6th INTERNATIONAL SCIENTIFIC CONFERENCE ON DEFENSIVE TECHNOLOGIES
OTEH 2014
Belgrade, Serbia, 9 – 10 October 2014

COMPUTATIONAL FLUID DYNAMICS OF THE TURBULENT AIR FLOW THROUGH A VANED DIFFUSER TURBOCHARGER

ABDELMADJID CHEHHAT
Mechanical engineering department, Faculty of technology, Abbes Laghrour University, Khenchela(40000), Algeria, achehhat@gmail.com

MOHAMED SI-AMEUR
Laboratory of Energetic and Industrial Systems, University of Batna(05000), Algeria, msiameur@yahoo.fr

Abstract: *In this work a numerical study with moving reference frame (MRF) technique is made. Many research works both experimental and numerical on the impeller-diffuser interactive phenomenon have been undertaken so far. But it is found from the literature that the study on the impeller-diffuser-volute interaction as well on the performance of the turbocharger centrifugal compressor by varying the number of diffuser vanes has not been the focus of attention in these works. Hence a numerical analysis was accomplished in order to extensively explore impeller-diffuser fluid interaction in a real turbocharger used in military engines applications, when the number of diffuser vanes was changed at unaltered number of the impeller blades. It was concluded that the number of diffuser vanes gives a considerable effect on the pressure and temperature at the compressor outlet.*
.

Keywords: *centrifugal compressor, turbocharger, diffuser vanes, CFD analysis*

1. INTRODUCTION

The turbocharger has accompanied the development of automobile diesel engines, and will be more and more imposed on the S.I engines to improve the performance and reduce the pollutant emission, for this reason, the centrifugal compressor has been widely used with an exhaust recirculation technique (EGR). Actually this technique can lead to the production of more than 50% of the engine total intake charge. However, the turbocharger compressor performance is limited by the engine operating conditions. In this context and in the last years, several investigations have been analyzed the centrifugal compressor performance alteration with its elements design. Two numerical studies of K Jio et al (2009) have shown that: - the vaneless diffuser and the open angle diffuser give the stable operating range and high efficiency. – the study of the air flow through the turbocharger compressor with dual volute design, revealed that the dual volute design could separate the compressor into two regions treated separately with dual diffuser design, this investigation confirmed that the dual volute design improve a stable operating range comparing with single volute design. The impeller-diffuser - interaction in a centrifugal compressor have been studied by Anish et al (2007), four different types of diffuser configurations were generated, by varying the radial gap between the impeller and diffuser. It is shown that the dependence of the compressor stage efficiency is well correlated to such interaction. Y Dai et al (2008) studied the performance of two different volutes with the same impeller. A good agreement has been obtained between the numerical computation and the experimental measurements in volutes, at least at moderate mass flow rate. It is well known that the turbulence and flow field studies, may also give a good idea on the pressure variation in the compressor, in this subject, Ali Penarbasi (2009) presented a detailed flow measurements at inlet plane of a centrifugal compressor vaneless diffuser and concluded that four regions of high shear are identified within the flow : within the blade wake, between the passage wake and jet, within the thickened hub boundary layer and between the blade wake secondary flow passage vortex. This study confirmed that each of these regions is associated with high turbulent kinetic energy. R Aghaaei (2008) et al proposed a comparison of turbulent methods in CFD analysis of radial turbo machines and introduces the best way to choose turbulence parameters when using FLUENT software, and confirmed that the standard k-ε and RNG k-ε models are superior turbulence methods in CFD analysis of radial turbo machines. It can be observed from most of these numerical studies that the effect of number of diffuser vanes on system performance as well as on the flow field and turbulence in the turbocharger centrifugal compressor. Therefore, in order to analyse the effect of diffuser vanes number on the compressor performance, in this study four types of diffuser are investigated with the same impeller and volute, they only differ in their vanes number (5, 7, 9 and 11 vanes). For this purpose, a commercial CFD software FLUENT 6.3 is employed considering a steady state method, for simulating the three-dimensional turbulent air flow through the full stage of a centrifugal compressor, used in turbocharger of automobile diesel engine, It is found from the analysis that the number of diffuser vanes presents a considerable effect on the pressure and temperature at the compressor outlet.

