International Journal of Electronics, Communication & Soft Computing Science and Engineering
ISSN: 2277-9477, Special Issue of" International Conference on Recent Trends in Engineering & Technology "Apr-2012

NUMERICAL SIMULATION OF CENTRIFUGAL BLOWER USING CFX

Yogesh R. Pathak **Beena D. Baloni** **Dr.S.A.Channiwala**

***Abstract*— Numerical analysis of the single stage centrifugal blower is carried out for different flow coefficient. To analyze the 3-D flow field, fluid domain is created and simulation is done with the CFD software code ANSYS CFX. Three dimensional Navier-Stokes equations are used to analyze the flow. Standard *k-ε* turbulence model and unstructured grid is adapted to solve the Navier-Stokes equations.**

Results of numerical analysis suggest that the pressure coefficient at an outlet of fluid domain is continuously decreased with increase in flow coefficient. The detail flow analysis is also carried out for the practical case of flow coefficient. The flow is analyzed at different angular and axial positions inside the volute.

Keywords: Single stage centrifugal blower, numerical analysis, CFD, CFX.

I. INTRODUCTION

There are two basic types of air moving devices: the centrifugal fan or blower, and the axial flow fan. The centrifugal fan creates higher pressures than that of the axial flow fan and is applied on jobs with higher resistance to the air flow. Centrifugal Blower consists of an impeller with small blades on the circumference, a shroud to direct and control the air flow into the center of the impeller and out at the periphery. The blades move the air by centrifugal force and throwing it out, thus creating suction inside the impeller and suction duct.. The pressure rise and flow rate in centrifugal blowers depend on the peripheral speed of impeller and blade angles [6]. The stage losses and performance also vary with the blade geometry. To reduce the variables, all variables related to turbo machines are identified and grouped into non dimensional quantities. Thus, the dimensionless parameters are varied instead of large number of variables [6] in design and performance test.

Choon man jang [2] had done the performance characteristics of the turbo blowers having an inlet vane are analyzed by experimental measurements and numerical simulation with three-dimensional Navier-Stokes solver. Detailed flow analysis inside a turbo blower, an inlet vane and a connecting piping system is also performed using the results of numerical simulation. The objective of the present paper was to carry out 3-D simulation study of a single stage centrifugal blower with the help of CFD software ANSYS CFX. The simulation was carried out at different flow coefficient, and effect of that is analyzed on the performance of the blower. Also the detail analysis is carried out by analyzing the different flow properties at various angular positions inside the volute for experimental value of flow coefficient. Comparison is done based on simulation data. Flow properties at outlet of fluid domain are selected for comparison.

II. MODELING AND NUMERICAL SIMULATION

In present work, the geometry is constructed in ANSYS CFX workbench [8]. The geometry of the impeller blade is created by using point by point method [7]. Impeller, inlet duct and volute casing are created and assembled. After creating the model, meshing is also done in CFX itself. To analyze the flow through numerical simulation, the single stage blower is used which is run by a motor having 7.5 HP (5.5 KW) capacity and rotational speed of 2900 rpm in clockwise direction of impeller. The backward straight blade type tapered impeller with shroud and volute with "constant mean velocity" design concept is used in the blower design. The details of an impeller and the volute specifications are provided in table 1.

Table 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	20^0	Volute width (cm)	26
Outlet blade angle	48^0	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

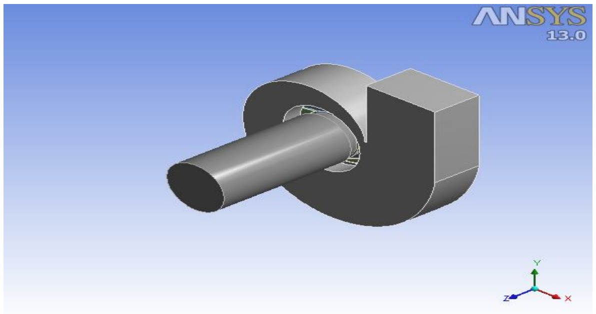


Fig.1 Model of centrifugal blower

The geometrical model of the centrifugal blower is created in ANSYS CFX as shown in figure 1.

