International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161
Available at http://inpressco.com/category/ijcet

Research Article

Computational Study on Effect of Parameters on Stress of Centrifugal Compressor Blades

Gurdeep Singh Atwal†* and Sanjeev Kumar Dhama†

†Rayat and Bahra Institute of Engineering and Bio-Technology, Punjab Saharaun, India

Accepted 24 May 2015, Available online 27 May 2015, **Vol.5, No.3 (June 2015)**

Abstract

The main aim of this paper is to analyse the stress variation on the blades of centrifugal compressor by varying two parameters i.e. number of blades and inlet velocity. Air is used as working fluid and material used for impellers is structural steel. The only boundary condition is that the inlet temperature is constant i.e.300k.The meshing used is tetrahedral meshing. The 3-D model of the blades was created in PRO-e and then the file was imported to ANSYS for further analysis. By increasing the number of blades and inlet velocity, pressure increases therefore the stress also increases. ANSYS fluent fluid flow application is used to import the pressure as it is the part of CFD. With the help of ANSYS fluent fluid flow and static structural application of ANSYS, Results were obtained for velocity contours, pressure contours, total deformation and equivalent stress values. With the help of Two-way analysis of variance (ANOVA) method, the significant percentage obtained with respect to the input velocities and number of blades. The best significant result comes to be in the case of input velocities (68%) and the significant percentage for number of blades is 28%. The error percentage is also very low.

Keywords: *ANSYS, PRO-e, CFD, ANOVA, Compressor, Blade, Stress and Significant Percentage.*

1. Introduction

In today's era as the technology rests its feet in every new invention, production and discovery. After the inception of any new idea, human beings start preparing for the planning and procedure in which the new inventions has to be take place. For conducting experiments, designing the new products, initializing the new production lines, for designing complex figures, new software and management tools are required to handle the work behind new production line carefully, as in the similar case of centrifugal compressors. Before the invention of new designing and analysis software all the work has to be done manually, and it takes a lot of time. The work becomes chafe and boring. A caprice can be seen from the ancient times to medieval and from primitive to modern times. With God's grace and bliss many meaningful and exhaustive experiments had been conducted by number of scientists to study and evaluate the various kind of stresses, fluid flow phenomenon and laws governing that requisite and well defined phenomenon to obtain the maximum efficiency from the centrifugal compressors. Problems also start taking birth with every new invention. To abrogate such problems new discoveries has been made in the different fields of engineering by thinking about the various aspects of the new product design and their working. Initial works has been done on the intake system of the centrifugal compressor and especially in the impeller and vicinity around the hub and shroud. Modifications were made in the inlet guide vanes, inlet guide valves, by using fuzzy logics and control performance parameters of centrifugal compressor. Improvements has been made in the tandem blade designing in the field of turbocharger centrifugal compressor. Using different methods for the redesign of new blades of centrifugal compressor, study the dynamic stress and vibration resonance for further analysis in the field of centrifugal compressor. Centrifugal compressors plays a great role in fields of refrigeration and air conditioning, automobile industry and various type of turbines. The main aim of this paper is to get the optimum results by varying the number of blades and input velocity.

2. Literature Review

Phull Gurloveleen Singh *et al* (2010):-Centrifugal compressors are used in process industries today especially in gas service industries. Pro-E software and ANSYS as a tool can be used for simulation of the flow field characteristics inside the turbo machinery. Numerical simulation makes it possible to visualize the flow condition inside a centrifugal compressor, and provides the valuable hydraulic design information of

*Corresponding author: **Gurdeep Singh Atwal**

the centrifugal compressor. Present work is aimed to analyse the stress and velocity distribution inside the compressor passage using the Pro-E and ANSYS softwares. Computer program code in C++ has been developed to calculate the dimensions of the compressor components. A numerical model of an impeller has been generated and the complex internal flow fields and stress distribution are investigated by using the ANSYS-CFX computational code. The internal flow is not quite smooth in the suction and pressure side of the blade due to non-tangential inflow conditions which results in the flow separation at the leading edge. Stress distribution and deformation of the impeller depends with speed of the rotation. Pressure and velocity distribution inside impeller of the centrifugal compressor has direct influence due to change of stream wise location. Similar computational simulation models can also be used for analysing the pressure, velocity and stress distribution of the turbines, pump, fan and blower.

Atsushi HIgashio *et al* (2010):- Thesis work provide a unique stress measurement technique for rotating blades. To estimate the vibrational stress on the blades. Unsteady CFD analysis s applied for calculation of the external excitation force and the vibration response is analysed. The basic vibration characteristics of the nodal diameter interference diagram and Campbell diagram are explained.

