ISSN : 2455-9679
Impact Factor : 3.805

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"NUMERICAL INVESTIGATION ON PERFORMANCE CHARACTERISTICS ON EFFECTS OF INLET FLOW FIELD CONDITIONS OF CENTRIFUGAL COMPRESSOR"

Gaurav Dubey [1], Shantanu Roy [2]

[1, 2] *Department of Mechanical Engineering, Indore Institute of Science and Technology, Indore, India*

ABSTRACT

A lot of studies have been accomplished to make it extra efficient. With the improvement of Computational Fluid Dynamics (CFD), the parametric have a look at of a centrifugal compressor has become a lot simpler and the complicated internal flow may be properly predicted to expedite the compressor design procedure. The primary goal of this work is to examine the impact of trailing part perspective of twisted blades of a radial float impeller at the hydraulic overall performance of an impeller the usage of 3-dimensional steady-state analysis with the assist of commercial CFD software program Ansys- 16.0. In this analysis, 3 impellers are modeled the usage of Ansys Blade Gen device with specific trailing part angles retaining the blade wrap perspective constant. The calculations are accomplished best at the impeller. The consequences discovered that at layout factor the impeller head will increase through 4% with an growth in blade trailing part perspective from sixteen to 27 degrees. It is located that the enter energy additionally will increase. Tip leakage loss of small scale turbomachines has more impact on the impeller performance than that of large scale ones. Use of 10% tip gap was found to reduce impeller efficiency from 99% to 90%. Because the splitter was located downstream of the impeller leading edge, any incidence at the impeller leading edge leads to poorer splitter performance.

Key Words: *Radial compressor, turbo jet , turbo-prop, turbofan radial inlet, Tip leakage loss, impeller efficiency.*

I. INTRODUCTION

The merit of radial design for turbo machinery has been properly identified in an extensive sort of applications, such as hydraulic turbo machines, small jet engines, helicopters, in process industries, compression of gases and vapours, refrigeration, turbocharger for commercial field and industrial gas turbines. Due to fact that they can provide optimum pressure ratios and big scale of operating ranges with comparatively high efficiencies at same configuration so The operation range can be defined as the range between the choke point and the surge point. A lot of research work going on to increase operating range without too much loss in efficiency.

The maximum working efficiency of all the compressor is less, and a lot of energy is wasted as a result of this. Huge amount of energy may be stored by means of increasing the performance of those compressor.. With the advances in computer technology and more in-depth understanding of unsteady flow phenomenon, it is now feasible to carry out numerical calculations to investigate the flow phenomenon within compressors in an economic way and a reasonable time. Actually, various parameters of the designing the impeller are based totally on empirical formulae derived from experience and thumb rules. Therefore, analyzing the parameters scientifically will serve the twin cause of increasing the performance and additionally organising a systematic technique of designing the impeller. This project was carried out with the help of Ansys CFX software to investigate some unsteady flow patterns and dynamical features under different working conditions. The main objective of this project was to investigate some vital characteristics of centrifugal compressors like compressor efficiency, flow patterns, flow field analysis and pressure fluctuation analysis

ISSN : 2455-9679
Impact Factor : 3.805

numerically. Once those analyses have been fulfilled, the correlations among the ones crucial traits and the compressor performance may be established.The maximum risky flow instability taking place in centrifugal compressors is the surge. It takes place whilst the mass flow charge is reduced underneath its critical value. The surge is called low-frequency fluctuations of the pressure and the mass flow charge that during maximum extreme instances would possibly even result in opposite flows in the compression device [1]. Surge usually results in huge decrease of the system overall performance however is likewise able to destroying compressor in only few seconds. This is because of the reality that flow oscillations generate excessive thermal and mechanical masses acting on device structure and additionally growth blades vibration degree which may be cause of a fatigue fracture [2]. Moreover, the surge is a international instability affecting entire compression device which include different additives of industrial rig like pipes, reactors etc. [3]. Undisturbed operation of the compressor is a key in producing incomes in maximum of industries and that is why heading off or suppressing of this phenomenon is essential. The putting of so called the surge margin is the maximum extensively used technique of prevention from the surge. Compressor working variety is constrained with the aid of using automatized device which continues operational factor sufficiently a ways from the surge onset. It is normally set someplace near 10%-15% far from the surge restriction. This outcomes in narrowing the operation variety of the gadget. It is likewise uneasy to analyze surge on actual machines due to its damaging nature and the surge restriction is in lots of instances set primarily based totally on theoretical calculations. This leaves a area for CFD evaluation that is probably more appropriate for that application. Numerical evaluation lets in to analyze the surge margin with out inflicting threat of machine failure. Nevertheless, it's far very at risk of non-bodily situations which include the turbulence version, timescale putting and lots of different simulation parameters. It is likewise now no longer clean which boundary situations are high-quality in reproducing the situations present withinside the compression machine stricken by a international flow instability which include the surge.