III. Meshing of the Fluid Domain

Whole domain of centrifugal blower is meshed using the unstructured type of grid. In unstructured grids, typically utilize triangles in 2D and tetrahedral in 3D. The advantage of unstructured grid methods is that they are much automated and, therefore, require little user time or effort [2]. Table 3 shows the details of meshing of complete domain. The meshing of the fluid domain is shown in figure 2.

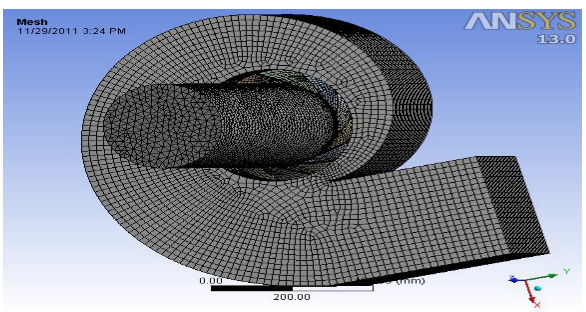


Fig. 2 Meshing of fluid domain

Table 3 Meshing details of fluid domain

Domain	Mesh Range	Type of mesh	Number of nodes
Impeller	Fine	Unstructured	237756
Inlet Duct	Fine	Unstructured	107265
Casing	Fine	Unstructured	347652

IV. Simulation of blower

The numerical simulation of the centrifugal blower is done in ANSYS-CFX. It solves incompressible Reynolds averaged Navier-Stokes equations and continuity equations. The governing equations are analyzed using a finite volume formulation. Discretizations of convection and diffusion terms of the equations are adopted by modified upwind scheme and central difference scheme, respectively. K-ε turbulence model with scalable wall function is employed to estimate the eddy viscosity. The impeller region is operating in rotating frame of reference and casing region in stationary frame of reference. As boundary conditions, suction pressure is specified at the inlet, and mass flow rate is specified at the exit. No-slip and adiabatic wall conditions are used on blade, casing and hub surfaces. Boundary plane between the impeller and casing regions is imposed frozen rotor interference. Various values of flow coefficient and their respective mass flow rate are summarized in the table 4.

Table 4 Flow coefficient and corresponding mass flow rate

Case	Flow coefficient	Mass flow rate(Kg/sec)
1	0.4	1.81
2	0.3	1.358
3	0.2563	1.16
4	0.15	0.3789

V. Results and Discussion

The numerical simulation carried out for centrifugal blower in ANSYS CFX solver. The simulations are performed for different flow coefficients starting from 0.15 to 0.4. Simulations are carried out for same inlet boundary condition but different boundary conditions at outlet. From table 5, it can be observed that as the flow coefficient is decreased, pressure is increased and hence the pressure coefficient is increased [1].

Table 5 Values of flow coefficient, average pressure and pressure coefficient.

Flow coefficient	Average pressure	Pressure coefficient
0.4	108.8	0.089
0.3	359.613	0.188
0.2563	519.6645	0.251
0.15	836.54	0.376

Figure 3 shows the variation of the pressure coefficient with the flow coefficient which matches with the theoretical behavior of the variation of the same. [6]

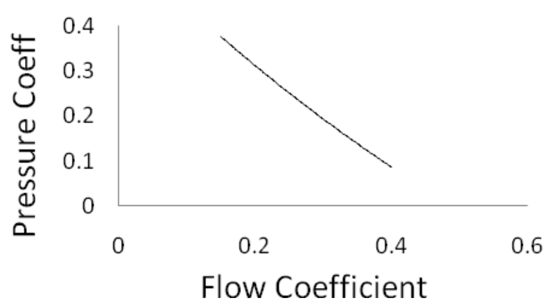


Fig.3 Variation of pressure coefficient with flow coefficient

International Journal of Electronics, Communication & Soft Computing Science and Engineering
ISSN: 2277-9477, Special Issue of" International Conference on Recent Trends in Engineering & Technology "Apr-2012