2. COMPRESSOR GEOMETRY

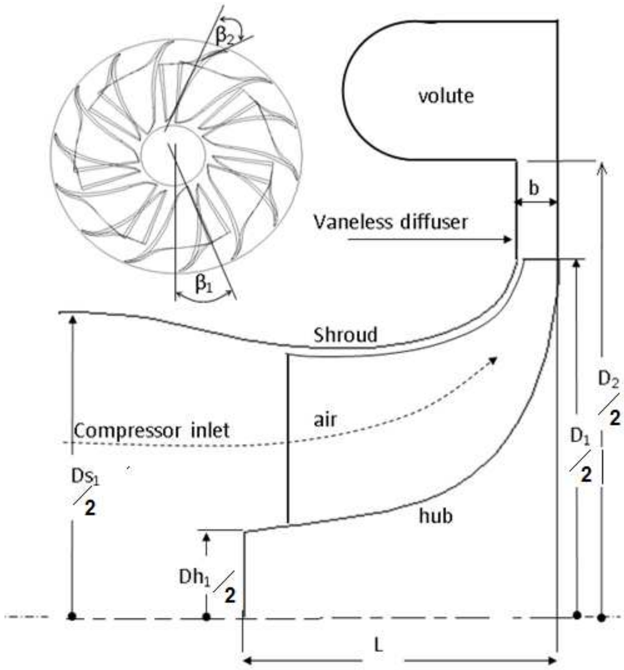

Figure 1. Schematic of the centrifugal compressor stage

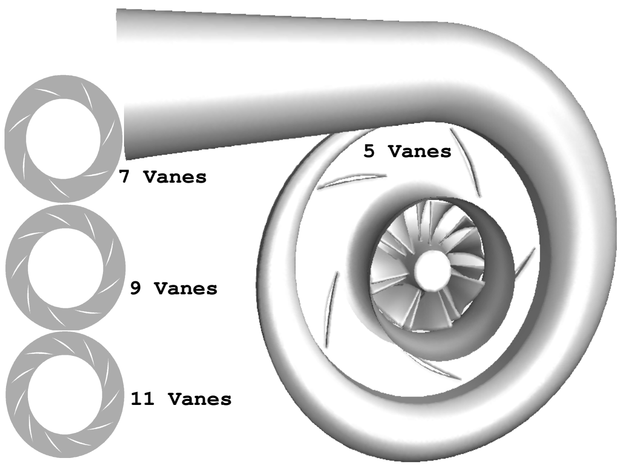

Figure. 2. Centrifugal compressor stage designed using GAMBIT with different diffuser vanes number

The compressor geometry development is carried out on GAMBIT software the various components of the compressor are designed individually and all are assembled. The blade profiles are generated with help of coordinates that have been introduced, a single blade profile is made and then the blades are arranged by further rotating over an angle 360°/z , z is number of blades. For generating the volume of impeller inside the shroud volume, the blade volume must be subtracted from the shroud volume, a single rotational volume is obtained, it is limited by the impeller inlet surface, outlet surface, and hub and shroud surfaces, including the blade passage. The diffuser is modeled as a hollow disc, its vanes are modeled as an airfoil located in base angle position. The dimensions of the impeller and the diffuser are shown in the (table 1). The volute is modeled by assembling a casing of non-symmetrical cross section beginning from 0° ending at 360° (when a tongue is located) and a conic pipe discharging in engine admission manifold.

Table 1: Geometrical data for the designed compressor, see fig. 1 for the nomenclature

Description	Symbol	Dimension
Number of full blades	z	7
Number of splitter blades	z_s	7
Impeller outlet diameter	D_1	0.08 m
Diffuser outlet diameter	D_2	0.126 m
Impeller outlet vane height	b	0.0045 m
Inlet shroud diameter	D_{s1}	0.068 m
Inlet hub diameter	D_{h1}	0.018 m
Impeller axial length	L	0.026 m
Inlet mean line blade angle	β_1	50°
Outlet blade angle (back sweep)	β_2	30°

3. NUMERICAL PROCESS

The high complexity of the flow in the rotating impeller makes the CFD modelling very difficult, only steady state flow is investigated, the governing equations implemented in a commercial CFD code Fluent 6.3.26 are solved using the finite volume method, with the most appropriate discretization scheme (pressure based implicit solver), the fluid (air) is treated as an ideal gas, the momentum and energy equations are solved using first order scheme, the turbulent kinetic energy and dissipation rate equations are solved using the power low differencing scheme (PLDS) the coupling velocity-pressure correction is treated with SIMPLE (Semi-Implicit Pressure Linked Equation) algorithm.

