Proceedings of the 4th International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'17)
Toronto, Canada – August 21 – 23, 2017
Paper No. 109
DOI: 10.11159/ffhmt17.109

Numerical Analysis of the Centrifugal Compressor Stage for an APU

Beena D. Baloni, Ashutosh Singh, S. A. Channiwala
Deptartment of Mechanical Engineering SVNIT
Surat, Gujarat-395007, India
pbr@med.svnit.ac.in; ashu.pmg@gmail.com; sac@med.svnit.ac.in

Abstract **-** The paper describes numerical analysis of the centrifugal compressor used in an auxiliary power unit by using *CFD* software *ANSYS* 15.0. The fluid domain of centrifugal compressor comprises of impeller, channel diffuser and volute casing. The impeller is designed based on 1-D calculations and impeller geometry is developed based on a code for the blade generation using Kaplan method. Present techniques of correlations are used to develop the diffuser and volute. Whereas; geometry of diffuser and volute are developed using *SOLIDWORKS* 2013. The components of the compressor stage are individually meshed in the MESH component of *ANSYS* 15.0 workbench. The turbo mode of *CFX* 15 is used to develop the setup for analysis. The shear stress turbulence model is used as turbulence model. The *frozen rotor* interface is applied to take care of stator- rotor interactions. Inlet mass flow and pressure outlet at outlet are kept as boundary conditions. A High resolution advection scheme with first order numeric are chosen and convergence criteria is kept at 10^{-4}. Grid independence study and satisfactory comparison of simulation results with theoretical design calculations are also carried out. The validated simulation case is opted to study the effect of the volute tongue angle and change in shape of the diffuser blade on compressor performance. At the end, all the cases are compared with each other on the basis of uniformity in variation of properties and performance parameters like efficiency and pressure recovery.

Keywords: Centrifugal compressor, Channel diffuser, Numerical flow analysis, Auxiliary power unit

Nomenclature

b	Width, m	r	radius, m	ρ	Radius of the volute section, m
D	Diameter, m	Z	number of vanes	θ	Diffuser wall angle, degree
K	Volute design constant	α	Flow angle, degree		
Ns	Specific speed	β	Blade angle, degree		
p	pressure, N/m^2	η	Efficiency		

Subscript

1	Impeller inlet	3'	Volute inlet	t	tip
2	Impeller outlet	4	Volute outlet	v	volute
2'	Diffuser inlet	isen	isentropic		
3	Diffuser outlet	h	hub		

1. Introduction

Auxiliary Power Unit, APU is a secondary unit in various automobiles and aircrafts used for secondary functions. These secondary functions include startup of engine, ignition of the engine as well as an oil system. It also provides bleed air for the air-conditioning of the cabins in aircrafts. It drives a generator, thus providing AC electrical power and as a source of AC backup as well. The altitude limit for this function in aircrafts is limited to 20,000 feet. It consists of a gas turbine engine with other ignition, integrated oil and start systems. APU consists of load compressor, power section and gearbox system. Load compressor runs at the same speed as of the power section’s shaft [3, 4]. Centrifugal compressors are a sub-class of the dynamic axi-symmetric work absorbing turbomachine. It achieves a pressure rise by adding kinetic energy to the continuous flow of fluid through the impeller. The kinetic energy is then converted to an increase in static pressure by slowing the fluid through a diffuser. As the fluid flow past the impeller, the kinetic energy of flow increases.

As the flow moves past the diffuser, there is an increase in pressure due to the reduction in kinetic energy. The flow then passes through volute leading to recovery of pressure. The efficiency of compressor becomes important as the functioning of compressor is critical for the functioning of APU.

The experimental analysis of centrifugal compressor is an expensive deal. It is preferable to do preliminary design and analysis of the compressor with the use of a well-developed numerical simulation using fluid dynamics computation software like *ANSYS CFX* [5-7]. A Computer aided design procedure for impeller was discussed in detail by Whitfield [8]. Literature [9] suggests that changing the skewness of the impeller blade gives an increase in efficiency and pressure ratio. Cutting the vanes of impeller leads to increase in the pressure ratio with comparable efficiency and decrease in boundary layer separation [5]. Implementing motion to diffuser also provides an increase in efficiency [6]. The efficiency of the compressor can be increased by giving a velocity to the diffuser same as that of impeller [1]. One can derive an optimum flow angle at the diffuser exit with the optimization techniques [10]. The leading edge of volute is very important and its bluntness should be dealt with care for decreasing losses [11]. Cross section of volute as circular is best for an efficiency point of view [12] with an asymmetry leading to loss in pressure ratio [7].