VI. Comparison of present cases for flow properties

Comparisons of different cases are carried out by flow analysis of simulated data. For comparison of different flow coefficients, outlet plane of domain is selected. On this plane, results of numerical simulation are analyzed and represented in figure 4 to figure 6 in the form of contours of flow properties. Figure 4 represents the contours of static pressure for different flow coefficient. It is observed that as the flow coefficient i.e. mass flow rate reduces, the pressure increases. This is because for reduced mass flow rate, the volume of the air is also reduces. All other physics of the problem is same such as power input, rotation and geometry so this reduced volume causes increase in the pressure as well as contours. Also, there are more colour contours observed at lower mass flow rate compared to higher. The contours of the total pressure are shown in figure 5. It reveals that for higher flow coefficient the value of total pressure is less and as the flow coefficient is reduces the value of total pressure is increases. As we move towards lower flow coefficient, the gradients of contours are reduced. The contours of the velocity show that with the increase in flow coefficient, velocity at the outlet is also increases as shown in figure 6. This is because the outlet area is same for all cases, so as mass flow rate is increased the velocities will also increase. Higher velocity contours are observed at upper side and it reduces radially outwards direction i.e. near impeller region.

VII. Detailed flow analysis for □=0.2563

The experimental value of flow coefficient is 0.2563. Therefore, detail flow analysis is carried out for the simulation case where □=0.2563. For detail analysis four different angular positions i.e. 90^0, 180^0, 270^0 and 360^0 and five different axial positions i.e. a to e where a is starting of impeller, b is mid of impeller, c is end of impeller and d and e are overhung positions are selected. The study of flow parameter like velocity and pressure are carried out at these locations.

Figure 7 shows the contours of pressure at different angular position for □=0.2563. It can be observed that as we move from suction to exit i.e. from 90^0 to 360^0 along the volute static pressure is increases. This is because in volute casing, the conversion of velocity into pressure takes place. This is the process of pressure recovery occurred in the volute casing. Also, near suction only one colour of contour is observed and as we move away more colours are observed.

Figure 8 shows the contours of the total pressure at different angular position for □=0.2563. Figure 8 depicts that as we move from inlet to outlet along the volute total pressure is also increases. Also, the colours of contours are reduced as we move towards the outlet.

Figure 9 shows the contours of the velocity at different angular position for □=0.2563.At the plane of 90^0, two regions of velocity are observed and these two regions are of high velocity and of low velocity. Also more colours are observed on this plane. But at the plane of 360^0, only the region of low velocity is observed. Fewer colours are observed on this plane. The velocity is reduced from 90^0 to 360^0 because the conversion of velocity into pressure. This is the process of the pressure recovery. The volute has served its function of conversion of velocity into pressure.

Figure 10 shows the contours of the pressure at different axial locations. Different regions of the pressure are observed into the domain. As we move away from the impeller fewer colours are observed. The pressure recovery can be observed from the figure 10. The pressure is seen to be increased along the volute.

Figure 11 shows the contours of velocity at different axial planes. High velocity contour is observed near the impeller region. This is because of the rotation of the impeller. As we move away from the impeller high velocity contours are reduces. The velocity is seen to be reduces from impeller to exit due to conversion of velocity into the pressure.