II. CFD Evaluation of An Unstable Centrifugal Compressor Operation

In literature, numerous papers thinking about CFD evaluation of an unstable, near-surge compressor operation may be found. Most usually it's far an unsteady RANS (URANS) simulation applied in commercial codes like ANSYS CFX or Fluent, however, the inlet and the outlet boundary situations range in huge variety. In [4] set of obstacles endorsed with the aid of using CFX modelling guide [5] has been used. Those settings consist of steady mass flow rate at the inlet and consistent static pressure on the outlet of the compressor. Advantages of such method are robustness and complete manage of operational factor even as they unnecessarily stiffen the simulation and might introduce uncertainties. Those settings were in part reversed in [6] in which the mass flow fee is about at the outlet and on the inlet the whole pressure is about to maintain actual strength level of inflowing air. In this situation the mass flow may also nevertheless reveal a few unnatural flow systems however the inlet boundary guarantees higher strength conservation. The simulations with out direct solving of the mass flow rate were defined withinside the literature. For example, in [7], the whole pressure is about on the inlet, even as the static pressure is about at the outlet. In [8] the outlet boundary with stress based of plenum mathematical model is used at the outlet in pair with general pressure boundary on the inlet. Despite the reality that there's no direct putting of the mass flow fee withinside the CFX model, the consistent mass flow fee is described as an outlet boundary in equations describing the plenum. Another course of gaining knowledge of near-surge working situations is through in-residence made codes like the ones supplied in [9] . In all of these papers the whole pressure is about because the inlet condition. In [10] the static stress is described at the hole even as in [11] extra state-of-the-art stress feature depending on plenum model has been used. In neither of referred to papers the outlet plenum has been modeled and in simplest few of them the inlet pipe turned into added. Very frequently the boundary is about now no longer even on the quit of volute however at the hole of diffuser. Only one of these research contained surge evaluation [12] and the last ones - the near-surge unsteady flows. It is due to the reality that it's far tough to go into the surge location with out the mass flow definition at inlet/outlet boundary. This is why the ones simulations do now no longer offer clean statistics whether or not they're appropriate for the surge investigation. Next to RANS simulations additionally few LES simulations were performed [13]. All of them had mass flow fee at inlet and static pressure at outlet as boundaries. Also inlet and outlet pipe turned into covered in simulations. The precision of this sort of simulation is taken into consideration as a sizeable benefit over different styles of simulations however they're rather time and aid ingesting which makes them unpractical for normal use.[14] On single stage with numerical Unsteady simulation found that the flow field fluctuation is remarkable only in the semi- vaneless gap, whereas the diffuser blade channel is not substantially affected by this phenomenon. On multistage vanless through CFD AND MADE Simplest

ISSN : 2455-9679
Impact Factor : 3.805

assumption of axial uniform inlet flow leads to overestimation of the impeller efficiency by about 1~1.5% and does not provide the correct positioning of the operating envelope.[16] The overall diffuser pressure recovery coefficient, based on suitably averaged inlet total pressure, was found to correlate well with momentum-averaged flow angle into the diffuser. The pressure recovery coefficient was found to be essentially independent of the axial distortion at diffuser inlet and the Mach number, over the wide flow range investigated. The generally accepted sensitivity of diffuser pressure recovery performance to inlet flow distortion and boundary layer blockage can be largely attributed to inappropriate quantification of the dynamics pressure at the inlet. The two types of diffusers, discrete-passage and straight-channel diffusers, showed similar behavior regarding the dependence of pressure recovery on diffuser inlet flow angle and the insensitivity of the performance to inlet flow field axial distortion and Mach number.