The present work incorporates numerical analysis of the centrifugal compressor stage comprising of impeller, diffuser and volute. The fluid model is developed in *SOLIDWORKS* 2013 and analyzed in *ANSYS* 15.0 workbench. The effect on performance and flow inside the compressor stage is analyzed for different shape of diffuser and volute tongue. The flow property contours are studied and compared where, variations of pressure and efficiency are taken as the main factors in design. The better design from the selected cases is suggested at the end.

2. Numerical Simulation

Numerical simulation of the compressor stage comprises of impeller, diffuser and volute. They are used to analyze the performance of the centrifugal compressor. The impeller and straight channel diffuser for the compressor is designed with 1-D calculations and proper co-relations [2]. The free vortex method is preferred for the design of the compressor volute at high flow rates for efficiency point of view. The circular cross section is opted for the design of volute. The expression for the radius of the volute cross section is defined as represented in Eq. 1.

$$\rho = \frac{\theta}{K} + \sqrt{2\frac{\theta}{K} r_{3'}} \tag{1}$$

Design parameters of impeller, straight channel diffuser and volute are represented in Table 1. All the design components are developed in a *CAD* package of *SOLIDWORKS* 2013 and illustrated in Figure 1, 2, and 3. Components are meshed individually in the MESH component of *ANSYS* 15.0 workbench. For the meshing of the components, “Proximity and Curvature” is taken as an advanced size function with medium to fine relevance, medium smoothening and fast transition. Mesh curvature of 50° to 70° is taken with the minimum face size of 2 mm to 4 mm and maximum face size of 4mm to 9 mm. The meshing of a stage so developed can be illustrated in Figure 4. An optimum size of elements are finalized with the grid independence study of fluid domain. The results for the same are shown in Figure 5.

The mesh so developed is imported into *CFX* 15.0. The turbo mode is used to develop the setup for analysis. The shear stress transport model is used as turbulence model. The impeller is given a velocity of 6238.87 rad/s while diffuser and volute are stationary. A constant mass flow inlet of 1.164 kg/s, a constant pressure outlet of 3.58 bar and rest surfaces being the smooth wall are the boundary conditions applied to the fluid domain. The interface between the components, i.e. impeller-diffuser and diffuser-volute are kept as the *frozen rotor* type. High resolution is the advection scheme used with convergence criteria kept at RMS value of 10^{-4}.

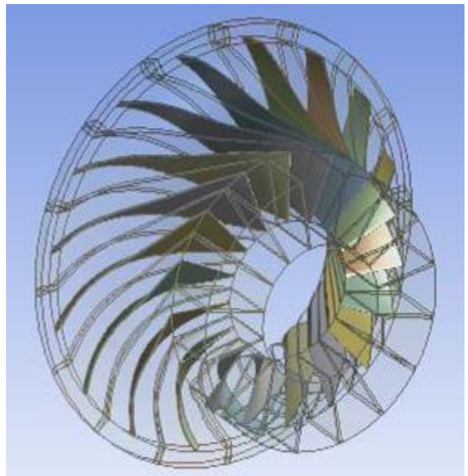
Fig. 1: Geometry of designed impeller.
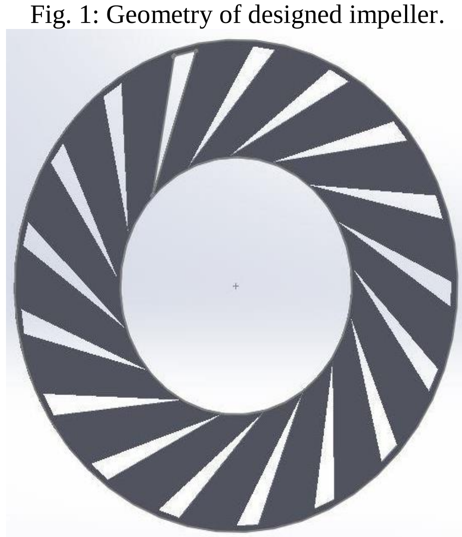
Fig. 2: Geometry of straight channel diffuser.

Fig. 3: Geometry of designed volute.

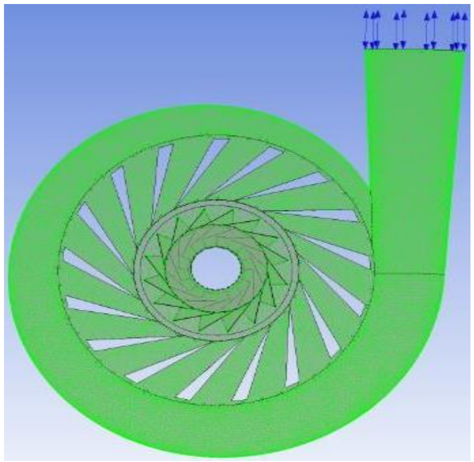

Fig. 4: Mesh of the centrifugal compressor.