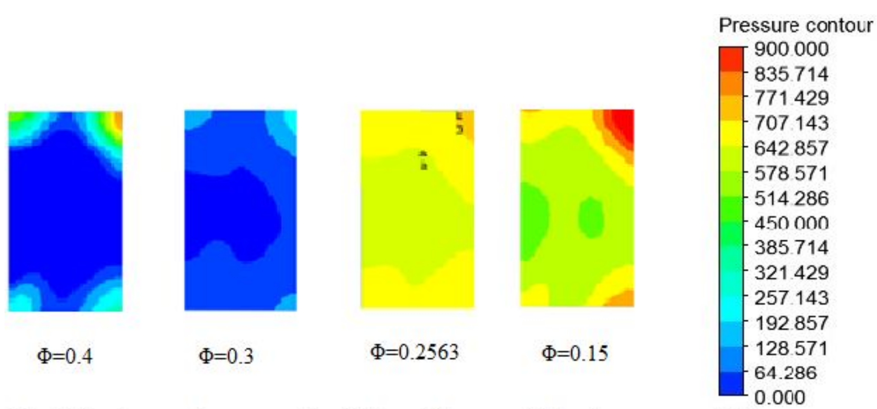


Fig.4 Contours of pressure for different flow coefficient

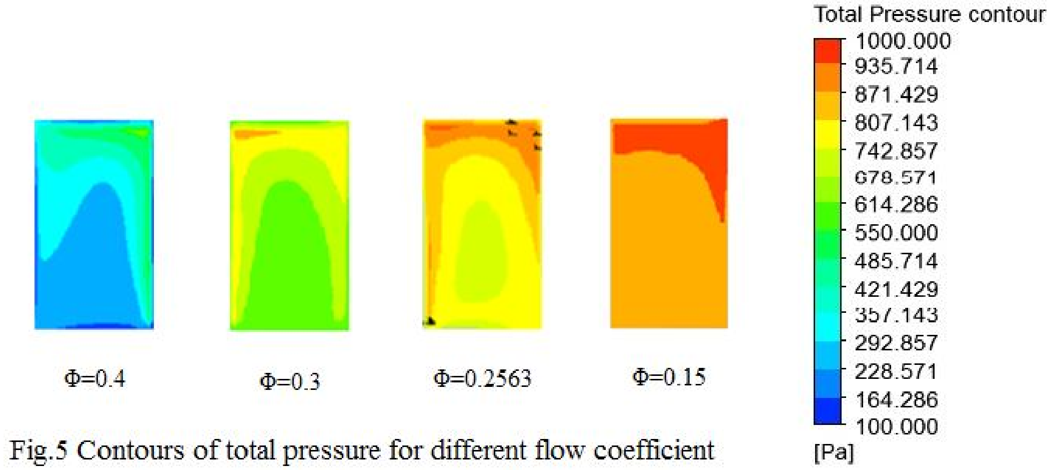


Fig.5 Contours of total pressure for different flow coefficient

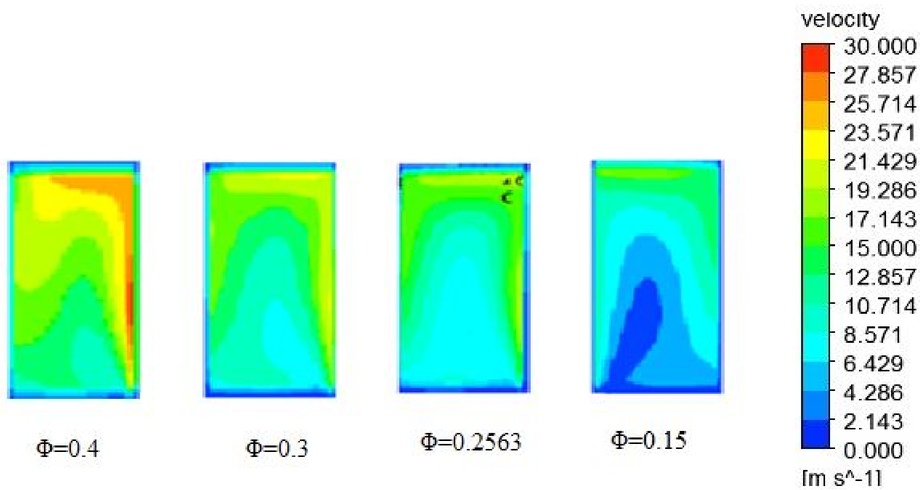


