EXPERIMENTAL AND NUMERICAL ANALYSIS ON CENTRIFUGAL BLOWER

Anchal Paroha[a], Dr. M.K. Chopra[b], Vivek Singh[c]

[a]M.Tech (Thermal), Research Scholar RKDFIST Bhopal
[b] Vice Principal Dean Academic & HOD ME, RKDFIST Bhopal
[c]Assistant Professor (Mechanical Dept.) RKDFIST Bhopal

A B S T R A C T

In turbo machines, the transfer of mechanical energy occurs into or out of machine in a steady flow process. The present paper consists of experimental as well as analytical based approach for centrifugal blower. The study is carried out and results for velocity distribution at the outlet and stagnation pressure value is calculated. Further the results are optimized using Taguchi L9 orthogonal array. Contour plots have been shown in the results and the values obtained are compared for both the cases of analytical as well as experimental approach.
Keywords:
Turbo machines, velocity, pressure, contours

1. Introduction

Turbo machines comprises that types of machines which help in making heat and force like centrifugal pumps and compressors are those which create energy such as turbines. The liquid is not definitely limited but rapidly drifts with the help of machine that are in undergoing changes in force by sources of energetic things. Thus, the alter name we can give to turbo machine is dynamic machines.

The two main groups of turbo machines are bounce. Initially, are those that grip the energy to develop the liquid force or heat (fans, blowers and compressors); secondly are those that generate energy by increasing liquid to a lesser force or heat (turbines).

Turbo machines are too divided as open and closed turbo machines. Open machines which acts on an endless extent of liquid like propellers, windmills, and unshrouded fans while closed machines functions on a limited standard of liquid as it comes from housing or casing. Turbo machines are also divided on the grounds of the types of movements. When the movement is similar to the axis of spin of the machine then this type of motor is also termed as axial movement motor and when the movement is vertical to the axis of spin of the motor then this type of motor is called as radial movement motors. When liquid comes axially and leaves outward then this type of motor is known as mixed flow machine

Centrifugal blower

The used of centrifugal blowers is for manufacturing procedure and for regulating the air contamination system these are too common in central heating and cooling systems. It is used for moving the air endlessely

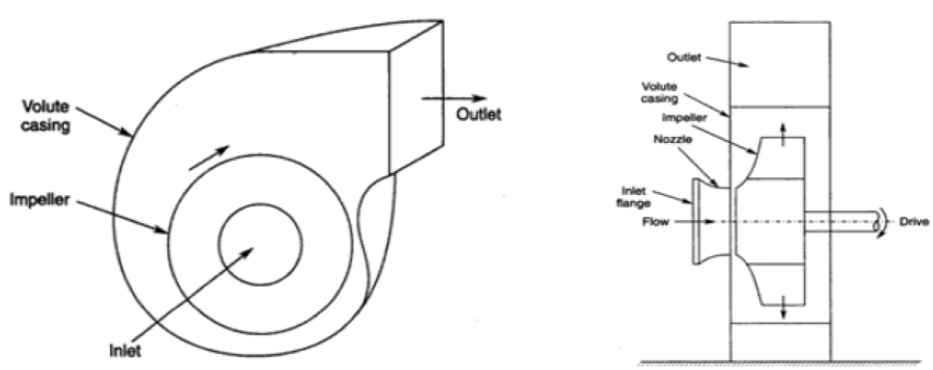


Fig.1 Centrifugal Blower

Computational Fluid Dynamics (CFD)

Different flow marvel's happening inside turbo machine can be numerically broke down with the assistance of financially accessible CFD programming. Presently multi day's three dimensional flow computations are doable, enabling architects to have a superior gauge of the impact of spatial flow non-consistency on execution of machine. Any reasonable flow reenactment must be done on a three dimensional premise. Because of the bended flow entries inside the impeller and volute, the flow must be considered as three dimensional. Computational Fluid Dynamics is presently a set up modern plan instrument, diminishing structure time scales and improves forms all through the designing scene. CFD gives the savvy and exact choice to scale model testing, with minor departure from the reenactments being performed rapidly, offering clear favorable position. The arrangement of conditions which depicts the procedures of energy, warmth and mass exchange are known as Navier-Stokes conditions. These halfway differential conditions were determined in the mid nineteenth century and have no realized general logical arrangement however can be discretized and understood numerically.

