ISSN : 2455-9679
Impact Factor : 3.805

IJRTSM
INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"NUMERICAL INVESTIGATION OF RADIAL COMPRESSOR WITH ANSYS CFX- A REVIEW "

Gaurav Dubey [1], Shantanu Roy [2]

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

ABSTRACT

Now a day's centrifugal compressor are preferable equipment for modern production because of space limitation, the compressor is composed of a radial inlet guide vane, a radial impeller and an axial-direction diffuser (which reduces the radial size because of smaller diameter). 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. it is found that when the rotational speed increases the pressure at the suction side decreases. At the high rotational speed over the design point the pressure in the suction side became below the vaporization limits causes cavitations.

Key Words: *Centrifugal compressor, radial inlet, CFD software, rotational speed.*

I. INTRODUCTION

In the past 40 years, the radial compressor becomes more popular for use in process industries, aviation oil and gas that were growing in size. During the Second World War in 1939 as the part of early stage of design of gas turbines it was used by American and British fighter aircrafts. After further development in centrifugal compressor for air compression, a big scale manufacturing has been started for turbo jet , turbo-prop, turbofan and auxiliary power units due to higher compression ratio in first stage and robustness in case of foreign object damage.

Now a day's centrifugal compressor are preferable equipment for modern production. Apart from that the main component of centrifugal compressor, the impeller as it is used to convert the kinetic energy into pressure energy. It plays a vital role in centrifugal compressor performance. An impeller is a rotor which is provided with a sequence of backward curved vanes. It is positioned on a shaft that is couple to on outside source of energy which imparts the required energy to the impeller there by making it to the rotate.

Dating back to the 5th century the wooden double curved blades of centrifugal impeller was discovered in san Dominigos in copper mines in 1772. Centrifugal fans were used for mine air flow as early as in the 16th century. The invention of the centrifugal impeller is a disputed problem whether or not the credit score is going to Leonardo Da Vinci (1452 - 1519), who counseled the concept of the use of centrifugal force for lifting liquid, or to Johann Jordan approximately 1680. Most area the foundation of the centrifugal impeller with Denis Papin in 1689. The significance of Papin's contribution lies in his expertise of the idea of making a forced vortex inside a circular, or spiral casing by way of blades. Following Papin, Kernelien Le Demour in 1732 and Daniel Gabriel Fahrenheit in 1736 defined different designs for centrifugal impellers, however there's no proof in their realistic use. Euler offered in his 1754 memoir an idealized theoretical application of Newton's law to centrifugal impellers, primarily based totally on a conceptualization of his tubular turbine run backwards and now universally referred to as the "Euler equation". His publication induced a extremely good development of hydraulic turbines withinside the 18th century, however did little to steer the

ISSN : 2455-9679
Impact Factor : 3.805

improvement of centrifugal impellers, which needed to regularly expand thru tedious cut-and-attempt methods. The one factor Euler contributed become to provoke a real mathematical inquiry into the employment of centrifugal force as a way of elevating fluid. About the identical time as Euler, John Smeaton delivered in 1752 the examine of turbomachinery through models. He additionally described energy as equal to the charge of lifting of a weight, a idea this is nonetheless essential in thermo-fluids. Today each centrifugal compressors and compressors have reached performance ranges above 90% and are constructed in sizes from some Watts to Megawatts. The centrifugal impeller in compressors and compressors has once in a while helped withinside the improvement of every other.

The analytical/empirical strategies from the length of 1950-1970 may be seemed lower back upon in light of the effective computational strategies now available. Horlock et al [1] suggested that indexed 4 examples: flow deviation, annulus wall blockage, tip clearance flows, and secondary flows. Interested readers who need to understand approximately the information of those analytical strategies can talk over with their paper. Till the early Nineteen Eighties whilst completely third-dimensional (three-D) strategies first have become available, the definition of blade-to-blade (S1) and hub-to-tip (S2) flow surfaces ruled the subject (quasi-third-dimensional Q3D). This approach become first delivered via way of means of Wu (1951). This eliminates the modeling assumptions of the quasi-third-dimensional (Q3D) method however calls for far extra computer electricity and so become now no longer usable as a habitual layout device till the overdue Nineteen Eighties as suggested by Denton et al[2]. Early strategies needed to use coarser grids that delivered large numerical mistakes than withinside the Q3D method. As pc era has been swiftly growing in latest years, CFD simulations which contain a big amount of grid factors have come to be feasible. For instance, 12 blade rows, with 70 000 grid factors utilized in every row, may be calculated on a cutting-edge notebook in run instances at the order of 24 hours. Experimental size of those flows is tough due to the fact their period scale is frequently very small. CFD prediction may be very depending on the turbulence modeling withinside the codes. Regarding the exercise of referring to CFD to numerous crucial aerodynamic standards in turbomachinery, Casey et al [3] concluded 8 factors: keep away from negative occurrence onto blading, lessen friction on surfaces, keep away from kinetic strength loss, keep away from flow separation, offer a uniform distribution onto downstream blades, look at classical blade loading standards, and decrease the pre-surprise Mach number. Some important troubles associated with the cutting-edge country of turbomachinery CFD, along with validation and requirement, have been addressed by Hirsch et al [4]