III. Computational Model Parameter

A 3d axi-symmetric computational domain was considered, the initial design parameters for pump impeller. This was obtained by vista CCD and blade gen software. Here the grid arrangement for impeller is shown below. After meshing of the model of pump assembly commercial CFD code CFX-16 is used for simulation of the compressor performance. The boundary conditions are applied. The performance results are obtained at given mass flow rate conditions with constant speed by taking turbulent modeling. The numerical simulation is checked in centrifugal pump and to get safe range of operating at given flow rate and operating speed.

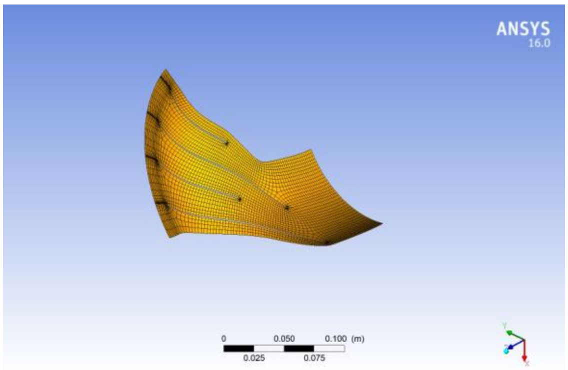

Figure 1 Mesh elements of Impeller

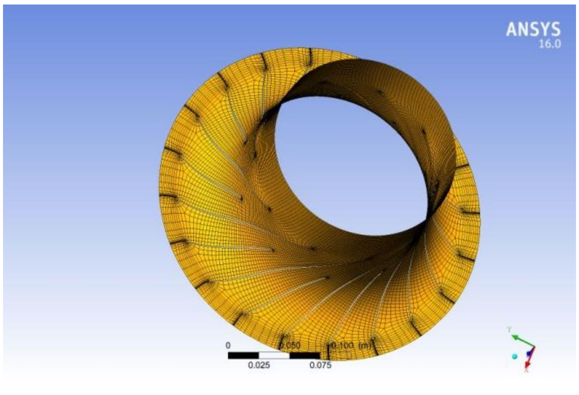

Figure 2 Mesh elements of centrifugal Compressor

Assumptions

The simulation of flow inside the centrifugal pump is done on basis of following basic assumptions:

- Steady state condition.
- Constant fluid properties.

ISSN : 2455-9679
Impact Factor : 3.805

- Incompressible fluid flow.

The walls were assumed to be smooth hence any disturbances in flow due to roughness of the surface were neglected

IV. RESULTS AND DISCUSSION

The rotating speed of the centrifugal compressor is maximum, as a result, the blade velocity at the eye of the impeller (U) is much higher than that of a conventional impeller.

The velocity at the internal wall is higher than the outer wall. Secondary flow reasons more low momentum flow to move from the outer to internal sides. Figure 5.10 suggests the tangential velocity at the IGV exit. The tangential velocity is higher at the hub side compare to the shroud side and is a result of the conservation of angular momentum.

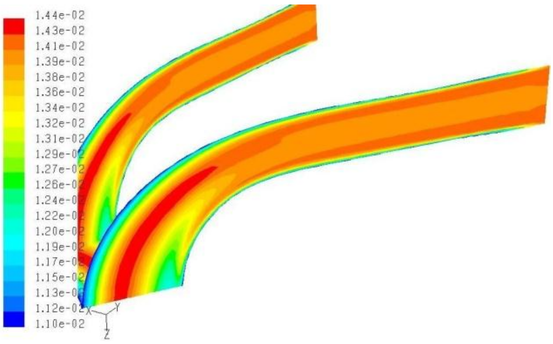

Figure 3 Total pressure

The pressure loss coefficients of the IGV from simulation of the three geometry arrangements are listed in Table 1.