Taguchi method

The hypothetical establishments of Taguchi Methods were spread out by Genichi Taguchi, a Japanese designer in 1950. This is the technique for upgrading the control parameters for the exhibition and the expense. Taguchi's way to deal with parameter configuration gives the structure engineer a methodical and proficient strategy for deciding close ideal plan parameters for execution and cost. The goal is to choose the best mix of control parameters so the item or procedure is most strong regarding clamor factors. The Taguchi strategy uses symmetrical exhibits from structure of examinations hypothesis to think about an enormous number of factors with few tests. Utilizing symmetrical exhibits essentially decreases the quantity of test setups to be contemplated. Moreover, the ends drawn from little scale examinations are legitimate over the whole test area crossed by the control factors and their settings. They were found extensively before. Be that as it may, Taguchi has streamlined their utilization by giving organized arrangements of standard symmetrical exhibits and comparing direct diagrams to fit explicit undertakings.

The initial phase in the Taguchi strategy is to decide the quality trademark to be improved. It is the yield or the reaction variable to be watched. The subsequent stage is to recognize the clamor factors that can negatively affect framework execution and quality. Clamor components are those parameters which are either wild or are too costly to even consider controlling. The third step is to recognize the control parameters thought to effectsly affect the quality trademark. Control (test) parameters are those structure factors that can be set and kept up. The following stage is to structure the lattice test and characterize the information examination method. In this progression, a symmetrical cluster is to be chosen. The subsequent stage is to lead the network test and record the outcomes.

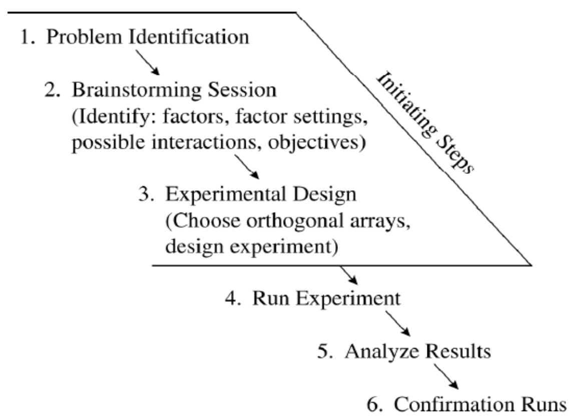


Fig: General overview of Taguchi method

After the experiments have been conducted, the optimal test parameter configuration within the experiment design must be determined. To analyse the results, the Taguchi method uses a statistical measure of performance called signal to noise (S/N) ratio. There are three modes of S/N ratio to decide optimal parameters and these are as follows:

1. Higher is best
2. Nominal is best
3. Lower is best.

Depending upon the application, any of these three modes has been selected for optimization purpose

2. LITERATURE SURVEY

Songling Wang et. al. in his work did the numerical investigation and test deal with the radiating fan. The motivation behind his work was to improve the proficiency of the fan by bringing down the vitality misfortune which is brought about by the auxiliary stream vortex, the volute tongue, the wake-fly and the approach. He did the streamlining utilizing least square strategy. He reasons that with the advanced impeller the force of vortex arrangement is decreased. Subsequent to enhancing the impeller, the all out weight and effectiveness expanded 3.7% and 0.5% individually.

Zhang Bin et. al. the motivation behind the examination was to improve the exhibition of blower through the advancement plan of the cutting edge's profile appropriately. In his work a modern ultra-low explicit speed outward blower with parallel center point and cover had been chosen as a source of perspective case for improvement plan. The presentation examinations of the divergent blowers with various kinds of cutting edges were led. In view of artificial neural networks (ANN) and various leveled reasonable challenge hereditary calculations with dynamic specialty (HFCDN-GAs), the improvement configuration approach was set up.

Straight edge, backswept cutting edge, forward bended sharp edge (enhanced CTB) and streamlined VTB edges were tried for plan and advancement reason. The streamlined exhibitions of the blower sharp edges were acquired by the arrangement of Reynolds-arrived at the midpoint of Navier-Stokes (RANS) conditions. An accessible CFD code, FINE/TURBO, was utilized for this work. The computational area comprised of the impeller and the volute. The three-dimensional progression of the blower was recreated by utilizing the organized work and second request cell-vertex limited volume discretization strategy

Andrea Toffolo et. al. cross-stream fan execution was unequivocally impacted by the geometry of the packaging, as the last mentioned, thus, influences the position and the quality of the unconventional vortex that describes the activity of this class of fans. The work incorporates an efficient test examination of the stream field inside the impeller at various throttling conditions and for various geometries of the fan packaging. The two weights and speeds were estimated utilizing a three-dimensional five-gap test that was embedded in the stream. The investigation decides the connection between the plan parameters of the packaging and the stream field design. The reenactment was performed with the business CFD code FLUENT, utilizing the RNG k–ε disturbance model and a lattice with around 100,000 cells.