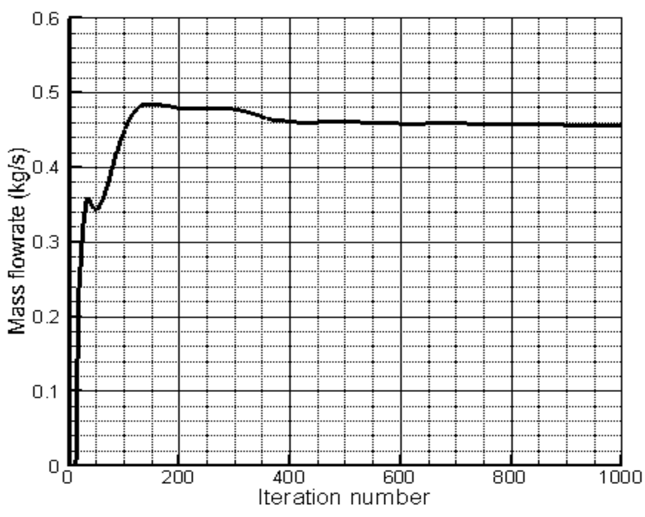

Figure 3. Monitoring of the mass flow rate computed over the compressor outlet during the iteration process

The under relaxation factor for the pressure is taken 0.2 it seems very conservative, a common value of 0.5 is taken for the momentum, energy, and turbulence k-ε model equations. The convergence criteria of 10-4 are used for all the governing equations. The maximum number of iterations is estimated so that the mass flow rate computed for the outlet cross section become unchanged, at about 2000 iterations the convergence can be observed, but the mass flow rate become unchanged since about 400 iterations for each case of simulation (fig. 3).

Boundary conditions

The meshed fluid volume has only one flow inlet (the compressor inlet) and one flow outlet (the volute exit). For the flow inlet, the pressure inlet boundary condition with atmospheric pressure (101325 pa) is taken as total pressure for the compressor entry, the total temperature is taken as the ambient (288.15K), the default values of inlet turbulent kinetic energy and its dissipation rate are taken respectively 1.21 m^2/s^2 and 1.82 m^2/s^3.

frame of reference, that are part of the rotating reference frame, are treated as moving walls with a rotational speed of absolute zero.

Grid specifications

The entire computational domain is meshed with tetrahedral grids due to the complexity of the compressor geometry, the clearance gap between the blades and the shroud is included, and the total number of grid is about 1067378 tetrahedral cells, the detail for each element of the compressor is shown in fig. 4; it contains a number of cells as shown in Table 2.

Table 2: Number of mesh cells for each compressor element, with different diffusers

Compressor elements		Impeller	Diffuser	Volute
Number of mesh cells	5 vanes	272360	53559	741459
	7 vanes	272360	53526	741954
	9 vanes	272360	189360	741459
	11 vanes	272360	185161	741459