T.CH.Siva Reddy *et al* (2010):- CFD studies have been carried out using ANSYS Tascflow software on the flow through return channel system of a typical centrifugal compressor stage for the purpose of finding out the losses associated in the return channel system. Study covered the flow passing through impeller, diffuser, u-bend and return channel passage. Parameters such as pressure loss coefficient, static pressure recovery coefficient flow angle etc. were used for the purpose of comparison of performance in the existing and modified configuration. Improvement in performance was noticed with the modification. Boundary conditions are- stagnation pressure and static pressure and static temperature corresponding to ambient conditions is imposed at entry o inlet duct, and design mass flow rate is imposed at return channel passage exit. The analysis was carried out to an impeller Mach number of 0.33 and loss coefficient reduced for the range of flow coefficients studied with the modifications carried out on the return channel vane.

Wen Guang Li *et al* (2011):- Work has been done in the inverse design of impeller blade in centrifugal compressor with singularity method. Cubic Bezier curves are used to establish the density function of bound vertex intensity along the blade camber line. Impellers blades are redesigned and three-dimensional viscous flows inside redesigned blades has been calculated with the help of CFD FLUENT software increases the hydraulic efficiency of centrifugal pump. The direct and inverse problems have been validated with a typical experimental centrifugal pump impeller. The impeller blades were redesigned by using the method and three dimensional turbulent viscous flows inside the original and redesigned impellers were calculated numerically by a means of CFD code FLUENT.A density distribution of bound vortex intensity on blade camber line was defined by using a cubic Bezier curve. The angle of attack has been involved in such a distribution. The defined density of bound vortex intensity can ensure the designed blade to have a carefully controlled loading coefficient and smooth camber line to guarantee an improved hydraulic efficiency.

M.Zangenesh *et al* (2012):- development of a high performance centrifugal compressor using a 3- d inverse design technique. The main features of the inverse design method and its design parameters are described. The structural and vibration characteristic of the new impeller are computed and compared with the conventional impeller by using 3D finite element analysis. The flow through the inverse designed and conventional impellers are analysed by using a 3D CFD code both for impeller only whole stage configurations. The results of the study confirm that by using an aft-loaded blade loading distribution on the shroud and careful control of the work ratio between the full and splitter blades the shock strength in inducer can be reduced. As a result significant improvement in measured stage performance of the compressor of up to 3 point in efficiency and 4.5% in pressure ratio can be obtained without any effective reduction in stable operating range. 3D finite element analysis was used to investigate the effect of the 3D shape on centrifugal stresses and vibration characteristics of the impeller.

D.P.S. Abam *et al* (2012):-This paper presents the optimization of centrifugal compressors. The purpose is to obtain optimal flow conditions in centrifugal compressors. The analysis is with respect to the effect of the degree of pre-whirl on the optimal inlet relative flow angle and hence the optimal mass flow rate achievable at various inlet relative Mach numbers for different values of the compressor inducer geometric parameter. A mass flow function incorporating all these parameters was developed and analysed in a MATLAB environment to obtain the optimal inlet relative flow angle and the corresponding optimal mass flow rate at a given rotational speed of the compressor. The optimal inlet relative flow angle increases with the inlet relative Mach number at the given degree of pre-whirl, but decreases with an increase in the degree of pre-whirl at the given inlet relative Mach number. Also the optimal mass flow rates for computed values of the geometric parameter fall between those for two extreme allowable values of this parameter. The optimal rotational speed at a defined mass flow rate could be obtained from the analysis, and the results of this work will serve as guide for both design and operation of centrifugal compressors. The decrease in the optimal inlet relative flow angle with increase in degree of pre-whirl at a given inlet relative Mach number does not favour the operation of the compressors at very high degree of pre-whirl due to turbulence in the flow and associated pressure losses. The one dimensional flow analysis that led to the realization of the mass flow function fails. Also, high inlet relative Mach number should not be targeted to obtain large air flow rates due to the inherent pressure losses and decrease in system efficiency. A parameter of interest is the compressor inducer geometric parameter K which relates the hub/tip radii radio of the inducer inlet. The results obtained, especially the optimal mass flow rates at the given conditions evaluated for assumed compressor inducer geometric parameter, K only depicts the maximum and the minimum values achievable for smooth operation of the compressor at the given conditions. The optimal mass flow rates obtained using the computed K values fall between the minimum and the maximum values even at larger values of pre-whirl degree where the K values are only approximations. From the results obtained, decrease in the K value with inlet relative Mach number for a given degree of pre-whirl means an increase in the velocity of the fluid at the inlet which should not be too high for normal operation. Although optimum mass flow rate at various conditions were evaluated here, it is possible to compute the optimum shaft speed N in rpm for a specified value of mass flow rate under optimum conditions. The results obtained in this work could serve as guide for both design and operation of centrifugal compressors.

