Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 36 (2013) 746 – 755
Energy Procedia

TerraGreen 13 International Conference 2013 - Advancements in Renewable Energy and Clean Environment

CFD Analysis of the Volute Geometry Effect on the Turbulent Air Flow through the Turbocharger Compressor

Chehhat Abdelmadjid[a]*, Si-Ameur Mohamed[b], Boumeddane Boussad[c]

[a] *Science and technology department, Abbes Laghrour Universty,of Khenchela 40000, Algeria*
[b] *LESEI Laboratory, University of Batna, Batna 05000, Algeria*
[c] *Mechanical engeneering department,University of Blida, Blida 09000, Algeria*

Abstract

In this work a numerical solution with moving mesh technique is made. Many research works both experimental and numerical on the diffuser volute 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 geometry of volute has not been the focus of attention in these works. Hence a numerical analysis has been carried out in this work to extensively explore impeller-diffuser-volute fluid interaction as well as to predict the flow and turbulence characteristics of the centrifugal compressor by varying the volute geometry without changing the number of impeller blades. It is found from the analysis that volute geometry presents a considerable effect on the pressure and temperature at the compressor outlet.

Keywords: centrifugal compressor; turbocharger; volute; CFD analysis

1. Introduction

With the development of environmental protection in many countries, energy-saving emissions have become the goal of engine industry. The turbocharger has an irreplaceable role in improving engine power, reducing fuel consumption and decreasing emissions. However, because of complex geometry of the turbocharger volute, it cost a lot of time and labor power to adopt experiment to gain relevant data of volute. So it becomes more economical to simulate turbocharger internal flow field and analysis data using the CFD software.

* Corresponding author. Tel.: 213664613872 ; fax: +0-000-000-0000 .
E-mail address: achehhat@gmail.com

doi:10.1016/j.egypro.2013.07.087

Nomenclature	
P	pressure (Pa)
T	temperature (K)
Q	mass flowrate (kg/s)
Greek Symbols	
η_c	efficiency
π_c	pressure ratio (outlet/inlet)
π_m	pressure ratio (inlet/reference value)
τ_c	temperature ratio (outlet/inlet)
τ_m	temperature ratio (inlet/reference value)

Through looking up the literature of different turbocharger, the volute has been the subject of numerous numerical and experimental studies, the reason for this; is that the volute strongly affects the overall performance, stability, operating range and the location of the best efficiency point of the compressor. In addition, the radial force on the impeller is a result of the pressure non-uniformity caused by the volute. The numerical study of H. Mohtar et al [1] showed that the modification of the volute tongue location affects the compressor efficiency and pressure lines at high speed. The asymmetric influence of the volute on the flow in a transonic, high-pressure ratio centrifugal compressor at off-design conditions was investigated by X. Q. Zheng et al.[2] when the inter-passage variations in performance quantities and the influence of the volute tongue region are discussed in detail. The circumferential variations of incidence angle correlate with rotational speed, which, in combination with the higher sensitivity to incidence angle at transonic inflow conditions, seems to deteriorate stability when transonic inflow conditions are reached. A. S. Hassan [3] investigated theoretically and experimentally the effect of the volute design parameters on the centrifugal compressor range of stable operation and pressure rise coefficient, especially the theory is devoted to the effect of the area ratio on the stability, while the experience is made for understanding the effect of the gap between the diffuser and the volute casing. The study of the flow structure for different volute tongue geometries has been made by Cheng Xu et al. [4] and confirmed that the design of the volute tongue impacts the compressor operating range. The numerical and experimental analysis of different volutes, elaborated by A. Reunanen [5] showed that the change of the cross section, and the location of the volute inlet affect not only the compressor performance but even the non-uniformity of pressure and force related to it at high flow rates. H. Rezaei [6] has investigated the flow structure and loss mechanism in a centrifugal compressor volute, when the results of experience show that the non-uniform circumferential static pressure distribution was observed in all cases, and revealed that this is due to the effect of the tongue on the compressor performance. Coming in the same context, this paper will further investigate influence of three types of volute with the same vaneless diffuser and impeller they differ in their cross section geometry (circular or semi-circular), and their inlet location (symmetric or tangential). For this purpose, a CFD software FLUENT 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. The flow characteristics and performance level will be predicted.