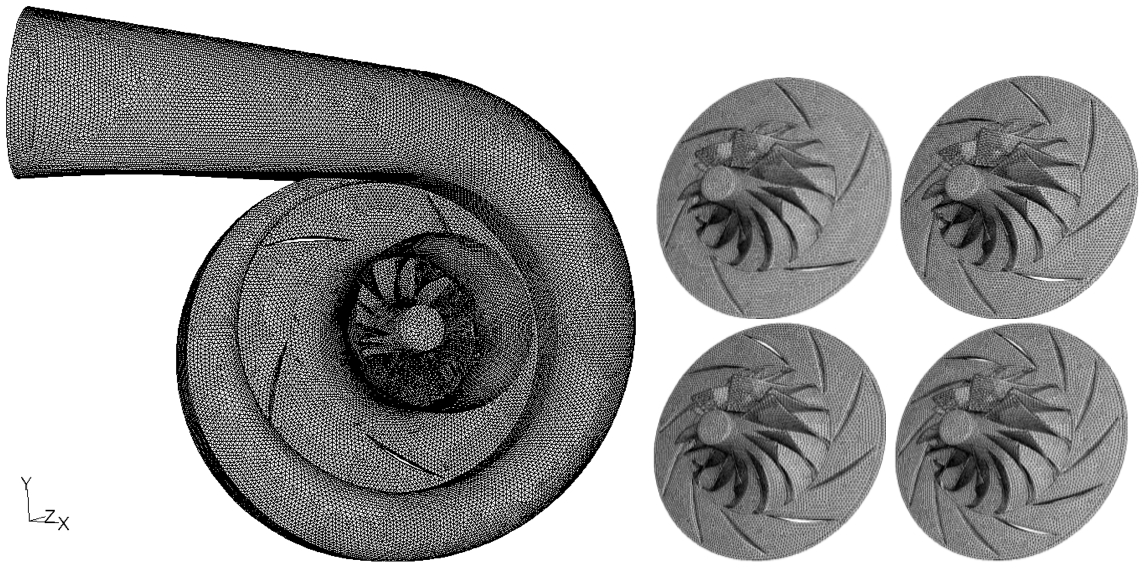

Figure 4. Computational mesh with different diffuser

At the volute exit, the pressure outlet boundary condition is used with a prescribed static pressure (this value is required by the engine operating conditions, it must be upper than the atmospheric pressure), the actual total temperature at the outlet is calculated or prescribed by the numerical simulation. At both the inlet and outlet, the default values of turbulent intensity and viscosity ratio (10 precent) are taken for the back-flow condition, which corresponds to the occurrence of reversed flow at same grid cell faces of the outlet cross-section. Since all the operating range no apparent reversed flow is observed when the simulation is converged. For the rotor, all reference frame moving walls are part of the rotating reference frame, These walls are treated as moving walls with rotational speed of zero relative to the adjacent cell zone, which rotating. No rotating walls in the inertia

4. RESULTS AND DISCUSSIONS

The turbulent flow of air through the turbocharger is studied, considering the full stage of the centrifugal compressor, consisting of an air inlet, leading to an impeller that discharges the air radial through a vaned diffuser in the volute. The objective of this study is the prediction of the influence of varying diffuser vanes number on the flow local characteristics velocity, pressure and temperature, for this, we consider four types of diffuser (5, 7, 9 and 11 vanes) that have the same size, they differ only in their vanes number.
All the simulations are carried out for the same rotational speed of 100000 rpm, and one point of the operating range is chosen for representing the results at flow rate of 0.47 kg/s. Analysing the obtained results, it is shown that the flow leaving the impeller has jets and wakes. When