Fig.6 Contours of velocity for different flow coefficient

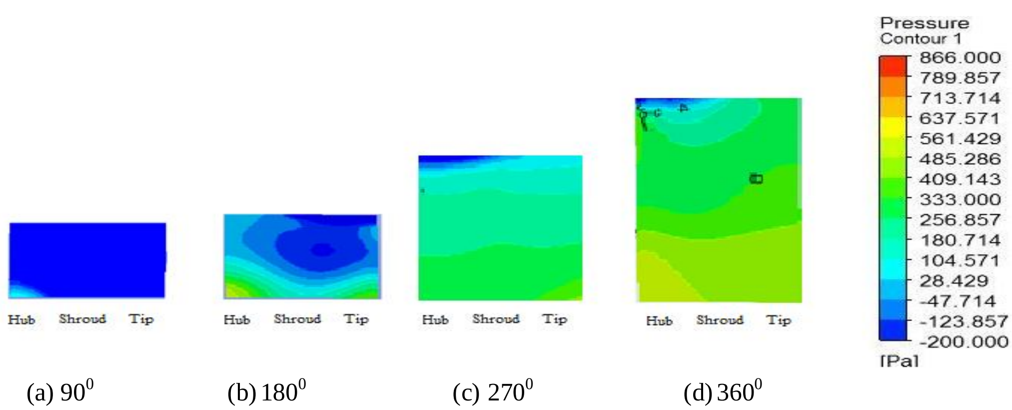
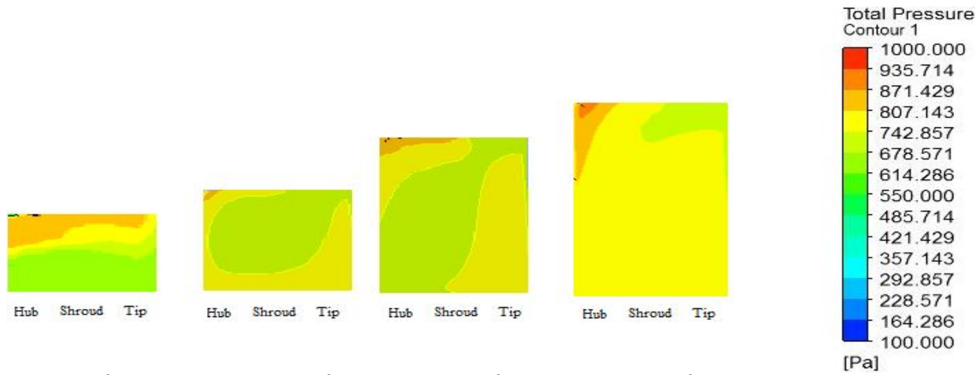


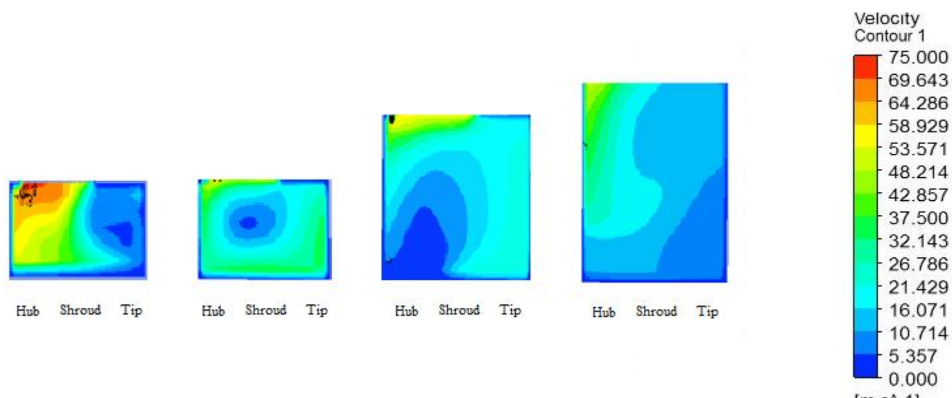
Fig.7 Contours of pressure at different angular positions of volute for Φ= 0.2563