Surendra Anish *et al* (2012): - A computational study has been conducted to analyse the performance of a centrifugal compressor under various levels of impeller-diffuser interactions. The study has been conducted using a low solidity vaned diffuser (LSVD), a conventional vaned diffuser (VD) and a vane less diffuser (VLD). The study is carried out using Reynolds-Averaged Navier - Stokes simulations. A commercial software ANSYS CFX is used for this purpose. The extent of diffuser influence on impeller flow is studied by keeping the diffuser vane leading edge at three different radial locations. Detailed flow analysis inside the impeller passage shows that the strength and location of the wake region at the exit of impeller blade is heavily depended upon the tip leakage flow and the pressure equalization flow. Above design flow rate, the diffuser vane affects only the last twenty percent of the impeller flow. However, below design flow rate, keeping vane closer to the impeller can cause an early stall within the impeller. Small negative incidence angle at the diffuser vane is helpful in order to reduce the losses at the impeller exit. A computational study has been conducted to analyse the effects of diffuser vane impeller exit flow. Three types of vane diffusers are used; a conventional vaned diffuser (VD), low solidity vaned diffuser (LSVD) and a partial vaned diffuser (PVD). The diffuser vane leading edge is kept at different radial locations to understand its influence on the impeller exit flow. Above design flow rate, the diffuser vane affects only the last twenty percent of the impeller flow. However below design flow rate, keeping vane closer to the impeller can cause an early stall within the impeller. Detailed flow analysis inside the impeller passage shows that the strength and location of the wake region at the exit of impeller blade is heavily depended upon the tip leakage flow and the pressure equalization flow. Even at design flow rate the flow angle near the shroud is too small and hence it causes large positive incidence. This enhances the reverse flow near shroud. A small negative incidence at the diffuser vane is desirable in order to reduce the losses at the impeller exit. Partial vane diffusers do not affect the impeller exit flow adversely instead they stabilize the impeller separation region at lower flow rates.

Yuanyangzhao *et al* (2014):- A method of compressor performance analysis under multiple working conditions is present based on the Time weighted average .Then the comprehensive analysis method can be used to get the overall performance of compressor. The performance of a basic centrifugal compressor was simulation by CFD method in this paper. The overall performance of the centrifugal compressor is calculated under different working conditions. The TWA analysis method can be used as a tool to evaluate the overall performance of compressor. And it can also be used during the design to improve the performance of compressor fundamentally. It is difficult to evaluate the performance of compressor working under multiple conditions reasonably. To solve this problem, the time-weighted average (TWA) method is proposed to get the overall performance of compressor. A centrifugal compressor is researched in this paper. The results show that the working condition and length of working time have a large effect on the performance of compressor. The different value is 13.3% for four cases. The method of multiple working conditions can present the real situation of compressor scientifically. This method can be used to evaluate the overall performance of compressor. And it also can be used to optimization design of compressor.

3. Design Procedure for impeller blade

Procedure for designing an impeller blade is as follows:

1. Revolve
2. Line- draw two lines one horizontal and one vertical.
3. Draw or drop the spline with respect to the above two lines to make it a curve.
4. Now blend the surface.
5. Provide the arbitrary thickness to the surface.
6. Draw the number of blades with the polar array.

Number of blades used:

- 6 blades
- 8 blades
- 12 blades

Varying Input Velocities used:

- -9 m/s
- -12 m/s
- -15 m/s

An initial 3D model with 6 blades, 8 blades and 12 blades was created, and .igs file was also created with respect to the each blade configuration. This .igs file was imported to analysis geometry. From geometry, .igs file was further imported to fluent fluid flow where meshing was done with fine mesh all over the figure. To make the mesh clearer one can use the relevance factor. Further more stress values are transported to the ANOVA table in which both the input parameter comes out to be significant.