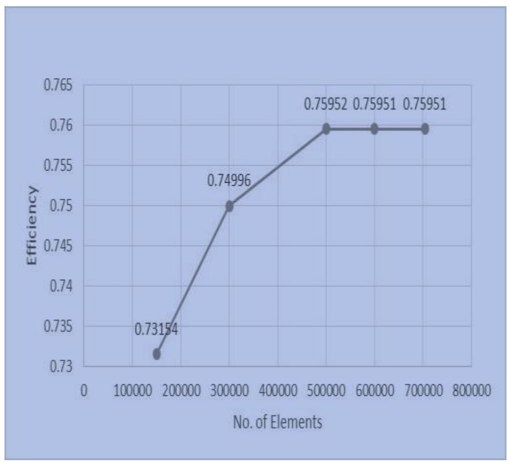

Fig. 5: The results of grid independence check.

Fig. 6: Geometry of curved channel diffuser.

Table 1: Design dimensional parameters of different components of radial compressor stage.

1. Radial impeller parameters [2]		
Non-dimensional Ns = 0.9	D_2 = 177.6 mm	β_2= 50.83°
η_{isen}= 0.85	α_1 = 30°	Z = 16
D_h = 59.2 mm	α_2 = 66.84°	b_2 = 8.02 mm
D_t = 118.4 mm	β_{1mean} = 56.66°	
2. Straight channel diffuser parameters [2]		
Area ratio = 2.07	$b_{2'}$ = b_3= 8.02 mm	Incidence angle = -4°
$D_{2'}$ = 191.8 mm	Z = 17	Length of channel = 82 mm
D_3 = 336 mm	$\beta_{2'}$ = 62.84°	Divergence angle, 2θ = 6^o
3. Circular volute parameters		
$D_{3'}$ = 362.9 mm	Volute throat = 38.4 mm	Tongue angle, α_v = 15.44°
D_4 = 439.6mm	b3' = 8.02 mm	Draft angle at volute outlet = 3°

After satisfactory comparison of designed compressor stage simulation results with theoretical design calculations, a validated simulation case is opted to study the effect of a volute tongue angle and change in shape of the diffuser blade on compressor performance. The straight blade diffuser replaced by the curved blade to analyze the effect of diffuser blade shape. The straight blades of diffuser are replaced with B-spline curves with 4 control points. The end points of the curve are same as that of straight bladed diffuser with the middle two points being taken in such a way that the variation of slope is uniform from an inlet to an outlet. The dimensions of curved bladed diffuser are similar to the straight bladed diffuser. Figure 6 represent the geometrical view of curved channel diffuser. The tongue angle from 11.44° to 19.44° with an interval of 2° is varied to analyze the effect of tongue angle on the compressor performance. The results of analysis are compared with each other for enhancement of performance of the compressor in terms of pressure recovery and efficiency.

3. Results and Discussion

Numerical simulation of designed centrifugal compressor stage, comprises of impeller, straight channel diffuser and circular volute, is carried out first. The detail flow analysis inside the different components of compressor stage is done with the help of different simulation values of flow properties and contour graphs of pressure variations and Mach number. Figure 7 represents the contours of pressure variations inside the fluid domain. The results suggest that rotational velocity imparted to the impeller increases the pressure, uniformly, up to 3.17 bar compared to a theoretical value of 3.337 bar [2]. As the flow moves past the diffuser, the flow deceleration leads to conversion of kinetic energy to pressure energy which increases the pressure to 3.47 bar whereas; the theoretical value is 3.49 bar [2]. The different area occupied by different colour contours inside the various vane passages of diffuser suggest non- uniform distribution of flow around the impeller outlet periphery and along the diffuser passage. Pressure loss is observed in between the impeller outlet and diffuser inlet periphery. Low pressure contours are observed almost in all diffuser vane passages near the throat region of the diffuser inlet. In two vane passages of diffuser (Refer: Fig. 7; 1^{st} near volute tongue and other at the 6^{th} position from tongue in anticlockwise direction) the sudden increase in pressure is observed near the throat of diffuser. This indicates an occurrence of the shock phenomenon in the flow. Also, one can observe high pressure contours inside the diffuser vane passages near volute tongue compared to other diffuser vane passages. This may be due to the tongue effect, as an area near tongue region is less compare to other regions of volute which leads to high pressure near these regions. The simulation temperature is 300K at the inlet which increases to 401 K at the impeller exit with 2.82% variation from the theoretical temperature of 390K [2]. As the flow moves through diffuser, the temperature further increases to 510K at the diffuser exit compared to the theoretical value of 490K [2]. The recovery of pressure leads to increase in temperature to 570K at the compressor stage exit.

