results of both solver at each speed line. Here both the Fluent and CFX solver use the finite volume method which discretizes the spatial domain using a mesh.

Various values such as mass, energy and momentum are stored in this control volume which is constructed with the help of mesh. The Fluent solver is based on cell centered discretization method where cell themselves serve as control volume where CFX solver is based on vertex centered discretization method in which control volume is formed by smaller sub control volumes surrounding the vertex and the variable value is stored in vertex and hence Fluent solver is more accurate and very sensitive to mesh quality that is the presence of pyramids in the mesh may change the overall mesh quality.

We observed from the compressor map that the lines of constant speed parameter start to become vertical indicating that the slight increase in mass flow results in rapid decrease in pressure ratio. The overall isentropic efficiency at impeller outlet is higher than the domain outlet because there are losses observed in diffuser outlet and critical area of the impeller due to fluid friction in stationary and rotating blade passages and also because of leakage losses. It was observed that Fluent required more iteration to stabilize and reached convergence than CFX. From above discussion it comes to know that the fluent solver is more accurate than the CFX solver but it requires more time to converge the solution.

5. CONCLUSIONS

The stage outlet overall isentropic efficiency calculated from CFX solver is 64.63% whereas in Fluent it's about 63.89% at 0.077 kg/sec mass flow rate. It shows that the 1.83 points difference in stage outlet efficiency between CFX and Fluent where the impeller outlet efficiency in fluent solver at 0.077kg/sec mass flow rate is 78.97% and in CFX solver is 77.74% so that 1.23 point difference observed in impeller outlet. The large point difference observed in stage outlet efficiency than the impeller outlet efficiency because the losses observed in diffuser outlet and critical area of the impeller in both the solver ,As we discussed earlier the fluent result is more mesh sensitive than the CFX and hence Fluent solver result is more accurate than the CFX solver result.

6. ACKNOWLEDGEMENT

I acknowledge Vishwakarma Institute of Technology, Pune for providing the necessary help during the course of this work.

7. REFERENCES

[1] Investigation of differences in Ansys solver CFX and Fluent: Siemens Industrial Turbo machinery AB: Daniel Lörstad
[2] Computational Fluid Dynamics: The Basics with Applications by John Anderson
[3] Applied computational Fluid Dynamics (solution Methods) by André Bakker
[4] Brdina, J.E., Huang, P.G., Coakley, T.J. (1997), "Turbulence Modeling Validation, Testing, and Development", NASA Technical Memorandum 110446