Centrifugal compressors have severa programs starting from aircraft and process industries to oil and gas industries. This kind of compressor is broadly used because of excessive single level pressure ratio, easy structure, lengthy existence and different beneficial characteristics [5]. Centrifugal compressors are frequently required to keep or enhance gas circulate the pipeline system, for the reason that properly flow and reservoir pressure have a tendency to lower over time. These appear because of depletion arise at reservoir. so, such type of compressor react as re-injection for the cause of reservoir pressure maintenance. Generally, the gas extracted from the oil wells composed of many moisture contents inclusive of water vapour. However, water vapour or moisture content material in gas is one of the motives why overall performance of centrifugal compressor drops. Most not unusual place contaminants which result in mechanical failure are carbon dioxide (CO2) and hydrogen sulphide (H2S), specially for components that synthetic in low alloy steel. These carbon dioxide (CO2) end up a risky contaminants while is related to water vapour or and excessive strain. Combination of carbon dioxide (CO2), liquid water and excessive pressure ends in the formation of carbonic acid (H2CO3), that could closely corrode the compressor component [6]. Besides, the impact of hydrates formation can result in the compressor's main aspect blocking, can lessen the pipeline flow capacities and has capacity to harm different system system which includes clear out out and valves [7]. In order to manipulate or save you those problems, moisture content material dimension in natural gas is important. These phenomenon have an effect on the performance of compressor as quantity manufacturing of natural gas in keeping with cycle lower [8]. Designation of impeller is one of the answers for DEC problem. For example, enhancing the blade angles of impeller [9], selecting distinct blade inlet and backward sweep of the impeller [10] and change of impeller geometry in time period of skew angle [11]. Another method is to dry the gas itself. There are diverse techniques to dry compressed air or fueloline which includes refrigerated dryers, chemical dryers, desiccant dryers and membrane kind dryers. In this study, we pick out membrane kind dryer because the mechanism to get rid of moisture content material from the fueloline. In the actual application, the mechanism of this type of dryer is while the compressed air is shifting thru a package deal of polymer hollow fibres, the polymer hole fibres entice the water vapour internal earlier than it diffuse into thin selective layer till it attain outdoor of membrane layer [12].

ISSN : 2455-9679
Impact Factor : 3.805

Razali et al [13] proposed to layout a centrifugal compressor with a membrane kind dryer to lessen moisture content material of a gas. The effects display that the outcomes of strain on dew point temperature of the gas alternate the composition of its moisture content material, wherein excessive price lead greater condensation to arise.
From the above literature survey, the investigation can be performed further in the following areas.

1. Detailed study of a model of the centrifugal Compressor and also consists of the detailed to identify, observe and determine complex internal flows in fluids impellers to speed up the compressor design procedure.
2. Design a blade by considering leading edge angle and trailing edge angle to enhance the performance.
3. Identify the Variations in impeller head and relative flow angle at the outlet.
4. Design and develop modified blade by taking equal divisions and varying vane inlet angle from hub to shroud.

II. MATHEMATICAL FORMULATION

Mathematical modeling is typically relevant to the evaluation of engineering systems, that are frequently very complicated. For a normal fluids system, this complexity arises in particular because of the time dependent, multidimensional nature of the fluid flow and the complicated supplementary situations that govern those systems.

• Physical model

The problem being considered is the centrifugal compressor with 10 blades for simulation.