2. The compressor geometry modeling

The compressor geometry development is carried out on GAMBIT software, the various components of the compressor are designed individually and all are assembled fig. 2. The blade profiles are generated with a help of coordinates that have been introduced,. The diffuser is modeled as a hollow disc, without guide vanes, the dimensions of the impeller and the diffuser are shown in the table 1. The volute is modeled by assembling a casing of various cross sections beginning from 0° and ending at 360° when a tongue is located and a conic pipe discharging in engine admission manifold. The mesh generated for each element of the compressor is shown in Fig.3.

Table 1. Geometrical dimensions of the designed compressor

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°

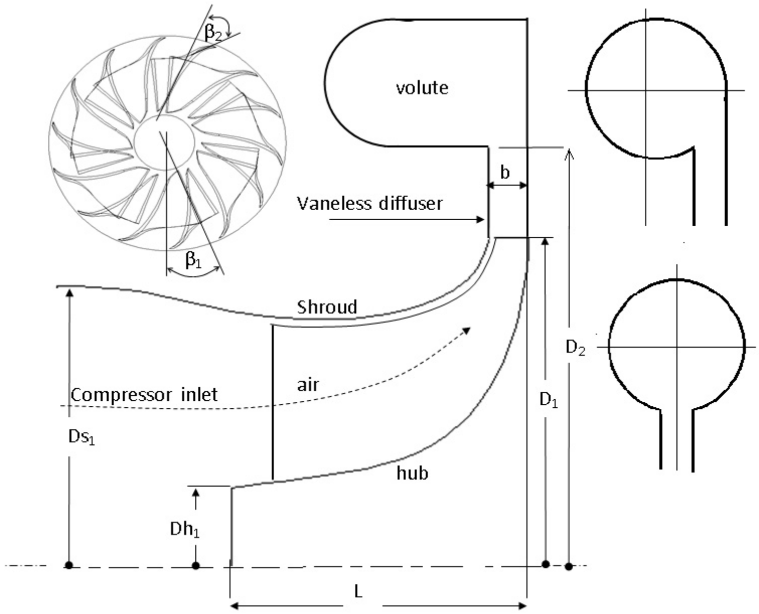

Fig. 1: Schematic of the centrifugal compressor

3. Numerical process

The high complexity of the flow in the centrifugal compressor volute makes the CFD modeling very difficult, only steady state flow is investigated, the governing equations implemented in a commercial CFD code Fluent 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. 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 1500 iterations for each case of simulation.

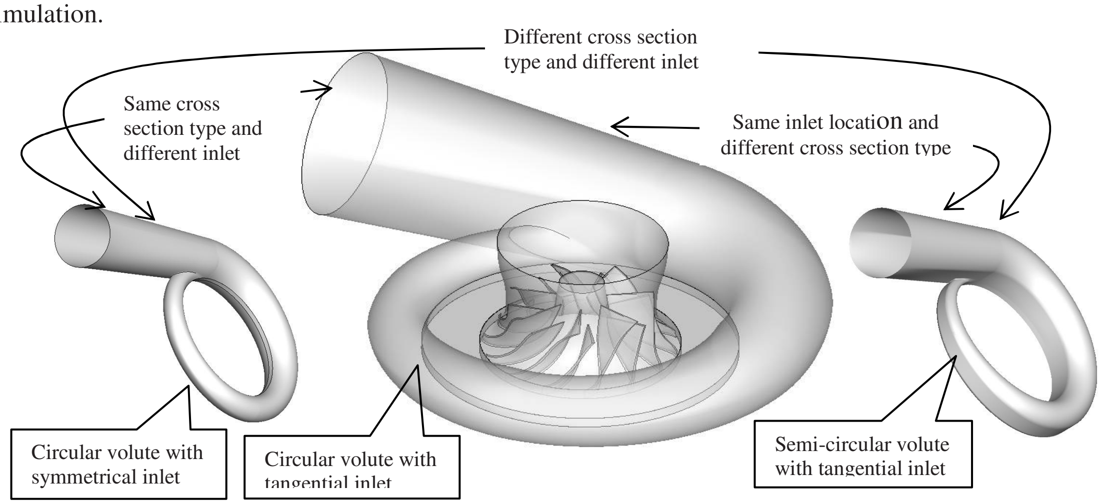

Fig. 2: The centrifugal compressor geometrical model with different types of volutes