The variations of the Mach number inside the fluid domain is shown in Fig. 8. This reveals that at the inlet of the compressor, the flow is subsonic at eye hub and transonic near the impeller eye tip. As the rotational velocity is imparted to the fluid, the Mach number increases and at the exit of the impeller its value is 0.76. Whereas; theoretical value is 0.82 [2].

When the flow enters the diffuser, some regions near the throat portion of the diffuser experience sudden increase in Mach number. Supersonic flow is observed near the throat regions of two vane passages. These are the same passages where, the pressure contours suddenly increases; and at these locations, flow is converted from supersonic to subsonic which indicates the formation of shock. Numerical simulation results suggest Mach number 0.73 at the exit of diffuser while theoretical value is 0.79 [2]. The results of simulation suggest non uniform distribution of flow inside the blade passages of diffuser. This leads to recirculation of flow at some regions near the diffuser wall (Refer: Fig. 8). Comparatively less variations are observed in the contours of the Mach number within the volute.

To check the effect of diffuser vane shape, straight diffuser is replaced with curved vane diffuser by keeping all other design parameters same for the compressor stage. Numerical simulation is carried out with a curved shape of the compressor stage and results are plotted. The contours of the pressure variations are represented in Fig. 9. Results indicate an increase in pressure, uniformly, up to a 3.32 bar at an outlet of impeller which shows 0.9 % variation from the theoretical design value of a 3.337 bar [2]. The non-uniformity at impeller outlet is observed at regions of impeller outlet near the volute tongue. This may be due to the effect of the volute tongue. The curved diffuser decelerates the flow and increase the pressure to 3.5 bar compared to the theoretical value of 3.49 bar [2]. The increasing cross section of the volute recovers the pressure from conversion of kinetic energy and the pressure at outlet is 3.58 bar. The high pressure contours are observed at an impeller outlet and diffuser vane passages near the volute tongue, due to the flow obstruction because of the minimum flow area at the tongue. Compared to straight diffuser the contour bar shows the high value of maximum pressure in case of curved diffuser. The high pressure at each location of the fluid domain is clearly observed in case of curved diffuser as compared to straight diffuser. Also, the low pressure contours near the throat of diffuser passages are reduced in case of curved one compare to straight diffuser. The temperature is at 300K at the inlet which increases to 410K at the impeller exit with 5.82% variation from the theoretical temperature of 390K [2]. Temperature further increases to 502K at the diffuser exit compared to the theoretical value of 490K [2]. Whereas; temperature is 540K at the exit of the compressor stage. Thus, in case of curved diffuser temperature is lower by 30K as compared to straight diffuser.

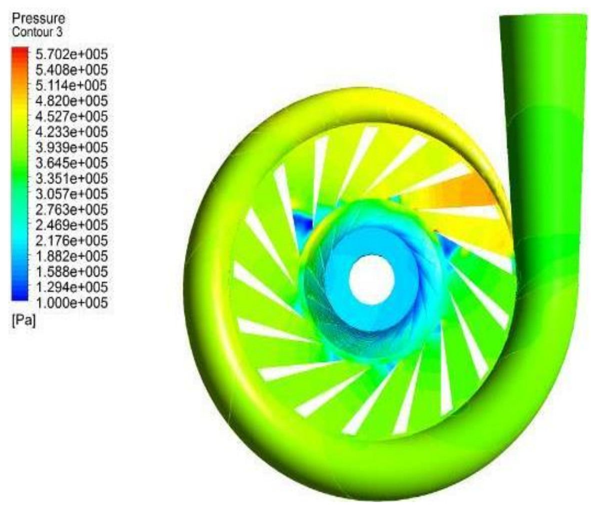

Fig. 7: Pressure variations of designed compressor (Straight diffuser, α_v= 15.44°).

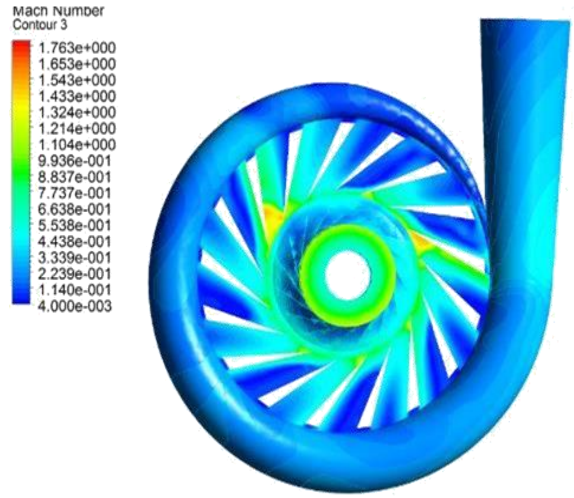

Fig. 8: Mach number variations of designed compressor (Straight diffuser, αv= 15.44°).