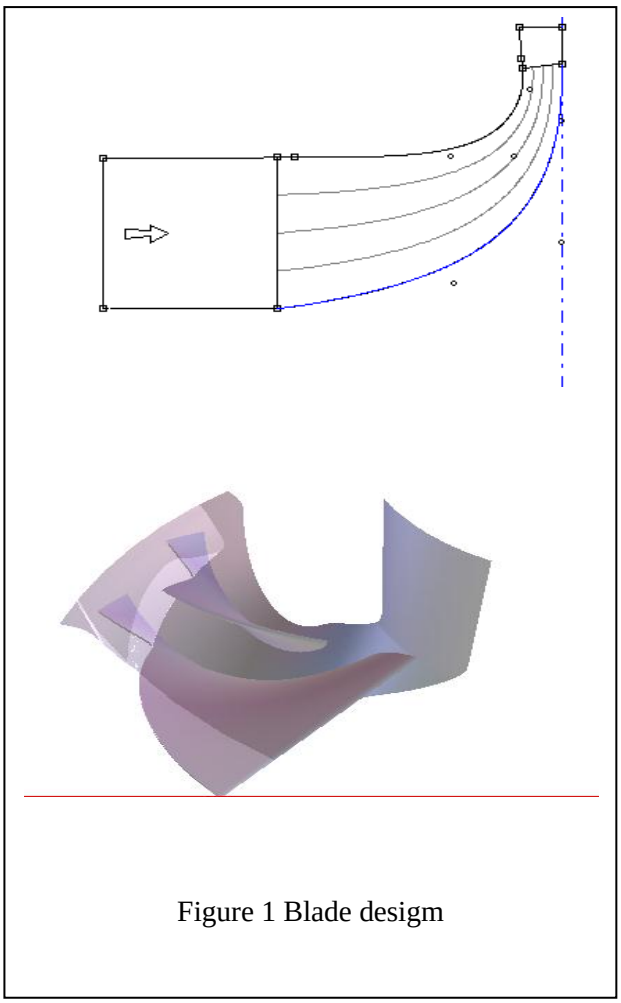

Figure 1 Blade desigm

ISSN : 2455-9679
Impact Factor : 3.805

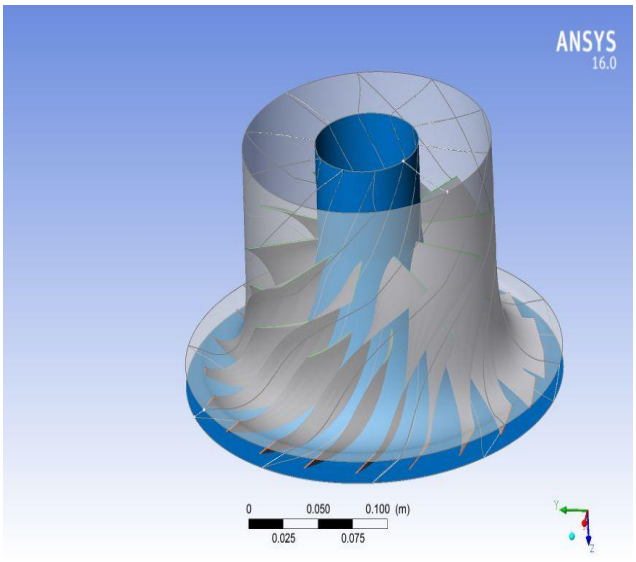

Figure 2: Physical model of Centrifugal compressor

III. SIMULATION RESULTS

The flow angles Alpha and Beta are relative to the meridional plane; a positive angle implies that the tangential rate is that the same direction because of the machine rotation

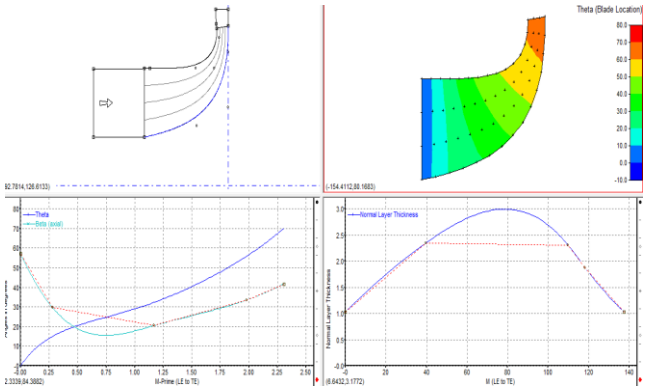

Figure 3 Blade Profile

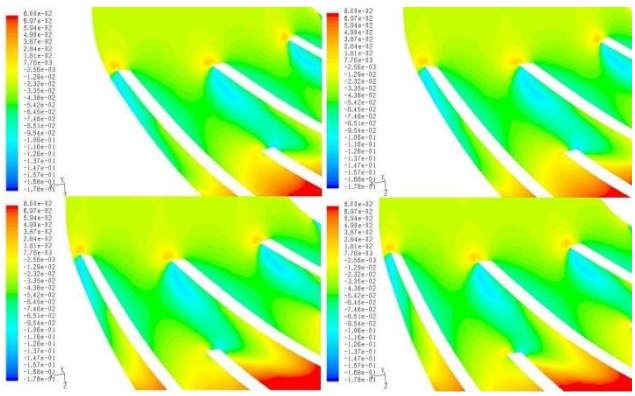

Figure 4 Relative total Pressure (Impeller eye)

Figure 4 depicts the contour of relative total pressure at the leading edge of the impeller blade versus time. There is a huge amount of total pressure loss at the leading edge of the impeller blade close to the pressure side of the blade. This