Geometry	Losses included	Pressure loss coefficient
i	All losses and loss due to interaction between IGV and impeller.	19.1%
ii	Friction loss of the wall and the vanes, leading edge separation loss, mixing loss and turning bend loss.	17.05%
iii	Friction loss of the wall and turning bend loss,	14.1%

After comparing the results from the three geometrical arrangements, the following can be found. Because of the interaction, the flow separation at the IGV exit vanishes. This adds some benefit to the IGV performance. Therefore, the loss due to the interaction between the IGV and impeller is a bit more than 2%, which is more than 10% of the total pressure loss. The friction loss due to the vanes, leading edge separation loss, and mixing loss is 2%, which is an additional 10% of the total pressure loss. The main loss is due to the turning bend loss and friction loss of the wall, which is 14.1%. Therefore, more than 70% of the total pressure loss of the IGV is due to the turning bend loss and friction loss of the wall. The Reynolds number of the IGV based on the inlet condition is about 3500. The flow is highly turbulent. Accordingly, the friction loss is low and the loss in the IGV is mainly due to turning bend loss. The overall IGV pressure loss coefficient is 19.1 %, which is acceptably high. Nevertheless, if higher IGV performance is needed, it is recommended to reduce the turning bend loss by changing the IGV shape.

ISSN : 2455-9679
Impact Factor : 3.805

V. Stage Performance Results

The following table(s) give a summary of the performance results for each stage.

Table 2 . Stage 1 Performance Results

Rotation Speed	-4188.7900	[radian s^-1]
Inlet Mass Flow Rate	4.3402	[kg s^-1]
Inlet Volume Flow Rate	3.5467	[m^3 s^-1]
Reference Radius	0.1227	[m]
Input Power	161083.0000	[W]
Inlet Flow Coefficient	0.1146	
Total Pressure Ratio	2.5929	
Total Temperature Ratio	1.4123	
Polytropic Head	95817.8000	[J kg^-1]
Polytropic Head Coefficient	0.3628	
Total-to-Total Polytropic Efficiency %	80.2701	
Diffuser Loss Coef., Y1	0.1073	
Diffuser Loss Coef., Y2	0.1203	
Diffuser Loss Coef., Y3	0.1704	
Diffuser Loss Coef., Y4	0.2050	
Diffuser Cp	-0.0017	

At the exit of the impeller, there is another 90 degree turning bend to the axial direction diffuser. The static pressure increases. The shroud side flow, which already separates, turns to the outside of the bend, which makes the situation even worse. As one can see, the loss in the third bend of the compressor reaches the peak. Both inner and outer have a great deal of loss. The pressure loss due to the turning bend and the wall friction is 30.8% based on inlet energy level.
This report summarizes the results of a CFD analysis performed for the centrifugal compressor geometry shown in Figure 1. In the following sections both quantitative and qualitative results are presented in the form of tables, charts and plots.

VI. Conclusion

CFD simulation predicts performance of the miniature centrifugal compressor with consideration of the interaction between blade rows. In this following conclusion has been drawn

- It is found that the interaction between the IGV and impeller is due to potential interaction. The IGV loss coefficient increases 2% due to the interaction. Upstream components are rarely affected by the performance of downstream com nents. ke interaction between the impeller and diffuser is time dependent. However, the performance of the diffuser is not a function of time. Therefore, the only concern of interaction is diffuser vane vibration due to the transient blade

- When the size goes down, precision loss and friction loss increase. A study of tip leakage shows there is 10% loss of efficiency with a 10% tip clearance, which is high compared to 4% loss in a conventional compressor.

ISSN : 2455-9679
Impact Factor : 3.805

REFERENCES

[1] Willems F and Jager B de 1998 Modeling and control of rotating stall and surge: an overview IEEE Int. Conf. Control Appl. 1 331–5.

[2] Meuleman C, Willems F, de Lange R and de Jager B 1998 Surge in a low-speed radial compressor Int. Gas Turbine Aeroengine Congr.

[3] de Jager B 1995 Rotating stall and surge control: A survey Decision and Control, 1995., Proceedings of the 34th IEEE Conference on pp 1857–62.

[4] Shahin I, Gadala M, Alqaradawi M and Badr O 2014 Unsteady CFD Simulation for High Speed Centrifugal Compressor Operating Near Surge Volume 2D: Turbomachinery (ASME) .

[5] ANSYS 2016 CFX-Solver Modelling Guide--Release 17.0.

[6] Ding M Y, Groth C, Kacker S and Roberts D 2005 CFD Analysis of Off-design Centrifugal Compressor Operation and Performance 2006-Int-ANSYS-Conf 252.

[7] Dickmann H-P, Secall Wimmel T, Szwedowicz J, Filsinger D and Roduner C H 2006 Unsteady Flow in a Turbocharger Centrifugal Compressor: Three-Dimensional Computational Fluid Dynamics Simulation and Numerical and Experimental Analysis of Impeller Blade Vibration J. Turbomach. 128 455.