4. Results and discussions

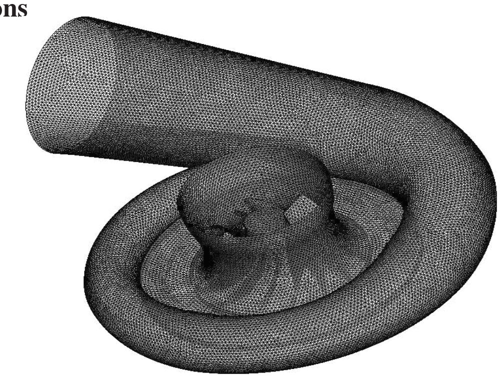

Fig. 3: Computational grid of the compressor full stage

Three different designs with the same impeller and diffuser but with different volutes (circular cross section with tangential inlet location, circular cross section with symmetrical inlet location, and semi-circular cross section with tangential inlet location) are simulated at 100 000 r/min, which is the design speed of the compressor. The widths of the stable operating range of the different designs are obtained, and the performances in these operating ranges are obtained as well.

4.1 Comparison of overall performance

The pressure ratios and compressor efficiencies at 100 000 r/min for different designs are presented in Figs 4 and 5. The pressure ratio π_c and isentropic efficiency η_c are calculated on the basis of the total (stagnation) pressures P_{total} (Pa) and total (stagnation) temperatures T_{total} (K) at both the inlet and the outlet of the compressor, as shown from the relations [8]:

$$\pi_c = \frac{P_{total,outlet}}{P_{total,inlet}} \tag{1}$$

$$\eta_c = \frac{(\pi_c)^{0,285} - 1}{\tau_c - 1} \tag{2}$$

$$\tau_c = \frac{T_{total,outlet}}{T_{total,inlet}} \tag{3}$$

The mass flowrate of air is corrected using the ambient conditions of temperature and pressure (288.15K and 101325 Pa).

$$Q_{corrected} = \frac{\sqrt{\tau_m}}{\pi_m} Q_{real} \tag{4}$$

$$\tau_m = \frac{T_{total,inlet}}{288,15} \quad , \quad \pi_m = \frac{P_{total,inlet}}{101325} \tag{5}$$