Table 1: Properties of Structural steel

Sr. No.	Properties	Values
1.	Young's modulus of elasticity	29,000 psi.(aprox.)
2.	Poisson's ratio	0.260
3.	Density of structural steel	7,800 kg/ m^3

Software's Used:

- Designing Software PRO-e
- ANSYS (Fluid Flow Fluent)
- ANSYS (Static Structural)

1. ANSYS-(Fluid Flow Fluent)

Procedure for ANSYS is as follows:

- Import .igs file of geometry
- Mesh
- Setup
- Solution
- Results

2. ANSYS- Static Structural

Procedure for static structural is as follows

- Engineering data
- Mesh
- Setup
- Solution
- Results

4. Results and Discussions

From the Following results study one can say that input velocity has the more significance over the centrifugal compressor. This implies that the input velocity factor has more influence on impeller as compared to the number of blades. The pressure obtained on the output side is higher than the pressure on inlet side and this is also the main function of the centrifugal compressor. Therefore centrifugal compressors are best suitable for small scale industry.

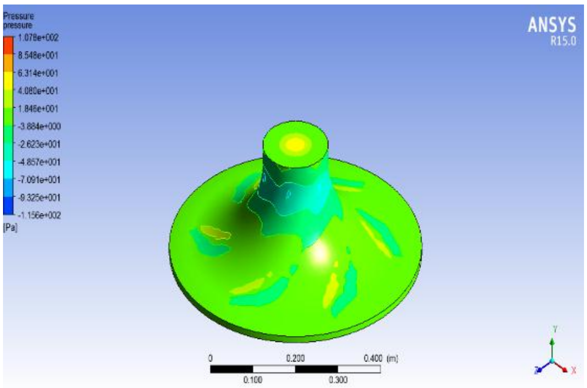

Fig.1: Pressure Contour At -9 m/s For 6 blades

Table 2: ANOVA Table

Source of variation	(SS)	dof	(M.S)	F.r	5 % F-limit (table values)
b/w number of blades	1165398	2	582699	14.113	F(2,4)= 6.94
b/w input velocities	2878962.66	2	1439481.33	34.864	F(2,4)= 6.94
error	165149.34	4	41287.33		
total	4209510	8			