ISSN : 2455-9679
Impact Factor : 3.805

loss region extends to the leading edge of the splitter, which is located downstream of the leading edge of the impeller. Curvature reasons the flow on convex surfaces to be stabilized with a reduction in turbulent shear stresses and to be de-stabilized on concave surfaces with an increase in turbulence shear stress. Therefore, turbulent energy switch in the direction of the wall is enhanced, energy dissipation is increased and the velocity gradient close to the wall is steep, permitting near-wall flow to resist unfavorable pressure gradients. The converse situation exists on the internal wall. Energetic particles have a tendency to be displaced away from the internal wall so that low energy fluid accumulates near the wall. Energy transfer is decreased, the speed gradient away from the wall turns into shallow, energy dissipation is decreased and the near-wall is unable to withstand unfavorable pressure gradients. The effect of the secondary flows is to re-energize the internal wall regions. When the inlet flow is nearly uniform, the secondary flows may be too weak to prevent flow separation.

IV. CONCLUSION

The centrifugal compressor is composed of three components, the radial IGV, radial impeller and axial diffuser. Fluid details are inaccessible by current available experimental methods. CFD simulation predicts performance of the miniature centrifugal compressor with consideration of the interaction between blade rows. In this following conclusion has been drawn

- The pressure increases gradually along the stream wise direction from minimum value at the suction side to the maximum value at leading edge.
- When the rotational speed increases the pressure at the suction side decreases

At the high rotational speed over the design point the pressure in the suction side became below the vaporization limits causes cavitation and wake region.

REFERENCES

[1] Horlock and Denton, J. D., (2005), “A Review of Some Early Design Practice Using Computational Fluid Dynamics and a Current Perspective”, ASME Journal of Turbo-machinery, Vol. 127 Jan. 2005, pp. 5-13.

[2] Denton, J. D. and Dawes, W. N., (1999) “Computational fluid dynamics for turbomachinery design”, ImechE, 1999, pp. 107-124.

[3] Casey, M., and Innotec, S., (2003), “Best Practice Advice for CFD in turbomachinery Design”, QNET-CFD Network Newletter Dec. 2003, Vol. 2, No.3, pp. 35-37.

[4] Hirsch, C., and Demeulenaere, A., (2003), “State of The Art in the Industrial CFD for Turbomachinery Flows”, QNET-CFD Network Newsletter Dec. 2003, Vol. 2, No.3, pp. 5-9.

[5] Sivaji Ganesh C, NagpurwalaCh and Bhaskar Dixit Q H 2010 Effect of leading edge sweep on the performance of a centrifugal compressor impeller SAS Tech J. 9 pp 55–62.

[6] Guedes AV, de Carvalho Ferreira R, et d’Audenhov F D N and da Silva Marques B 2013 CO2 corrosion on natural gas compressors: criteria and guidelines Proc. of the 42nd Turbomachinery Symp.

[7] Chung D D L 2001 Materials for thermal conduction Appl Therm Eng 21 pp 1593–1605.

[8] Kolass R and Parker C n.d.Moisture measurement in natural gas accessed 11 Nov 2016 Retrieved from: http://www.iceweb.com.au/ .

ISSN : 2455-9679
Impact Factor : 3.805

[9] Röyttä P, Grönman A, Jaatinen A, Turunen-Saaresti T and Backman J 2010 Effects of different blade angle distributions on centrifugal compressor performance Int. J. Rotating Machinery 2009 pp 1–9.

[10] Tough R A, Tousi A M and Ghaffari J 2010 Improving of the micro-turbine's centrifugal impeller performance by changing the blade angles Int. Conf. on Computational & Experimental Engineering and Sciences 14 pp 1–22.

[11] Arunachalam V C, Nagpurwala Q H, Deshpande M D and Shankapal S R 2008 Numerical studies on the effect of impeller blade skew on centrifugal compressor performance SASTECH J. 7 pp 33–40.

[12] Fish A 2017 The importance of drying compressed air Parker Hannifin accessed 28 June 2017 Retrieved from: http://blog.parker.com/facts-you-need-to-know-about-drying-compressed-air.

[13] M A Razali , M F Zulkafli , N Mat Isa and Z Subari, Numerical analysis on centrifugal compressor with membrane type dryer, IOP Conf. Series: Materials Science and Engineering 243 (2017) 012017 doi:10.1088/1757-899X/243/1/012017.

[14] Du, L. M., & Gong, Y. Z. (2013). Prediction and Analysis of Performance of a Centrifugal Compressor with Casing Treatment under Multiple Working Conditions. Applied Mechanics and Materials, 448–453, 3434–3439. https://doi.org/10.4028/www.scientific.net/amm.448-453.3434.

[15] 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.

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

[17] 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.

[18] 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).

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

[20] 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