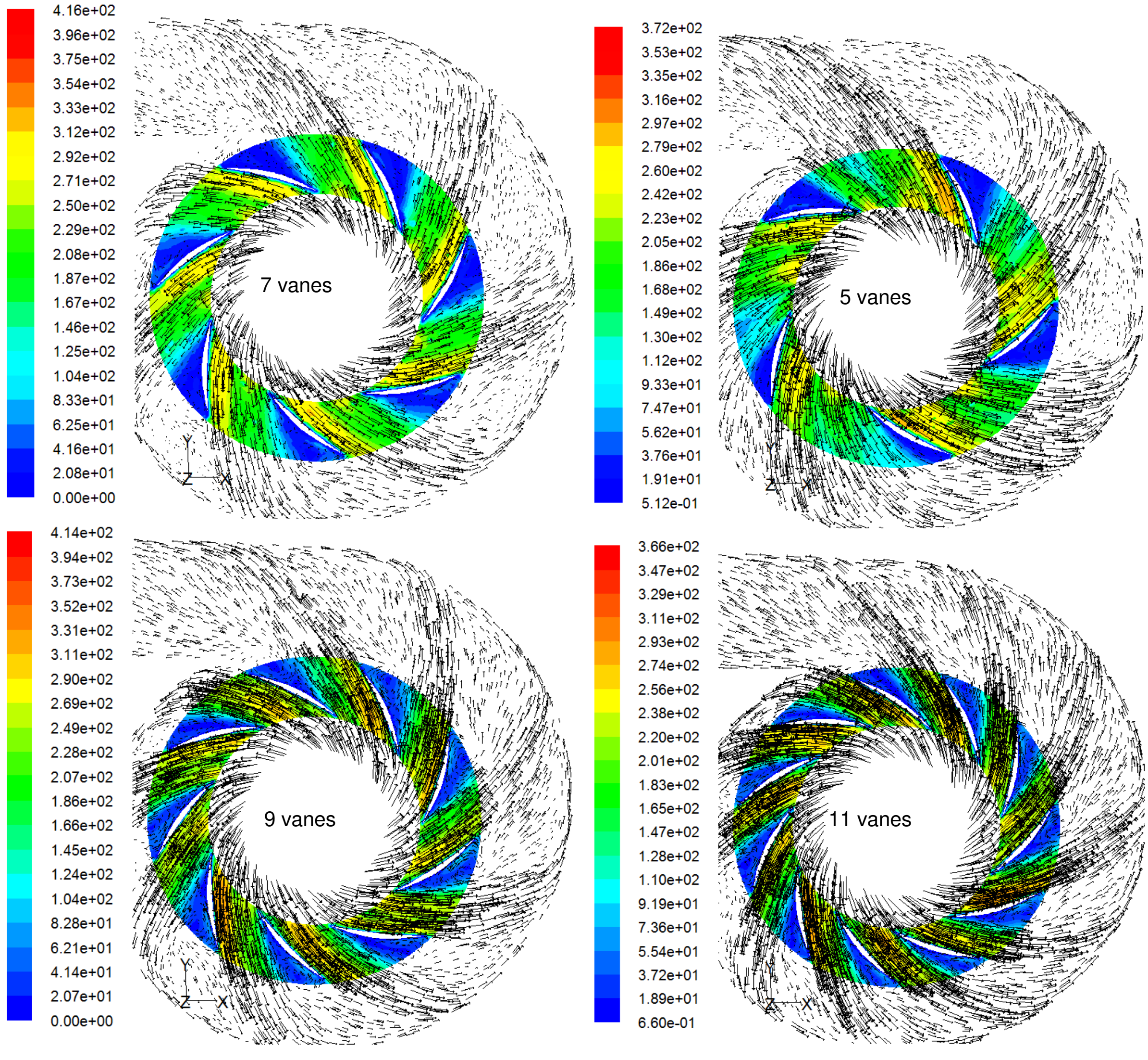

Figure. 5 . Velocity vectors and magnitude at different diffuser vanes number

such a flow enters a large number of diffuser passages, the quality of flow entering the diffuser vanes differs widely and some of the vanes may experience flow separation leading to rotating stall and poor performance. To avoid such a possibility, it is safer to provide a smaller number of diffuser vanes than that on the impeller [1]. When the number of vanes is reduced, the angle of diffusion for each vane passage becomes larger. Now, at larger diffusing angle of the passage the static pressure