International Journal of Electronics, Communication & Soft Computing Science and Engineering
ISSN: 2277-9477, Special Issue of" International Conference on Recent Trends in Engineering & Technology "Apr-2012



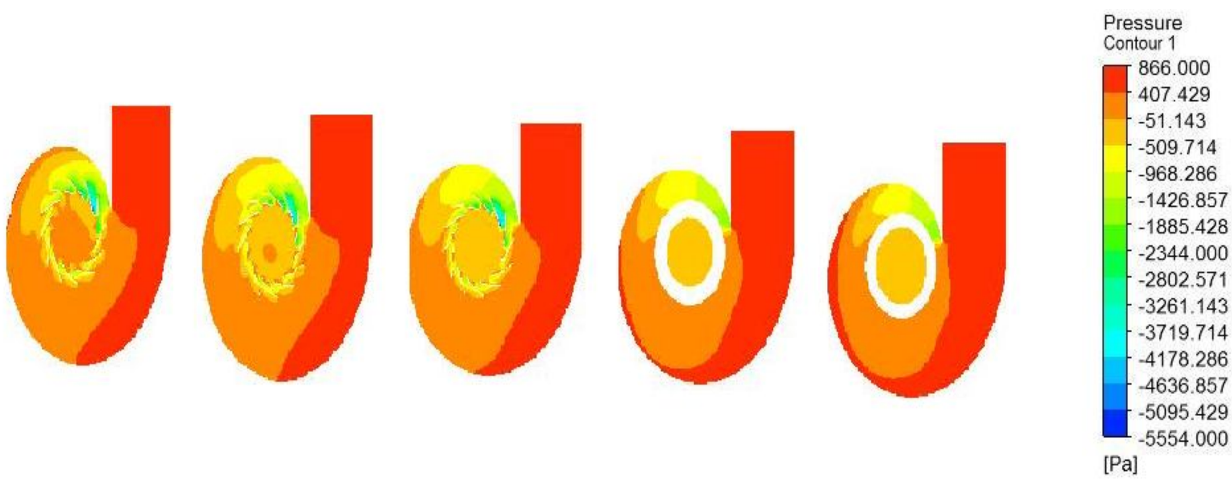
(a) 90^0 (b) 180^0 (c) 270^0 (d) 360^0

Fig.8 Contours of total pressure at different angular position for flow coefficient Φ= 0.2563



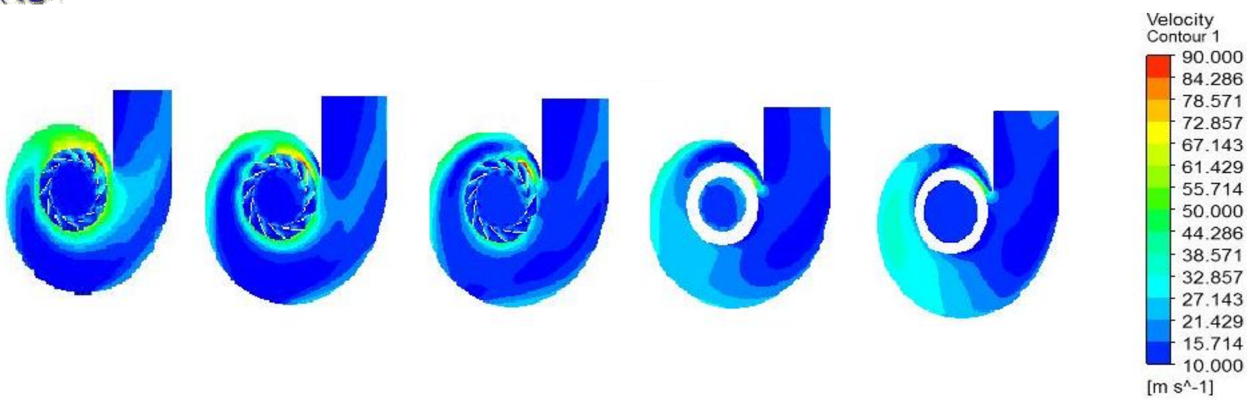
(a) 90^0 (b) 180^0 (c) 270^0 (d) 360^0

Fig.9 Contours of velocity at different angular position for flow coefficient Φ= 0.2563