The finishing up comments of his work were stream field designs in which the erraticism and the quality of the vortex were restricted were gotten at low stream rates utilizing little outspread widths of the back divider (RE) and thick vortex dividers. Under these conditions the all out weight coefficient was low, however the all out productivity was higher whenever contrasted with slender vortex divider arrangements. Besides, Maximum proficiency was accomplished when the vortex had the most extreme flightiness (for example it lies on internal fringe of the impeller) yet a Moderate quality, that implies when a thick vortex divider was utilized in mix with a little outspread width back divider at medium stream rates (around 0.6).

Qubo Li. et. al. through methods for 3-D CFD (Computational Fluid Dynamics) technique, a novel pivotal blower with various impeller shapes compacting water vapor as refrigerant was researched. The numerical reproduction centers around the liquid stream from blower impeller channel to outlet. The general execution level and range were anticipated. Diverse sharp edge designs with various center point sizes were analyzed in regards to the streamlined presentation. Enduring state CFD recreations had been led so as to think about the attributes of various hub impeller designs with various center point sizes. The 3-D work of the stream diverts was fit naturally in GAMBIT in Tetrahedral/Hybrid components. A framework reliance study was likewise completed by locally refining the networks of the entire impeller. For choppiness demonstrating, the standard k-ε model without feasible changes was utilized for its strength and lesser interest on figuring assets contrasted with the other disturbance models accessible in FLUENT. The second-request precise upwind differencing for the convection terms of each administering condition was utilized to limit the cross-stream numerical dispersion. An untypical geometry of a pivotal blower was broke down to acquire its streamlined attributes. Considering the application for pressure of the water vapors in refrigeration establishment, quite certain medium highlighted must be considered. The figurings were performed on 3-D spaces that duplicate the genuine geometry of the blower to get a virtual exhibition map at consistent tip speed of 220 m/s. Comparing to various examples and center point sizes, six cases were contemplated and the investigation of the stream qualities was additionally performed to acquire a superior comprehension of these novel pivotal blower practices.

Finish of this work depended on the correlation, it was discovered that weight proportion and proficiency increment with center size for the examined examples, while a more extensive stable working reach was found for the littler center for a similar cover width in each of the six cases.

K. Senthilkumar et. al. utilizes Taguchi technique to improve the framework parameters of radial evaporative air cooler. Framework parameters picked are geometrical and operational parameters. In the examination, it was discovered that circle speed, wind current rate and water stream rate are having real impact on the presentation and plate distance across, stick geometry and chamber length have less impact.

K. Vasudeva Karanth and N. Yagnesh Sharma examine the stream between impeller exit and diffuser section for example spiral hole among impeller and diffuser and because of this outspread hole impact on the presentation of divergent fan would investigation with the assistance of CFD device. As a noteworthy derivation from the above investigation, it is discovered that there is an ideal spiral hole at which better unique and static heads are created by the impeller edges just as better vitality change by diffuser vanes. The static weight recuperation and absolute weight misfortune for the diffusing segments of the fan change with the outspread hole. The stream and wake marvels as found in all the impeller entries are impacted by the spiral hole among impeller and diffuser.

N. Yagnesh Sharma and K. Vasudeva Karanth inspected the impact of splitter vanes comparing to different geometrical areas on the impeller and diffuser. When all is said in done, splitter vanes gave on impeller and diffuser at sensibly picked areas will in general improve the exhibition of the radial fan, as far as higher static weight recuperation coefficients and diminished absolute weight misfortune coefficients.

A three-dimensional, relentless, incompressible, violent stream field inside a multi-cutting edge diffusive fan has been broke down numerically by Jiabing Wang et. al.[18]. Reynolds-found the middle value of Navier-Stokes conditions with the standard k-ε disturbance model have been discretized by the limited volume strategy. The estimation results have appeared three-dimensional qualities of the stream, particularly in the sharp edge entries close to the cover side.

The outcomes have uncovered a limit layer detachment at the main edge on the sharp edge suction surface, the stream inversion from the high weight district inside the volute to the low weight area close to the impeller delta, the stream distribution close to the cover side, a fly wake design at the rotor leave, the weight variance on the cutting edge surface and so on.