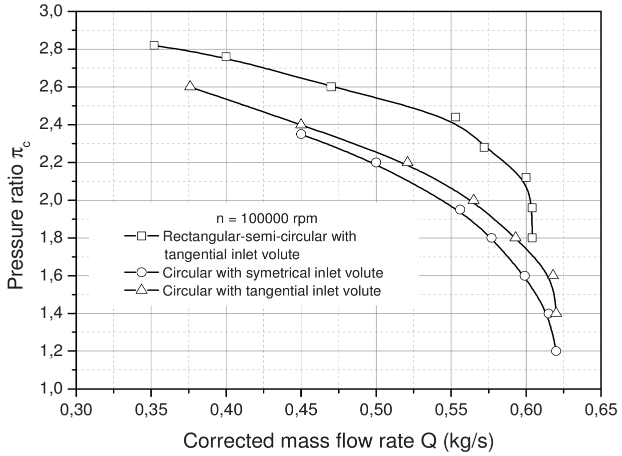

Fig. 4 : Comparaison of pressure ratios

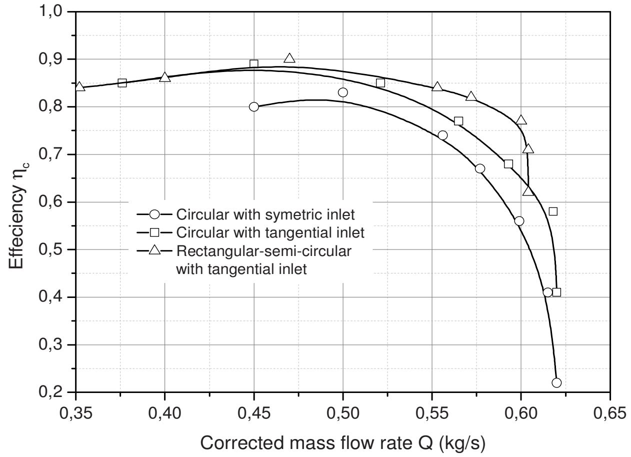

Fig. 5 : Comparaison of efficiences

Where $Q_{corrected}$ (kg/s) and Q_{real} (kg/s) are the corrected and real air mass flowrates respectively at the impeller inlet. $T_{total,inlet}$ (K) is the total (stagnation) temperature at the impeller inlet, and $P_{total,inlet}$ (Pa) is the total (stagnation) pressure at the impeller inlet. It should be noted that the total temperature and pressure are the same as the reference values (288.15K and 101 325 Pa); therefore the real and corrected air mass flowrates are identical for the simulations presented in this paper.

From the comparison between different designs, it can be observed that the peak efficiency is achieved at an air flowrate of about 0.48 kg/s for the compressor with the circular volute with symmetrical inlet location, and about 0.45 kg/s with both the circular and semi-circular volutes with tangential inlet location. This range of mass flowrates also matches those expected at the engine inlet. The peak efficiency reaches the maximum (85 per cent) for the compressor at with the semi-circular volute with tangential inlet location. The operating range for the compressor with the semi-circular volute with tangential inlet location is wider in comparison with the two other volutes.

4.2 Flow characteristics through the full stage of compressor

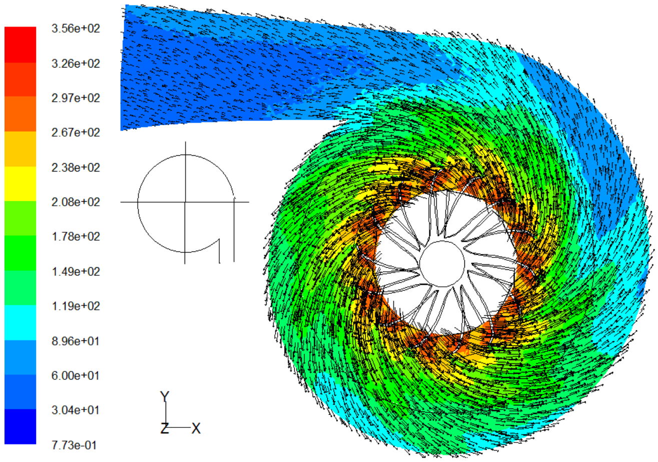

Fig. 7: Velocity magnitude contours and velocity vectors in middle plane, for the circular volute with tangential inlet

In order to understand the results shown, in the following sections, the flow characteristics for different designs are analyzed.

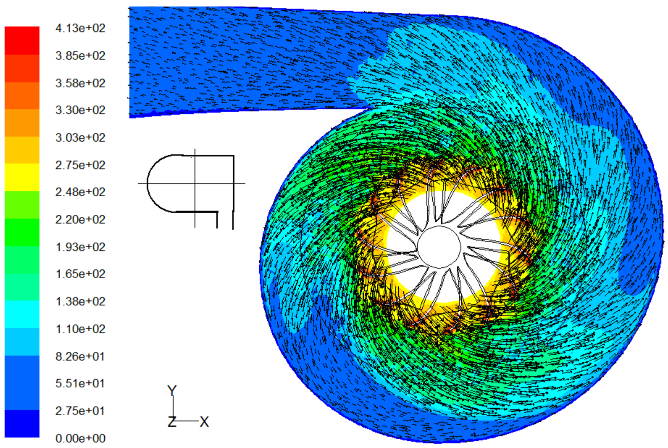

Fig. 6: Velocity magnitude contours and velocity vectors in middle plane, for the semi-circular volute

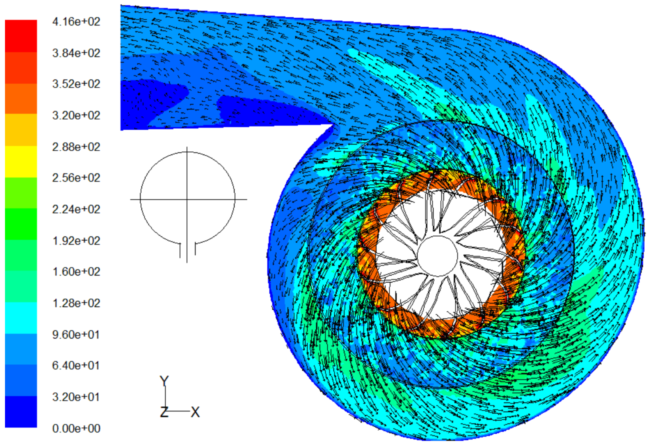

Fig. 8: Velocity magnitude contours and velocity vectors in middle plane, for the circular volute with symmetrical inlet

Figures 6, 7and 8 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.