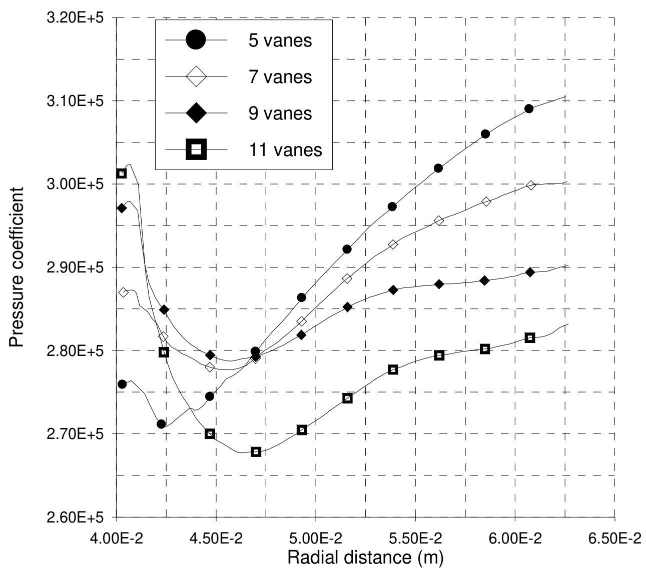

Figure. 6. Circumferential average radial of pressure coefficient

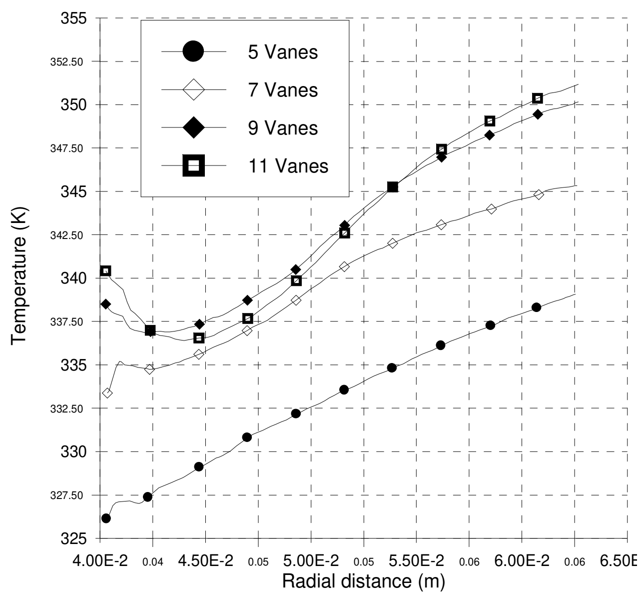

Figure 7. Circumferential average radial of temperature

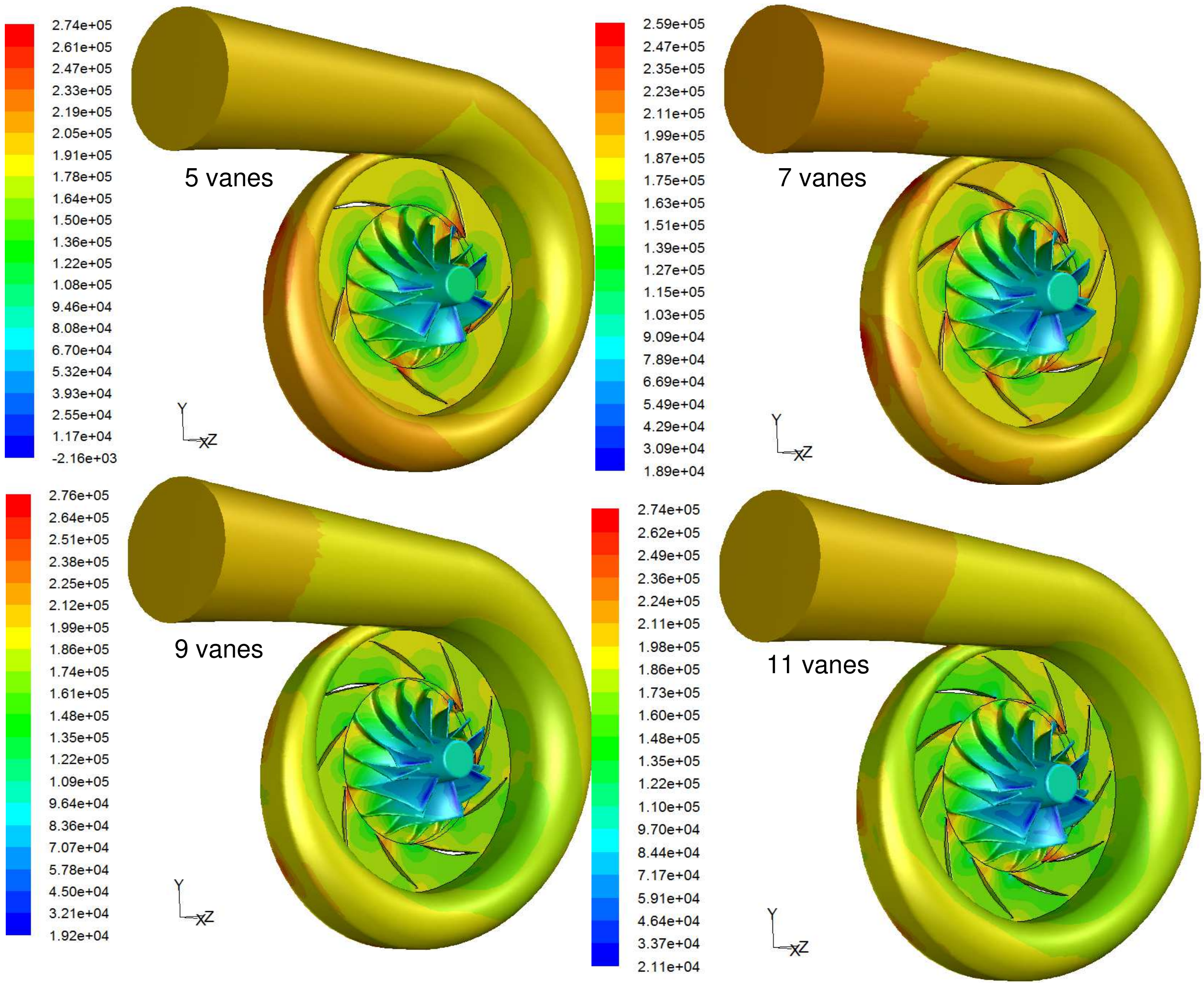

Figure 8 . Static pressure contour plots at different diffuser vanes number

conversion is more but attended by fluid flow losses due to poorer flow guidance. When the number of vanes is increased the angle of diffusion decreases and hence the static pressure conversion reduces but the flow losses tend to decrease due to better guidance but is offset partially by skin friction losses due to the larger contact surfaces of increased number of diffuser vanes. Hence there is a need to arrive at a trade off with respect to the number of diffuser vanes but diffusion process will be affected due to lower flow guidance within passage [6].