3. Methodology

Every one of these analyses were directed at Vacc Blow Engineering Pvt. Ltd. Nasik. The analysis is done on single stage outward blower. The instruments utilized for examinations incorporates five opening test, test navigating system and U-tube manometers for estimating the weight.

Table 3.1 Specification of impeller and volute casing

Specification of impeller		Specification of volute casing	
Parameter	Dimension	Parameter	Dimension
Inlet Diameter (cm)	30	Volute base circle diameter (cm)	46.75
Outlet Diameter (cm)	42.5	Volute exit diameter (cm)	125.26
Inlet blade angle (deg)	20	Volute width (cm)	26
Outlet blade angle (deg)	48	Throat length (cm)	39.26
Thickness of blade (cm)	0.5	Tongue angle (deg)	21.84
Number of blades	12	Tongue radius (cm)	25.51

To investigate the weight, tapping were made at azimuth edge from 00 to 3600 at an interim of 300 as appeared in figure 3.5. The test is embedded through each situation to examine the stream properties at all pivotal area. With the assistance of test adjustment steady, the estimations of static weight, stagnation weight and speed can be discover at all precise positions at impeller outlet.

3.1 Measurement of flow parameters to calculate static pressure, stagnation pressure and velocity

a) Measurement of stagnation and static pressure

Stagnation pressure is the pressure of the liquid which is accomplished by decelerating the liquid to zero speed at zero rise. To gauge the stagnation pressure of liquid, an impediment ought to be set into stream. The bearing of the obstruction ought to be symmetrically typical to the stream. Here, the test is set in the progression of the liquid. The middle gap of the test gives the stagnation pressure. All readings are taken when the pressure morally justified and left opening of the test is equivalent. The test is turned in the streaming liquid with the end goal that the pressure in the left and right opening of the test winds up equivalent. This condition guarantees that the bearing of the test is typical to liquid stream. Static pressure is the normal pressure of all other four gaps of the test.

b) Measurement of flow direction and velocity

The stream heading is imagined with the assistance of the gadgets, for example, tufts, streamers and vanes. Be that as it may, it produce aggravation in the stream. Pressure tests can demonstrate the stream course without stream unsettling influence. The stream bearing can be found either by adjusting the body to discover the variety of pressure contrast coefficient with yaw edge or by turning the body until the pressure distinction is zero.

The calibration constant used for flow direction measurements are given by,

$$F_1 = \frac{P_D - P_U}{P_O - P_S}$$

Where,

F_1 is pitch angle calibration constant

P_D and P_U are the pressure measured by lower and upper tube of the probe

P_0 is pressure measured by centre tube or stagnation pressure and

P_S is static pressure

And other constant used for measurement of the flow rate is equals to 1.7794 and is given by equation,

$$F_2 = C_P \sqrt{\frac{\Delta P}{P_O - P_S}}$$

Where,

F_2 is velocity calibration constant

C_P is calibration Pitot tube constant

ΔP is velocity pressure from Pitot tube.

After calculating the pitch angle and flow rate of the fluid, the velocity of the flow is given as

$$V = \sqrt{\frac{2Q}{\rho}}$$

Where,

Q is flow rate

ρ is density of the fluid

3.2 Analytical Approach

Numerical examination comprises of displaying the total liquid area in demonstrating programming and taking care of a similar liquid issue in solver. ANSYS inc. offers Gambit as pre-processor. Assignment of hubs according to structure geometry is done by utilizing Gambit programming.

Inlet suction pipe of blower is of 30 cm distance across and 50 cm of length. At the impeller side pipe is made little of 28 cm and length of that segment is 5 cm.

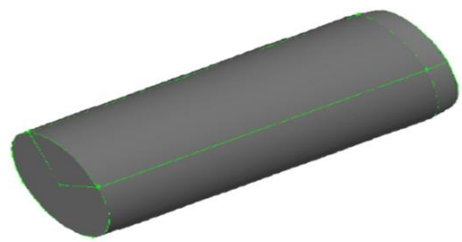


Figure 3.7 Domain of inlet duct

The co-ordinates of profile points with respect to global R-θ co-ordinate system of Gambit software are obtained. After generating the points, the profile is created by connecting all points by arcs as shown in figure 3.9. After that, the 2D profile is extended in Z- direction to develop 3D blade of thickness 5 mm. Then the single blade volume is rotated at an interval of 300 to developed 12 equally spaced blades.