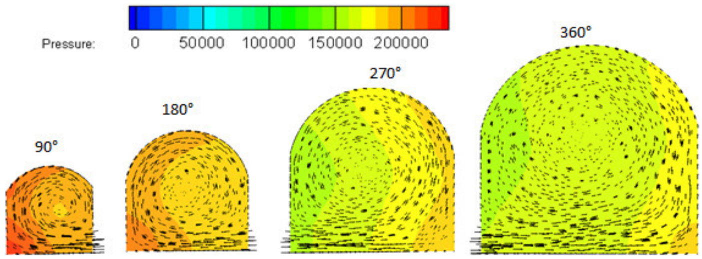

Fig. 9: Static pressure contours and meridional velocity vectors for semi-circular volute with tangential inlet

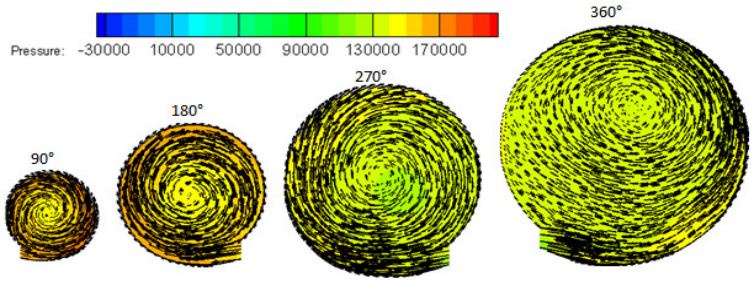

Fig. 10: Static pressure contours and meridional velocity vectors for circular volute with tangential inlet

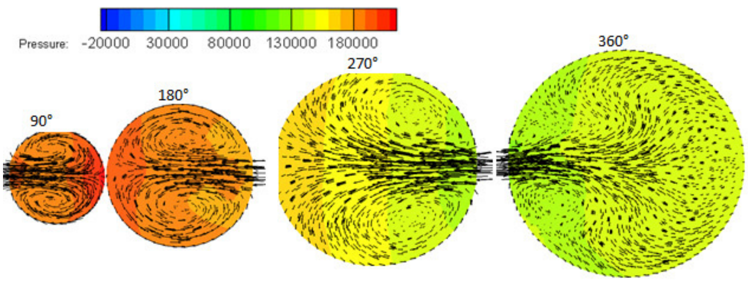

Fig. 11: Static pressure contours and meridional velocity vectors for circular volute with symmetrical

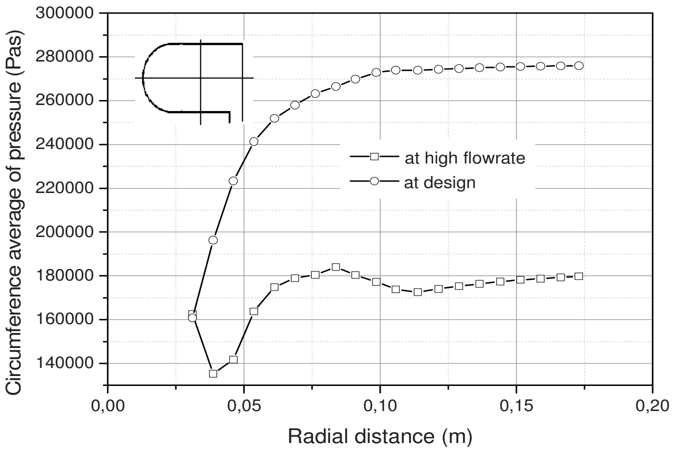

Fig. 12: Circumference average of pressure, at design and at high flowrate for semi- circular volute with tangential inlet