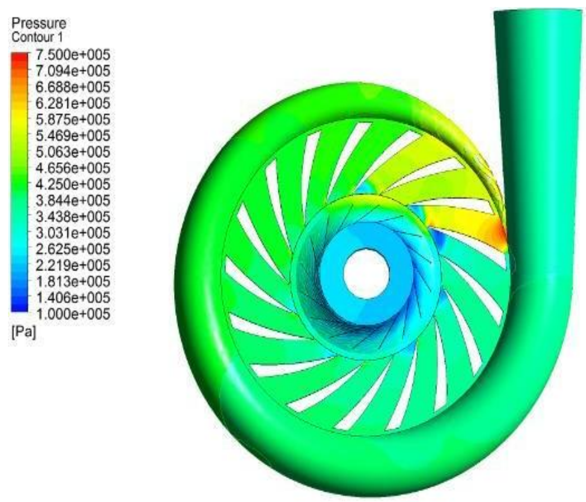

Fig. 9: Pressure variations of designed compressor (Curved diffuser, $\alpha_v = 15.44°$).

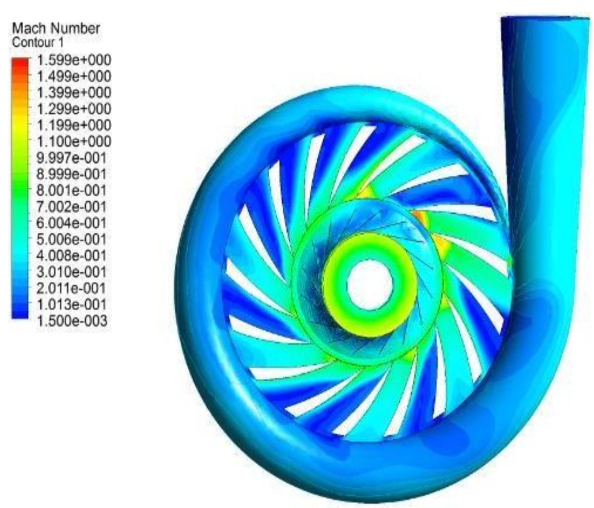

Fig. 10: Mach number variations of designed compressor (Curved diffuser, αv = 15.44°).

Figure 10 reveals the variation of the Mach number inside the fluid domain. From the subsonic value at entry, the Mach number rises to 0.7 at the exit of impeller compared to the theoretical value of 0.8243[2]. Diffusion inside the diffuser decreases the Mach number to 0.68 whereas; theoretical value is 0.79[2]. Within volute, it further decreases due to decrease in velocity and recovery in temperature. The Maximum value of the Mach number is reduced in case of curved diffuser compared to straight one. The non-uniform region close to the diffuser throat decreases considerably compared to straight diffuser. Also, an area occupied by low Mach number contours, at vane passages of diffuser, are comparatively less than straight diffuser. These indicates better flow distribution than in case of straight diffuser.

Both simulation results indicates flow non- uniformity inside the compressor near the tongue regions. Therefore, the analysis is further extended to check the effect of tongue angle on compressor flow. The tongue angle of the volute is changed from 11.44° to 19.44° with the interval of 2°. The numerical setup is developed in a similar manner as in the previous two cases and compared with each other on the bases of flow properties and performance parameters of compressor. The contours of pressure variations inside the fluid domain, at a different tongue angle from 11.44° to 19.44° for straight and curved diffuser, are shown in Fig. 11. In the case of straight diffuser, low pressure regions at diffuser throat increases with an increase in tongue angle from 11.44° to 17.44° and then the slight decrease in the case of 19.44° compared to the case with tongue angle 17.44°. Whereas; these low pressure regions are observed only at the throat of the diffuser vane passages near volute tongue in curved diffuser. One can also observe high pressure inside the diffuser vane passages near volute tongue in all cases of the curved diffuser fluid domain compared to straight diffuser. More uniform pressure contours are observed for tongue angle 11.44° with straight diffuser amongst all cases. Figure 12 reveals the variations in Mach number for straight and the curved diffuser at a different tongue angle. One can observe more supersonic flow contours at some of the diffuser throat in the case of straight vane compare to the curved vane. Maximum supersonic Mach number contours are observed in straight diffuser with 17.44° tongue angle. The contours of Mach number also suggest more uniform flow distribution in the case of the curved vane compared to straight vane diffuser.

Table 2: The results of numerical analysis for various cases of compressor stage.