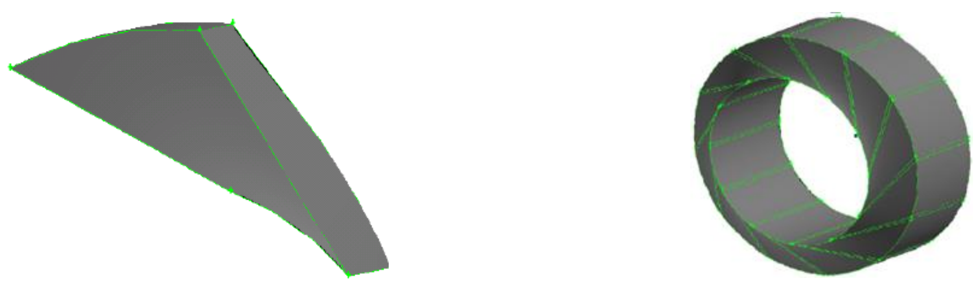


Figure 3: a) 3D Volume of domain of blade b)domain of the impellor

Volute casing

Volute casing of blower is developed as described follow; first upon the radius of peripheral of blower at different angles from 0^0 to 360^0 is calculated by the formula given below:

$$R_\theta = 39.2589\left(\frac{\theta}{360}\right) + 23.375$$

The tongue angle is 21.840 and at the radius of 25.51 cm. After creating the one face of volute it is extend in z-direction through 26 cm and develop the volume of volute casing. The complete assembly of different parts of centrifugal blower is shown:

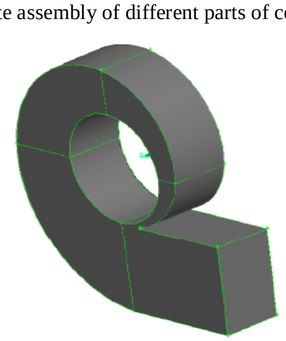


Figure 4: Domain of volute casing

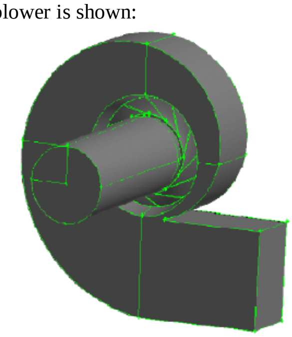


Figure 5: Assembly of blower

Optimization

Taguchi method is used for optimization

Table 3.1: Control factors and their levels for impeller

Factor/Parameter	Test levels		
	Level 1	Level 2	Level 3
Width at inlet (cm)	17	15	13
No. of blades	8	10	12

For the above case, array is L9 orthogonal array. This array is as shown in table 3.8 and is generated by Minitab software

Table 3.2: Orthogonal array

Experiment No.	Width at inlet (A)	Width at outlet	No. of blades (B)
1	17	13	8
2	17	13	10
3	17	13	12
4	15	13	8
5	15	13	10
6	15	13	12
7	13	13	8
8	13	13	10
9	13	13	12

4. RESULTS & DISCUSSIONS

The result of S/N ratio analysis is carried out furthermore and response table is generated. Response is shown in table 4.1 and parameters of impeller are optimized using response table

Table 4.1 Response Table

levels	Minimum variation ΔP			Maximum ΔP at outlet			Minimum ΔP_0		
	A	B	C	A	B	C	A	B	C
1	**-32.81**	-35.76	**-33.24**	61.28	62.46	60.02	-41.65	**-34.77**	-54.80
2	-35.88	**-34.74**	-36.33	63.05	**63.13**	63.69	**-39.24**	-49.84	**-32.97**
3	-37.31	-35.49	-36.43	**63.27**	62.02	**63.89**	-45.29	-41.57	-38.41

The stagnation pressure increases from impeller inlet to outlet. The maximum variation in stagnation pressure at impeller outlet from suction to exit is 13.6% in case of experiments while that for numerical is 27.4%.