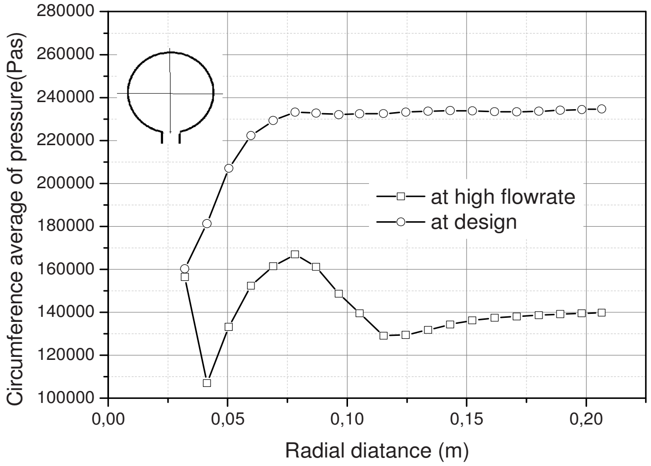

Fig. 13: Circumference average of pressure, at design and at high flowrate for circular volute with symmetrical inlet

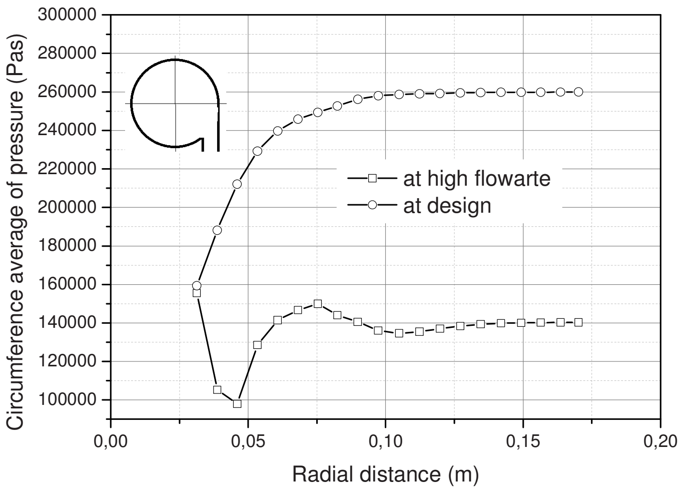

Fig. 14: Circumference average of pressure, at design and at high flowrate for circular volute with tangential inlet

Therefore, at a higher air flowrate, the radial velocity is higher, as well as the velocity magnitude. With similar tangential velocities at the diffuser inlet, tangential flow is more apparent when the radial velocity is power, and higher radial velocity results in stronger radial flow.
The radial component of the velocity at the vaneless diffuser outlet, generates a strong rotational flow in smaller areas of the volute, which in the form of forced vortex flow. Further downstream, fresh fluid warps around this rotational flow and as the volute area increases a counter vortex is generated.
Showing the tangential velocity, it can be observed that the flow goes through acceleration and deceleration in the smaller cross section and minimum mass flowrate, diffusion decelerates the flow inside the volute and the tangential velocity decreases. This decrease is more remarked when the flow arrives inside the conic diffuser because of the larger diffusion rate in this region.
In figures: 9, 10 and 11 velocity vectors in meridian planes are shown on the static pressure contours for cross section angles of 90°, 180°, 270° and 360°. The rotational velocity flow increases from zero at the centre to a higher value through the radial out. The velocity distribution at the vortex core is affected by the geometry of the volute cross section. For the volutes with tangential inlet location, a single vortex is observed, however for the volute with symmetrical inlet location a twin vortices are generated mainly in the small cross sections, while in the large cross sections, the diffusion is large as well.

Observing the static pressure contours, achieved at high flowrate , the region of high pressure is appear on the small cross section, this is due to the non-uniformity caused by the volute. This phenomenon vanish at design flowrate, this is observed for all cases (figures: 12, 13 and 14) when the uniform pressure is observed at design in the volute.

Note that the single vortex and twin vortices have moved axially further from the wall in the lowest flowrate. It can be observed that the strength of these vortices decreased downstream and the centre of the rotational flow is carried away from the hub to the shroud wall. However, it is not exactly predicting the location of these vortices.