Diffuser blade type	Volute tongue angle (Degree)	P_1 (bar)	P_2 (bar)	P_3 (bar)	P_4 (bar)	Total stage pressure ratio	η_{total}	Pressure recovery in impeller	Pressure recovery in diffuser	Pressure recovery in volute
Straight	11.44°	1	3.42	3.53	3.58	3.58	0.7699	0.6759	0.031	0.013
Straight	13.44°	1	3.3	3.47	3.58	3.58	0.7847	0.6424	0.047	0.031
Straight	15.44°	1	3.17	3.47	3.58	3.58	0.7595	0.606	0.084	0.030
Straight	17.44°	1.1	3.3	3.5	3.58	3.25	0.7683	0.614	0.055	0.022
Straight	19.44°	1.1	3.3	3.5	3.58	3.25	0.7814	0.616	0.055	0.022
Curved	11.44°	1.03	3.31	3.55	3.58	3.48	0.7791	0.636	0.067	0.008
Curved	13.44°	1	3.39	3.48	3.58	3.58	0.7810	0.667	0.025	0.027
Curved	15.44°	1	3.32	3.5	3.58	3.58	0.7706	0.6480	0.050	0.022
Curved	17.44°	1	3.27	3.51	3.58	3.58	0.7849	0.634	0.067	0.019
Curved	19.44°	1	3.32	3.51	3.58	3.58	0.7809	0.648	0.053	0.019

For better comparison, numerical results for all cases are compiled and represented in the Table 2. The details of the table reveals that maximum impeller pressure recovery (i.e. 0.6759) is obtained in the case of straight diffuser with tongue angle 11.44° whereas; the minimum pressure recovery of 0.614 occurs in the case of straight diffuser with 17.44° tongue angle. Pressure recovery from impeller is better in the case of curved diffuser compared to the straight one for all tongue angle except 11.44°. Pressure head from impeller and diffuser is good in the case of curved diffuser with volute angle 11.44° followed by straight