Fig. 5 present the velocity magnitude contours and velocity vectors fields in the middle plane. Because all the simulations presented in this paper are at the design speed of 100 000 rpm, the tangential velocities at the impeller exit are similar for all the simulations, and the radial velocity at the impeller exit is mainly dictated by the air flowrate. Showing vector plots of velocity, colored according to its magnitude. The recirculation zones on the impeller and diffuser suction sides are clearly visible for all the configurations. The influence of rotating stall comes into force for diffuser vanes with larger number of vanes. The reasons for this are two folds: Firstly the stalling phenomenon is associated with the blade passing frequency. With larger number of vanes on the diffuser, the blade passing frequency increases leading to rotating stall. Secondly this occurrence is due to narrower vane passages of the diffuser in which recirculation on the suction side of the diffuser vanes gets converted into a rotating stall [1].Similarly Figs. 6 and 7 show pressure coefficient and temperature evolution as circumferential average values along radial distance, it is clearly shown that the large pressure ratio and lower temperature are obtained for the less diffuser vanes number.

Fig .8 shows the static pressure contour plots. It is clearly seen that for diffuser with lower number of vanes (5 vanes), static pressure at the diffuser exit is large compared to diffuser with larger number of vanes (11 vanes).The reason for this could be attributed to fairly large angle of diffusion provided for diffuser with lower number of vanes. Since the number of diffuser vanes are less, the flow passage is wider and hence, the presence of recirculation flow disturbances do not affect the through flow in the diffuser vane passage, but diffusion process will be affected due to lower flow guidance within passage. Fig. 9 shows the static temperature contour plots. It is seen that for diffuser with lower number of vanes (5 vanes), static temperature at the diffuser exit is small compared to diffuser with larger number of vanes (11 vanes).This can gives a good chance for no using an intercooler at the turbocharger exit.