5. Conclusions

In this study, three volutes (circular cross section with tangential inlet location, circular cross section with symmetrical inlet location, and semi-circular cross section with tangential inlet location) with the same impeller were investigated using the computational fluid dynamics. The high complexity of the flow in the centrifugal compressor volute makes the CFD modelling very difficult, only steady state flow is investigated, the choose of the volute geometry is made on the base that two geometrical parameters, which affect the overall performance and operating range, are studied at the same time , and which are:

- **the shape of cross section of the volute (comparing the circular and semi-circular cross section)**
- **the location of the volute inlet (comparing the tangential inlet and the symmetrical inlet)**

From the comparison between different designs, the following conclusion can be achieved:

1- The modification of the volute cross section shape, affects the operating range more than the peak efficiency.
2- The modification of the volute inlet location impacts the peak efficiency more than the operating range.
3- Higher peak efficiency, is observed in the case of tangential inlet.
4- Wider operating range is observed in the case of modified circular case.
5- Non-uniform pressure is observed at high flowrates.

Future work of the three configurations should be focused on the effect of the volute throat and tongue position on the whole compressor performance. The computational model also needs to be improved to reduce the error at high mass flowrate in order to get a better whole compressor performance prediction. In addition, the unsteady problem should be investigated.

References

[1] H. Mohtar, P. Chesse, D. Chalet, J.-F. Hetet and A.Yammine, 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 (2011), No. 5, pp. 779-790

[2] X Q Zheng, J Huenteler,MYYang,Y J Zhang, and T Bamba, Influence of the volute on the flow in a centrifugalcompressor of a high-pressure ratio turbocharger, Proc. IMechE Vol. 224 Part A: J. Power and Energy, pp. 1157-1169

[3] A. S. Hassan, Influence of the volute design parameters on the performance of a centrifugal compressor of an aircraft turbocharger, Proc. IMechE Vol. 221 Part A: J. Power and Energy, (2007), pp. 695-704

[4] Cheng Xu, Michael Muller, Development and design of a centrifugal compressor volute, International Journal of rotating machinery , 3, (2005) , pp.190-196

[5] Arttu Reunanen, experimental and numerical analysis of different volutes in a centrifugal compressor, thesis of the degree of doctor science,(2001) , lappeenranta university of technology

[6] Rezaei, Hooman, Investigation of the Flow Structure and Loss Mechanism in A Centrifugal Compressor Volute. ,Dissertation Abstracts International, (2001)Volume: 62-03, Section: B, page: 1544

[7] 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, 86, (2009), pp. 2494-2506

[8] 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 Vol.223 Part D: J. Automobile Engineering ,(2009), pp.1061-1070

[9] 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 Vol.223 Part A: J. Automobile Engineering, (2009),pp. 657- 666

[10] Ali Pinarbasi, Turbulence measurement in the inlet plane of a centrifugal compressor vaneless diffuser, Ijhff ,30, (2009), pp. 266-275

[11] Y Dai, A Engeda, M Cave, and J-L Di Liberti, Numerical study and experimental validation of the performance of two different volutes with the same compressor impeller, Proc IMechE Vol.223 Part A: J. Automobile Engineering, (2008), pp.157- 166

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

[13] 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, (2008), pp.167- 178

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

[15] J. Galindo, J. R. Serrano, H. Climent, A. Tiseira, Experiments and modelling of surge in small centrifugal compressor for automotive engines, etfs,32, (2008), pp. 818-826

[16] Q Guo, H Chen, X-C Zhu, Z-H Du, and Y ZhaoNumerical simulations of stall inside a centrifugal compressor, Proc IMechE Vol.221 Part A: J. Automobile Engineering, (2007), pp. 683- 693

[17] Youn-Jea KIM and Dong-Won KIMAnalysis on the performance and Interaction between The Impeller and Casing in small-Size Turbo-Compressor, JSME International journal, serie B, 3, vol. 46, , (2003), pp. 385-391

[18] Fluent 6.3 user's guide, 2006 (Fluent Inc., Lebanon, New Hampshire)

[19] GAMBIT 2.2 user's guide, 2004 (Fluent Inc., Lebanon, New Hampshire)