[8] Guo S, Chen H, Zhu X and Du Z 2011 Numerical Simulation of Surge in Turbocharger Centrifugal Compressor - Influence of Downstream Plenum Proc. ASME Turbo Expo GT2011-451 1–12.

[9] Niazi S, Stein A and Sankar L 1998 Development and Application of a CFD Solver to the Simulation of Centrifugal Compressors AIAA Pap. 934.

[10] Stein A, Niazi S and Sankar L N 2000 Numerical analysis of stall and surge in a high-speed centrifugal compressor 38th Aerospace Sciences Meeting and Exhibit p 226.

[11] Stein A, Niazi S and Sankar L N 2001 Computational Analysis of Centrifugal Compressor Surge Control Using Air Injection J. Aircr. 38 513–20.

[12] Stein A, Niazi S and Sankar L 2000 Numerical studies of stall and surge alleviation in a highspeed transonic fan rotor 38th Aerospace Sciences Meeting and Exhibit (Reston, Virigina: American Institute of Aeronautics and Astronautics) .

[13] Hellstrom F, Gutmark E and Fuchs L 2012 Large Eddy Simulation of the Unsteady Flow in a Radial Compressor Operating Near Surge J. Turbomach. 134 051006.

[14] Shahin I, Gadala M, Alqaradawi M and Badr O 2015 Large Eddy Simulation for a Deep Surge Cycle in a High-Speed Centrifugal Compressor With Vaned Diffuser J. Turbomach. 137.

[15] Semlitsch B, Jyothishkumar V, Mihaescu M, Fuchs L and Gutmark E J 2013 Investigation of the surge phenomena in a centrifugal compressor using large eddy simulation ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE) vol 7 A pp 1–10.

[16] Benini, E., Toffolo, A. (2003), “Centrifugal Compressor of A 100 KW Microturbine: Part-2 NumericalStudy of Impeller-Diffuser Interaction”, ASME paper, GT-2003- 38153

ISSN : 2455-9679
Impact Factor : 3.805

[17] Bonaiuti, D., Arnone, A., Milani, A., (2003), "Aerodynamic Analysis of a Multistage Centrifugal Compressor", ASME paper, GT-2003-38495.

[18] Filipenco, V.G., Deniz, S., Johnston J.M., Greitzer, E.M., and Cumpsty, N.A., (2000), "Effects of Inlet Flow Field Conditions on the Performance of Centrifugal Compressor Diffusers: Part 1-Discrete-Passage Diffuser; Part 2-Straight-Channel Diffuser", ASME Journal of Turbomachinery, Jan. 2000, Vol. 122, pp. 1-21.

[19] Hillewaert, K., Braembussche, R.A.V.D. (1999) "Numerical Simulation of Impeller-Volute Interaction in Centrifugal Compressors", ASME Journal of Turbomachinery July 1999, Vol. 121, pp. 603-608.

[20] Tamaki, H., Nakao, H., and Saito, M., (1999), "The Experimental Study of Matching Between Centrifugal Compressor Impeller and Diffuser", ASME Journal of Turbo-machinery, Jan. 1999, Vol. 121, pp. 113-118.

[21] Ubaldi, M., Zunino, P., Ghiglione, A., (1998), "Detailed Flow Measurements With Impeller and the Vaneless Diffuser of a Centrifugal Turbomachine", Experimental Thermal and n the Fluid Science, Vol. 17, pp. 147-155.

[22] Ziegler, K.U., Gallus, H.E., and Neihuis, R. (2003), "A Study on Impeller-Diffuser Interaction-Part I: Influence on the Performance & Part II: Detailed Flow Analysis", ASME Journal of Turbomachinery, Jan. 2003, Vol. 125, pp. 173-192.

[23] Y. K. P. Shum,C. S.Tan, andN. A. Cumpsty, ―Impeller-diffuser interaction in a centrifugal compressor,‖ Journal of Turbomachinery, vol. 122, no. 4, pp. 777–786, 2000.

[24] A. Akhras, M. El Hajem, J.-Y. Champagne, and R.Morel, ―The flow rate influence on the interaction of a radial pump impeller and the diffuser,‖ International Journal of Rotating Machinery, vol. 10, no. 4, pp. 309–317, 2004.