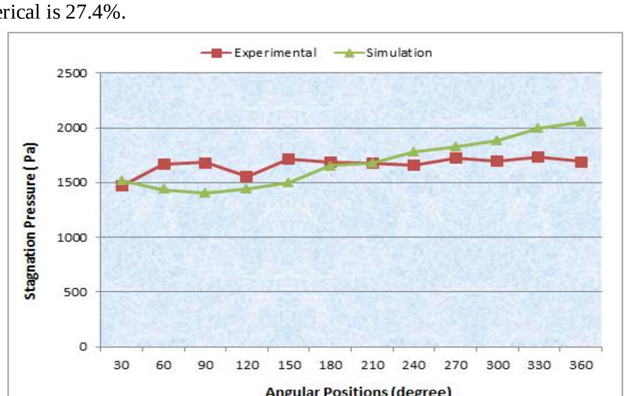


Fig 4.1: Comparison of numerical results with experimental results for stagnation pressure at the exit of impeller

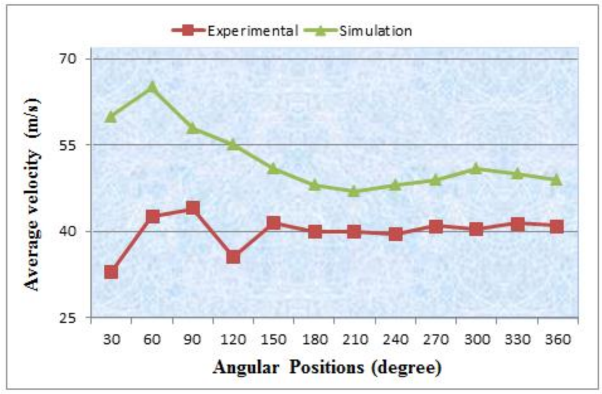
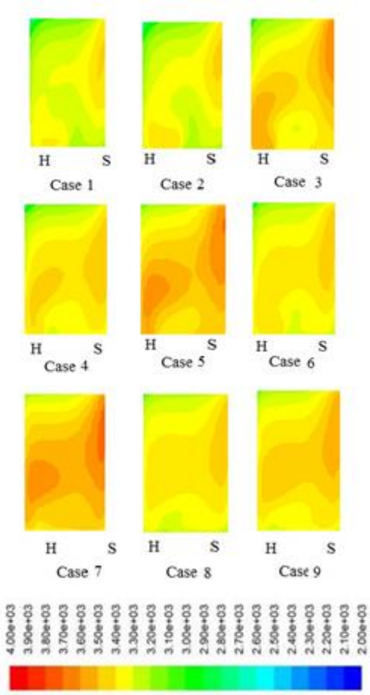


Fig 4.2: Comparison of numerical results with experimental results for velocity at the exit of impeller

Ansys numerical analysis plot results

The values of stagnation pressure and velocity at the outlet of the fluid domain are as shown in table 4.7. It is observed from table that a case 5 i.e. width of volute 22 cm, tongue angle 240 and 10% decrease in radial location gives maximum stagnation pressure and minimum loss in stagnation pressure



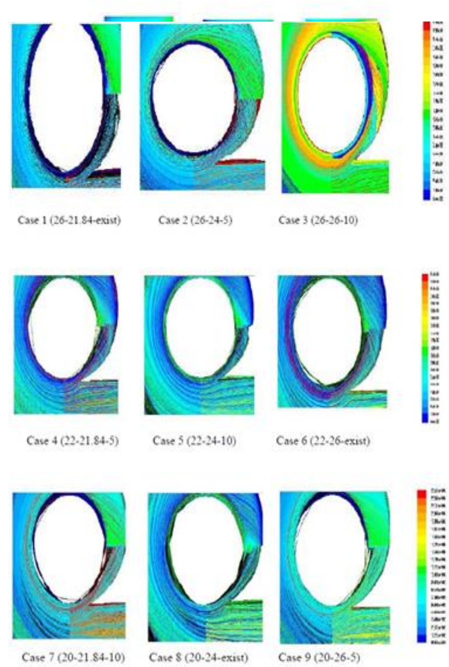


Fig4.3 Stagnation contours Fig 4.5 Velocity contours Fig 4.6: Velocity contours

6. Conclusion

1. The test examination uncovers that stagnation pressure continues expanding from suction to exit at impeller outlet. Static pressure is at first less close suction district and increments towards the exit of the impeller [20].

2. The normal stagnation pressure at impeller outlet from suction to exit is 1662.85 Pa for trial case though 1681.56 Pa for numerical case. Most extreme % deviation found in the two cases is 16.67%. The numerical outcome demonstrates great concurrence with exploratory one.

3. The Taguchi strategy proposes that the impeller setup 15-13-10 is upgraded case while 22-240-10% is streamlined case for volute design for present work.