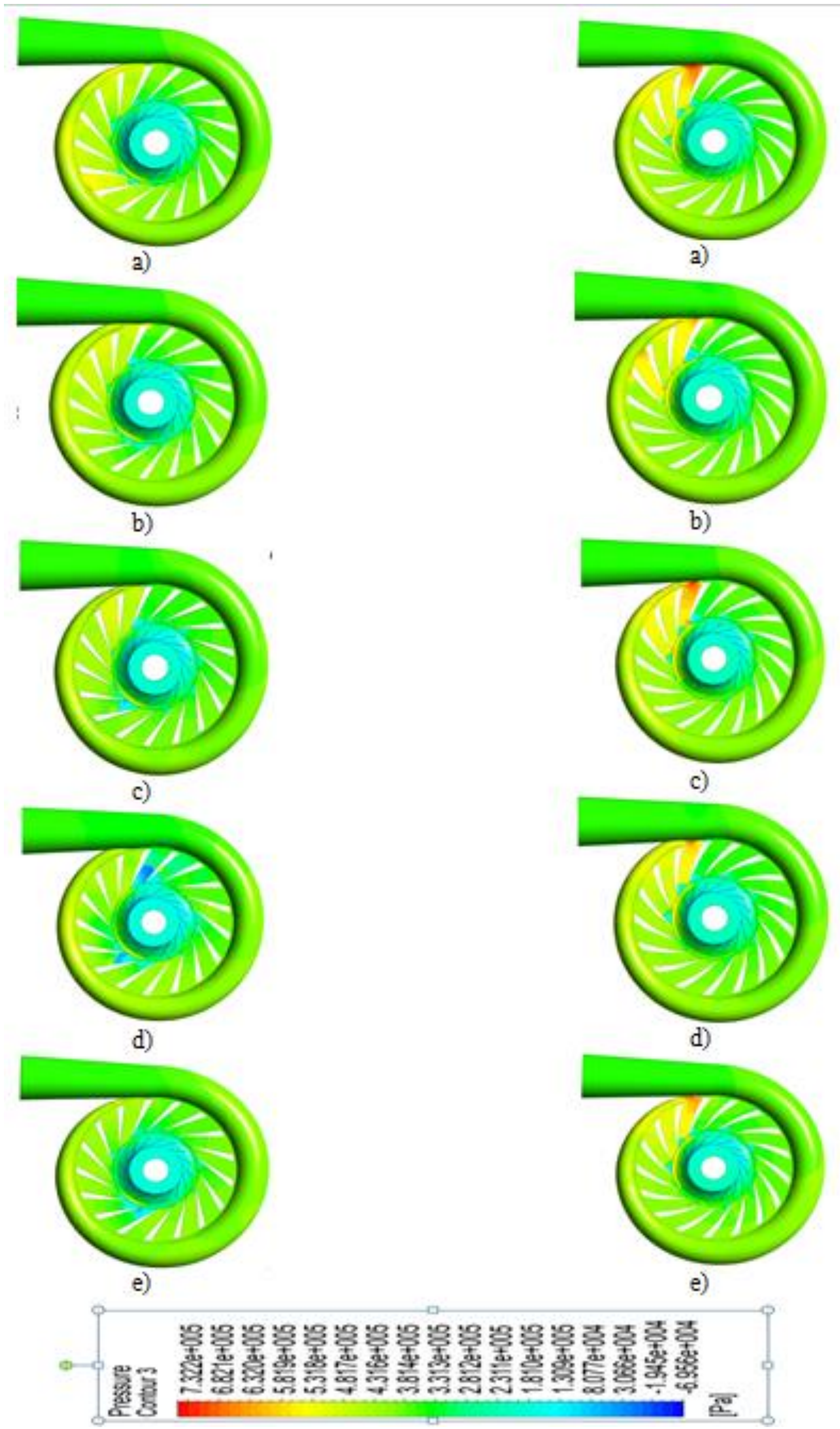

Fig. 11: Contours of pressure variations for straight and curved diffuser at a different tongue angle.

a) 11.44° b)13.44° c)15.44° d)17.44° e)19.44°

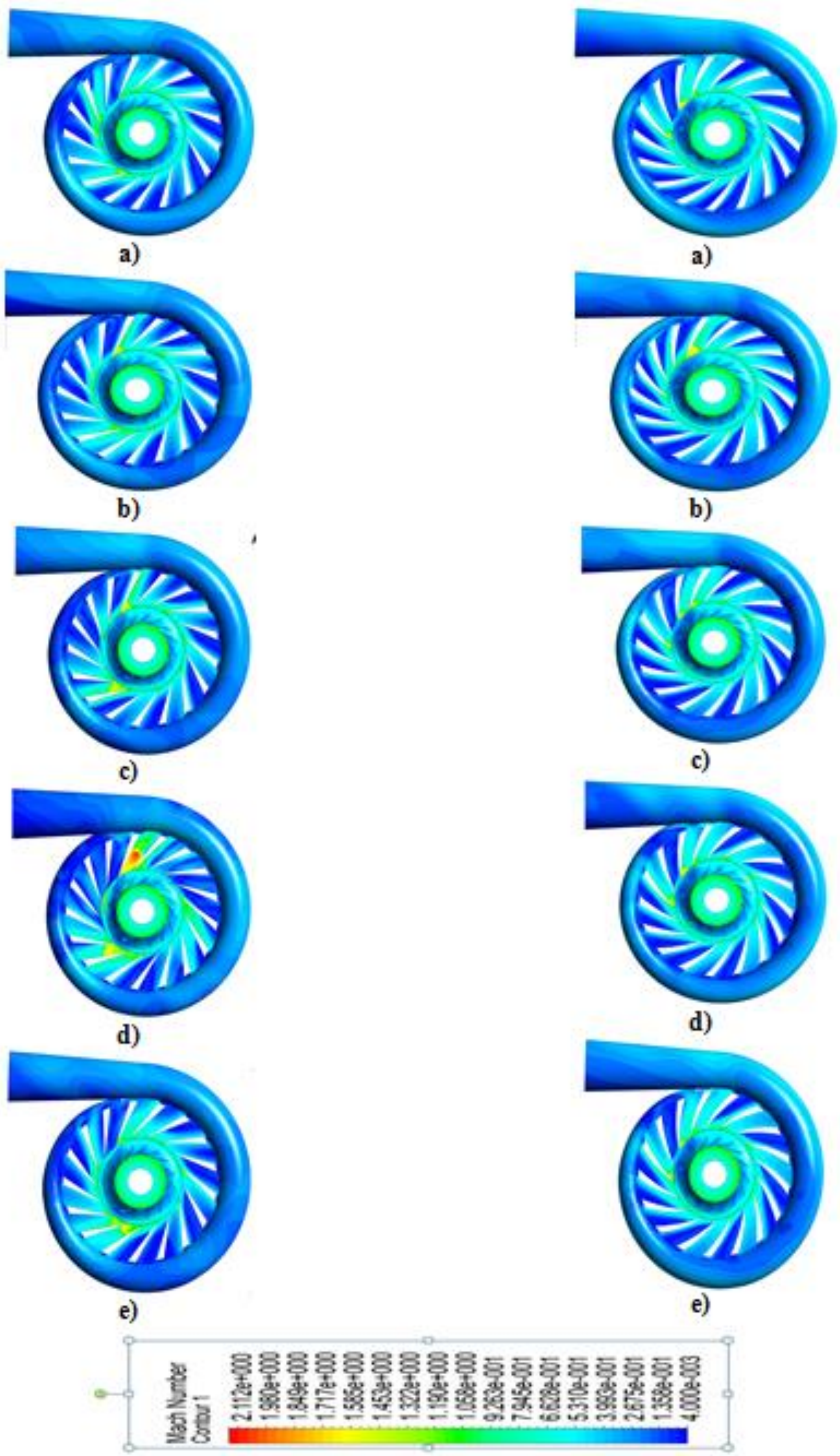

Fig. 12: Contours of the Mach number for straight and curved diffuser at a different tongue angle.
1) 11.44° 2)13.44° 3)15.44° 4)17.44° 5)19.44°