(a) (b) (c) (d) (e)

Fig.10 Contours of pressure at different axial plane



(a) (b) (c) (d) (e)

Fig.11 Contours of velocity at different axial plane

VIII. Conclusion

1. The numerical analysis has been done to understand the effect of flow coefficient on the performance of the centrifugal blower. From the results of the numerical analysis, it can be deduced that the pressure coefficient decreases with the increase in flow coefficient.
   The flow analysis for different mass flow rate shows that static pressure and total pressure has been increased with the decrease in flow coefficient and velocity is increased with the increase in flow coefficient.
2. Detailed flow analysis foe experimental value of flow coefficient of □=0.2563 is also carried out at different angular positions of volute and axial positions which shows that as we move along the volute from suction to exit, static and total pressure increases and velocity is reduces. So, it can be concluded that the volute serves its function of pressure recovery.

REFERENCES

[1] Choon-Man Jang, Gi-Young Han, Enhancement of Performance by Blade Optimization in Two-Stage Ring Blower, Journal of Thermal Science Vol.19, No.5 (2010) 383−389.
[2] Choon-Man Jang, Optimal Operation of Turbo Blowers Serially Connected Using Inlet Vanes, Journal of Thermal Science Vol.20, No.1 (2011) 26-31.
[3] Jin-Hyuk Kim, Jae-Ho Choi, Kwang-Yong Kim, Design optimization of centrifugal compressor impeller using radial basis neural network method, Proceedings of ASME Turbo Expo 2009: Power for Land, Sea and Air
[4] T. Meakhail, S. Park. "A Study of Impeller-Diffuser-Volute Interaction in a Centrifugal Fan". Journal of Turbo machinery, Vol. 127, PP. 84-90, 2005.
[5] F. Gu, A. Engeda. "A Numerical Investigation on the Volute/Impeller Steady-State Interaction due to Circumferential Distortion". Proceedings of ASME Turbo Expo 2001, pp. 1-8, 2001.
[6] S.M.Yayha "*Turbines compressors and fans*" third edition 2005 by Tata McGraw Hill publishing company limited, Delhi.
[7] Church AN. Centrifugal pumps and blowers. Joftr.witey and Sons, Hemisphere Publishing Corporation; 1989.
[8] CFX user manuals, ANSYS release

AUTHOR'S PROFILE

Yogesh R. Pathak Completed B. E. Mechanical from SSVPS's College of Engineering Dhule. Presently, studying at SVNIT Surat in M. Tech. with specialization as Turbo Machines

Beena D. Baloni Working as Assistant Professor at SVNIT, Surat. Presently, doing Ph.D. under the guidance of Dr. S. A. Channiwala. Completed B.E. Mechanical and M.E.with specialization Gas Turbine and Jet Propulsion from MS university, Baroda. Had guided 12 students for their M. Tech. project. The research areas are Turbomachines, Jet propulsion, Compressible fluid flow, Internal Combustion Engine

Dr.S. A. Channiwala Working as professor at SVNIT, Surat. Completed B. E. Mechanical from South Gujarat Univ., Surat. Completed M. Tech. from I.I.T. Powai. Completed Ph.D. from I.I.T. Powai. Had Guided 64 students for their M. Tech. project and 8 students for Ph.D. The research areas are Turbomachines; Hydrogen fuelled engines, Dedicated CNG engine technology, Hybrid Electric vehicle, Gas turbine combustion chamber, Combustion etc