4. The numerical examination for impeller setup proposes that the most extreme pressure recuperation inside the volute is watched 52.7% with design 15-13-10. Likewise, most extreme stagnation pressure is seen at outlet of liquid space with the case 15-13-10.

5. By numerical examination and streamlining system, 15-13-10 is enhanced instance of impeller design for the present divergent blower.

6. Numerical examination of volute setup uncovers that greatest static and stagnation pressure saw with case 22-240-10% at impeller outlet. In any case, static pressure varieties at impeller outlet are smidgen higher than the base esteem. Likewise, most extreme stagnation pressure saw at volute outlet for this case among all. There is least loss of stagnation pressure for example 0.1% saw with same case. The vortex development is likewise least with this case contrast with others.

7. The numerical examination and advancement method recommends that 22-240-10% is improved case for volute arrangements.

8. Reduction in width of volute just as decrease in % of volute outlet range influences the vortex development which prompts misfortunes inside the volute. Be that as it may, tongue edge does not assume much job in vortex arrangement.

With reference to this thesis work, we exceedingly prescribe case 5 (15-13-10) of outward blower with impeller parameters keeping volute parameters kept steady and case 5 (22-240-10%) 0f radial blower with volute parameters keeping impeller parameters consistent in regards to enhancement.

Case 5(15-13-10) for example width at inlet of impeller (15)- width at outlet of impeller (13)- number of cutting edges (10) is reasonable for higher execution of divergent blower of present set up, with respect to above case higher stagnation pressure saw at outlet of area which demonstrate less misfortunes contrasted with different setups.

Case 5 (22-240-10%) for example width of volute (22)- tongue edge (24)- rate decline volute outlet span (10%) is reasonable higher execution of radiating blower of present set up, concerning above case greatest static and stagnation pressure saw at outlet of impeller just as outlet of liquid area with less development of vortex contrasted with different arrangements

References

7. Songling WANG, Lei ZHANG, Zhengren WU Hongwei QIAN, Optimization Research of Centrifugal Fan with Different Blade Number and Outlet Blade Angle, 978-1-4244-2487-0/09/$25.00 c2009 IEEE.
8. Zhang Bin, Wang Tong, GU Chuan Gang, ShuXin Wei, Blade optimization design and performance investigations of an ultra-low specific speed centrifugal blower, Science China, January 2011 Vol. 54 No.1: 203–210
9. R. Barrio, J. Fernandez, E. Blanco, J. Parrondo, Estimation of radial load in centrifugal pumps using computational fluid dynamics, European Journal of Mechanics B/Fluids 30 (2011) 316–324.
10. Talib Z. Farge and Mark W. Johnson, Effect of flow rate on loss mechanisms in a backswept centrifugal impeller, Int. J. Heat and Fluid Flow, Vol. 13, No. 2, June 1992.
11. Tahsin Engin, MesutGur, Reinhard Scholz, Effects of tip clearance and impeller geometry on the performance of semi-open ceramic centrifugal fan impellers at elevated temperatures, Experimental Thermal and Fluid Science 30 (2006) 565–577.
12. Andrea Toffolo, Andrea Lazzaretto, Antonio Dario Martegani, An experimental investigation of the flow field pattern within the impeller of a cross-flow fan, Experimental Thermal and Fluid Science 29 (2004) 53–64.
13. Sheam-Chyun Lin, Chia-Lieh Huang, An integrated experimental and numerical study of forward–curved centrifugal fan, Experimental Thermal and Fluid Science 26 (2002) 421–434.
14. Qubo Li, JanuszPiechna, Norbert Muller , Numerical simulation of novel axial impeller patterns to compress water vapor as refrigerant, Energy 36 (2011) 2773 - 2781.
15. K.Senthilkumar, PSS.Srinivasan, Application of Taguchi Method for the Optimization of System Parameters of Centrifugal Evaporative Air Cooler, Journal of Thermal Science Vol.19, No.5 (2010) 473−479.
16. K.Vasudeva Karanth, N. Yagnesh Sharma, CFD Analysis of a Centrifugal Fan for Performance Enhancement using Converging Boundary Layer Suction Slots, World Academy of Science, Engineering and Technology 60 2009.
17. N.Yagnesh Sharma, K.Vasudeva Karanth, Numerical Analysis of a Centrifugal Fan for Improved Performance using Splitter Vanes, World Academy of Science, Engineering and Technology 60 2009