Though, the pressure recovery with 11.44° tongue angle is better, efficiency is observed the maximum in the case of straight diffuser with 13.44° tongue angle and curved diffuser with 17.44° tongue angle. Compressor with tongue angle 11.44° and straight diffuser has efficiency of 77%, 1.5 % less than maximum value. The results suggest that the combination of straight diffuser with 11.44° tongue angle gives the best pressure ratio but a little bit lesser efficiency. Curved diffuser with 11.44° tongue angle gives comparable pressure ratio and efficiency with optimal values. Curved diffuser gives more efficiency compared to straight diffuser due to more flow uniformity inside the compressor stage. However, the development of curved vane diffuser is a difficult task compared to straight one.

4. Conclusions

Numerical simulation of an APU compressor stage comprises of impeller, diffuser and volute is carried out to understand flow physics inside the compressor. The different configuration of the volute tongue angle is analyzed with straight and curved blade channel diffuser. Following conclusions are obtained from the above analysis of centrifugal compressor:

1. The pressure head from an impeller inlet to exit is highest in case of straight diffuser and 11.44° volute angle.
2. The pressure recovery is highest in case of straight diffuser and 11.44° volute tongue angle followed by curved diffuser of the same volute angle.
3. The curved diffuser is chosen as a better choice for diffuser on the basis of flow uniformity, efficiency and pressure recovery.
4. For the present design of compressor, volute of tongue angle 11.44° is chosen due to increase in uniformity in variation of properties, efficiency and pressure recovery.

References

[1] O. E. Balje, "*Turbomachines, A guide to design, selection and theory,*" Ch. 4, 1st Edition, John wiley & sons publication, New York, 1981.

[2] MehaSetiya, "Design and Numerical Analysis of Centrifugal Compressor," *Dissertation report, M. Tech-Turbomachines, SVNIT*, Surat, 2015.

[3] R. Aghazi, A. M.Tousi, M. Soltani "Design and CFD analysis of centrifugal compressor for a microgasturbine," *Aircraft Engineering and Aerospace Technology,* vol. 79, no. 2, pp.137-143, 2007.

[4] H. Pitkanen, H. Esa, A. Reunanen, J. Larjola, P. Sallinen, "Computational and Experimental Study of an Industrial Centrifugal Volute," *Journal of Thermal Science*, vol. 9, no. 1, pp. 77-84, 2000.

[5] J. T. Mo, C. H. Gu, X. H. Pan, S. Y. Zheng, "Enlarging the operation range of a centrifugal compressor by cutting vanes based on CFD," *IOP Conference Series: Materials Science and Engineering*, vol. 52, no. 4, 2013.

[6] H. B. Jin, M. J. Kim, W. J. Chung, "A Study on the Effect of Variation of the Cross-sectional Area of Spiral Volute Casing for A Centrifugal Pump," *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering,* vol. 6, no. 8, pp. 1756-1765, 2012.

[7] J. M. Sorokes, and J. P. Welch, "Experimental results on a rotatable low solidity vaned diffuser," *International Gas Turbine and Aeroengine Congress***,** ASME, no. 92-GT-19, 1992.

[8] F. J. Wallace, A. Whitfield, R. Atkey, "A Computer-Aided design procedure for radial and mixed flow compressors," *Computer Aided Design*, vol. **7**, pp.163-170, 2003.

[9] V. C. Arunachalam, Q. H. Nagpurwala, M. D. Deshpande, "Numerical Studies on the Effect of Impeller Blade Skew on Centrifugal Compressor Performance," *SASTECH – JOURNAL*, vol. 7, no. 2, pp. 33-40, 2008.

[10] C. Ji, Y. Wang, L. Yao, "Numerical Analysis and Optimization of the Volute in a Centrifugal Compressor," *International conference of power engineering*, pp 1352-1356, 2007.

[11] P. M. Lingrani, R. Van Den Braembussche and M. Roustan, "Measurement in the vaneless diffuser of a radial flow compressor," *Int. J. Heat and Fluid*, Technical note, vol. 4, no. 2, 1983.

[12] B. Sahoo, G. Gouda, "Failure analysis of compressor blade of typical Fighter-class Aero-engine-A Case Study," *Defense science journal*, vol. 52, no. 4, pp. 363-367, 2002.