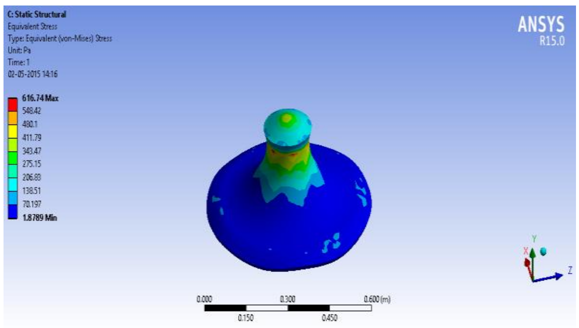

Fig. 2: Von- Mises Stress at -9m/s for 6 Blades

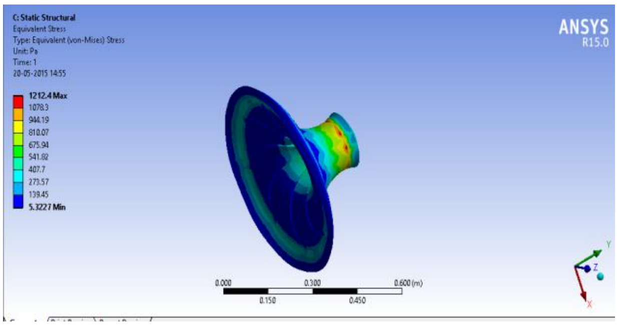

Fig.3: Equivalent stress at -12m/s for 8 blades

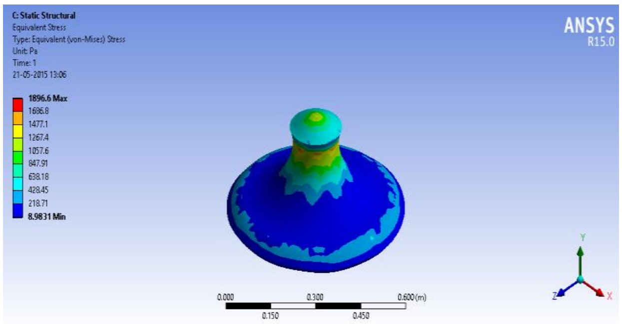

Fig 4: Von – Mises Stress at -15 m/s for 8blades

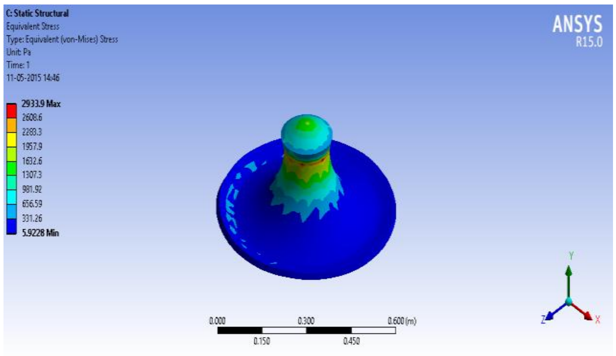

Fig.5: Equivalent Stress at -15 m/s For 12 blades

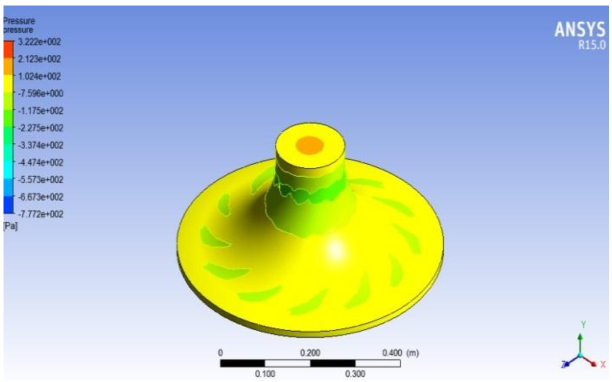

Fig.6: Pressure Contour at -15 m/s for 12 Blades

Conclusions

In this study, the effect of inlet velocity and number of blades has been studied on the centrifugal compressor when operated within safe limit. The stress value increases as the number of blades and inlet velocity increases. The significant results are calculated by ANOVA method for both the input parameters. The inlet velocity comes out to be the most significant for the different number of blades. The ANSYS software is used with its two applications i.e. fluent fluid flow and static structural- to calculate the outlet velocity, Von-Mises stress distribution, total deformation values and also to get the images for pressure and velocity contour. The best significant result comes to be in the case of input velocities (68%) and the significant percentage for number of blades is 28%, with low error percentage.

Future scope of the work

Even though lot of research has been carried out in the field of centrifugal compressor and its various components. But in future different modifications can be carried out in this particular field:

- The designing can be done by using the other inlet parameters such as blade, blade width etcetera and other mathematical techniques
- Other parameters can be used such as different sensing tools can also be attached to the impeller to know about the vibration levels and other factors.

References

V.C.Arunachalam *et al*(2008), Numerical studies of the effect of impeller blade screw on centrifugal compressor performance, Proceedings of the 4th BSME- ASME International Conference on Thermal Engineering", 27-29 December, 2008, Dhaka, Bangladesh.

Phull Gurloveleen Singh *et al* (2010), Modelling and stress simulation of centrifugal compressor, International Journal Of advances in thermal sciences and engineering, 1(1) June 2010, pp.99-10

Atsushi Higashio *et al* (2010), Dynamic stress measurement of centrifugal compressor impeller and study for strength criteria based on correlation by unsteady CFD, 39th Turbomachinery Symposium (2010) pg.43. http:// turbolab. tamu.edu/proc/ turboproc/T39/

Wen- Guang Li (2011) ,Inverse design of impeller blade of centrifugal pump with a singularity method, Jordan Journal Of mechanical and industrial engineering Volume5, Number2, April 2011 ISSN 1995-6665 Pages 119-128.

Surendran Anish *et al* (2012), Study of secondary flow modifications at impeller exit of a centrifugal compressor, Open journal of fluid dynamics.2012,2, 248- 256 http://dx.doi.org/ 10.4236/ ojfd.2012.24A029

M. Zangenesh (2012), Development of a high performance centrifugal compressor using 3D inverse design technique, on secondment from Ebara corporation japan, fluid machinery and systems company 11-1 haneda asahi-cho, ohta-ku, Tokyo 144-8510, japan.

D.P.S Abam and E.G. Saturday (2012), Optimization of centrifugal compressor: Acase study, Journal of emerging trends in engineering and applied sciences(JETEAS) 3 (4): 688-694© scholarlink research institute journals, 2012 (ISSN: 21421-7016) Jeteas.scholarlinkresearch.org

Surendran Anish *et al* (2012),Study of secondary flow modifications at impeller exit of a centrifugal compressor, Open journal of fluid dynamics".2012,2, 248- 256http:// dx.doi.org/ 10.4236/ojfd. 2012.24A029

H.P.Dickmann(2013), Shroud contour optimization for a turbocharger centrifugal compressor trim family, ABB Turbo systems ltd, R&D turbochargers/ fluid mechanics, 5401 Baden, Switzerland.

Alok.B.Timbrewala *et al* (2014). Flow analysis of upstream fluid flow using simulation for different positions of optimized inlet guide vane in centrifugal air compressor , American journal of engineering research, e-ISSN: 2320-0847p-ISSN: 2320-0936 Volume03, Issue-02, pp148-156 www.ajer.org