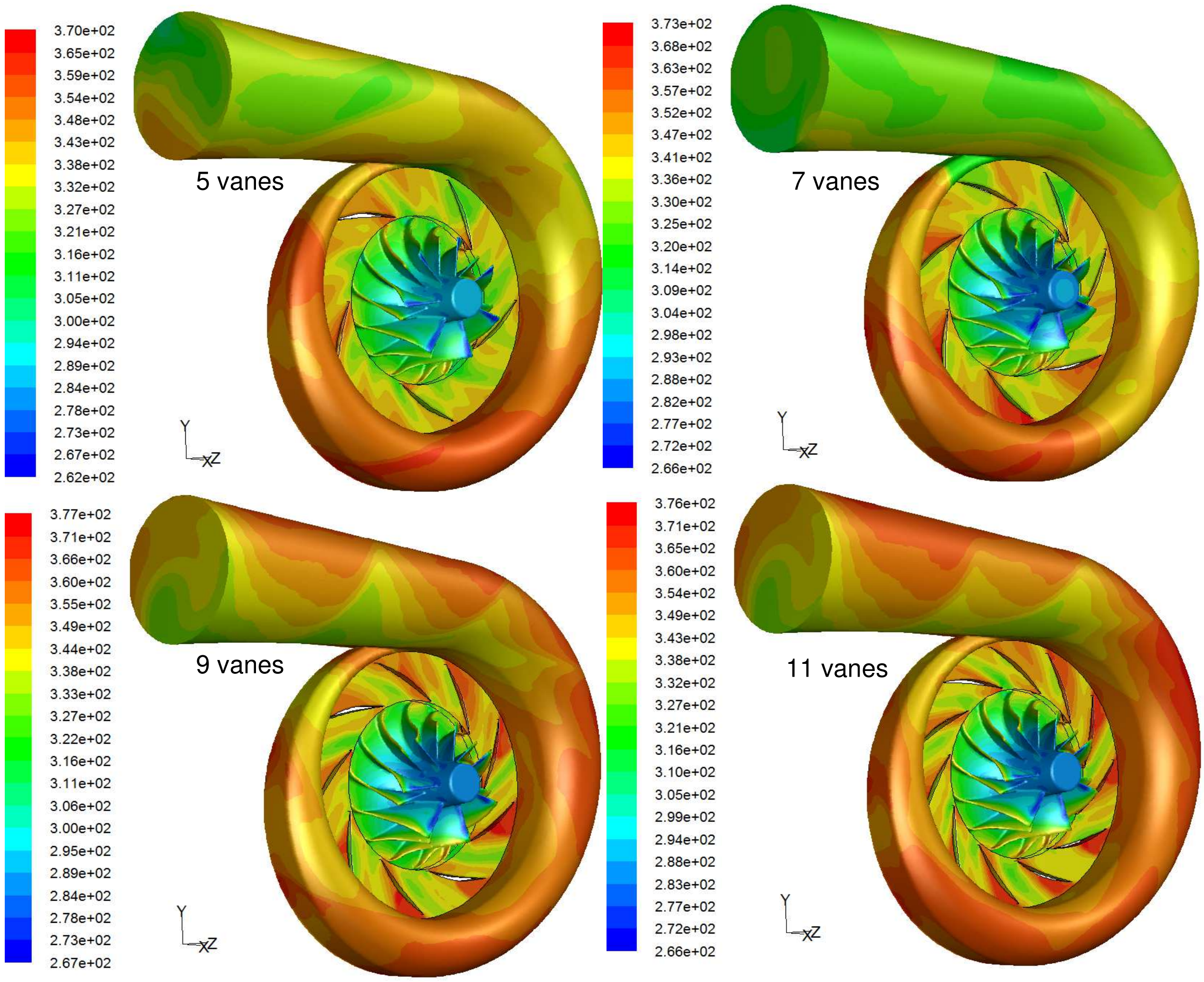

Figure 9 . Static temperature contour plots at different diffuser vanes number

5. CONCLUSIONS

From the analysis carried out in this paper, the following conclusions are derived:

1- The static pressure coefficient, which also takes into account the volute region of the compressor, decreases with increase in number of diffuser vanes. **2-** The rotating stall occurs in some of the alternate flow passages of the diffuser and this problem exists only when the number of diffuser vanes increase beyond a certain number. **3-** It is also contra indicating that static pressure reduction occurs in the volute casing when number of diffuser vanes is increased. **4-** There exist an optimum number of diffuser vanes which would develop maximum static pressure recovery.

REFERENCES

[1] S. Yahia *Turbines Compressors and Fan, 2nd edn. Tata Mc Graw Hill,2005*

[2] Chehhat Abdelmadjid, Si-Ameur Mohamed and Boumeddane Boussad, (2013), CFD Analysis of the Volute Geometry Effect on the Turbulent Air Flow through the Turbocharger Compressor, Energy Procedia 36, 746-755

[3] H. Mohtar, P. Chesse, D. Chalet, J.-F. Hetet and A.Yammine, (2011) Effect of Diffuser and Volute on Turbocharger Centrifugal Compressor Stability and Performance:Experimental Study, Oil & Gas Science and Technology – Rev. IFP Energies nouvelles, Vol. 66, No. 5, pp. 779-790

[4] Kui Jiao, Harold Sun, Xianguo Li, Hao Wu, Eric Krivitzky, Tim Schram, Louis M. Larosiliere, Numerical Simulation of air flow through turbocharger compressors with dual volute design , *Applied Energy, vol.* 86, pp. 2494-2506, 2009

[5] K. Jiao, H Sun, X Li, H Wu, E Krivitzky, T Schram, and L M Larosiliere, Numerical investigation of the influence of variable diffuser angle on the performance of a centrifugal compressor, Proc IMechE Part D: J. *Automobile Engineering* V.223 p.1061-1070, 2009

[6] K. Vasudeva Karanth , N. Yagnesh Sharma, Numerical analysis on the effect of varying number of diffuser vanes on impeller - diffuser flow interaction in a centrifugal fan, *World Journal of Modelling and Simulation* ,Vol. 5, pp. 63-71, 2009

[7] N Bulot, J Trébinjac, X Ottavy, P Kulisa, G Halter, B Paolitti, and P Krikorian, Experimental and numerical investigation of flow field in a high-pressure centrifugal compressor impeller near surge,

Proc IMechE V.223 Part A: J. *Automobile Engineering* p. 657- 666, 2009

[8] Ali Pinarbasi, Turbulence measurement in the inlet plane of a centrifugal compressor vaneless diffuser, *Ijhff*, 30, 266-275, 2009

[9] R. Aghaei tog, A. M. Toussi, A. Tourani, Comparison of turbulence methods in CFD analysis of compressible flows in radial turbo machines, *An Aircraft Engineering and Aerospace Technology: an International journal* 80/06, 657-665, 2008

[10] S Anish and N Sitaram, Computational investigation pf impeller-diffuser interaction in a centrifugal compressor with different types of diffusers, Proc IMechE Vol.223 Part A: J. *Automobile Engineering* 167- 178, 2008

[11] Reza Aghaei tog, A. Mesgharpoor Toussi, M. Soltani, Design and CFD analysis of centrifugal compressor for a microgasturbine, *An Aircraft Engineering and Aerospace Technology: an International journal* 80/06, 137-143, 2008