NUMERICAL STUDY ON THE EFFECT OF INTERACTION VANED DIFFUSER WITH IMPELLER ON THE PERFORMANCE OF A MODIFIED CENTRIFUGAL COMPRESSOR

L. H. Jawad *

Foundation of Technical Education
Ministry of Higher Education and Scientific Research
Iraq

Department of Mechanical & Materials Engineering
Faculty of Engineering & Built Environment
The National University of Malaysia, Malaysia
Ukm Bangi, Selangor, Malaysia

S. Abdullah **R. Zulkifli** **W. M. F. W. Mahmood**

Department of Mechanical & Materials Engineering
Faculty of Engineering & Built Environment
The National University of Malaysia, Malaysia
Ukm Bangi, Selangor, Malaysia

ABSTRACT

This paper is a numerical simulation that was made in the three-dimensional flow, carried out in a modified centrifugal compressor, having vaned diffuser stage, used as an auto-motive turbo charger. Moreover, the performance of the centrifugal compressor was dependent on the proper matching between compressor impeller and vaned diffuser, influencing significantly surge and the efficiency of centrifugal compressor stages. In addition, a modified compressor impeller, coupled with vane and vaneless diffuser, has been found to have similar internal flow patterns for both the vaneless and vaned diffuser design. The vaned diffuser effect has been paid particular attention in terms of better analysis where the diffuser was designed for high sub-sonic inlet conditions. Another aim of this research was to study and simulate the effect of vaned diffuser on the performance of a centrifugal compressor. The simulation was undertaken by using a commercial software, the so-called ANSYS CFX, to predict numerically the performance in terms of pressure ratio, poly tropic efficiency and mass flow rate for the centrifugal compressor stage. The results were generated from CFD and were analyzed for better understanding of the fluid flow through centrifugal compressor stage. Conclusively, it was observed that the effect of the vaned diffuser is to convert the kinetic energy into a high static pressure after analyzing the results of the simulation.

Keywords: Numerical study, Centrifugal compressor performance, Vaned diffuser, CFD.

1. INTRODUCTION

In these days, interest has more progressively been devoted to the development of turbochargers because of their compact size, large capacity, high performance, and ability to improve volumetric efficiency. Turbochargers are broadly used in many applications, such as marine diesel engines, automobile engines, and small gas turbines for aircraft engines. The improvement of turbocharger compressor performance and the extension of the stable operating ranges are becoming critical for the viable future of low emission engines. In centrifugal compressor case, it is known that unsteady behaviour becomes apparent when the air mass flow through the compressor is lower than the critical level. This unstable phenomenon is denoted as a surge and corresponds to a backflow of compressed fluid through the compressor into its inlet. Generally, the performance of a centrifugal compressor is expressed as a relationship between the mass flow rate and the pressure ratio of a line with a constant number of revolutions.

Furthermore, the influences of the different diffuser meridian channel width ratios on the compressor performance under design conditions show a remarkable significance in terms of improving the efficiency of the whole machine in a micro gas turbine centrifugal compressor [1]. The effect of pulsating flow inside a centrifugal compressor and the corresponding pressure pulses on the compressor surge line can be very important because the pulsating flow is in the range of 40 ~ 67Hz (corresponding to characteristic pulsation when boosting an internal combustion engine) which increases

* Corresponding author (Laihasan@yahoo.com)

DOI : 10.1017/jmech.2013.49
Copyright © 2014 The Society of Theoretical and Applied Mechanics, R.O.C.
https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

the surge margin [2]. The application of CFD to turbocharger compressor characteristic predictions over a range of speed, to develop an efficient methodology for analysing the turbocharger compressor performance, and to compare the computation versus rig measurements [3]. In addition, the stall flow phenomenon inside a turbocharger centrifugal compressor with a vaneless diffuser simulated numerical and the amplitude of the static pressure oscillation at this frequency in the diffuser is increased with the reduction in compressor mass flow, the results show that there is a distinct stall frequency at the given compressor speed [4]. An analytical model for the centrifugal compressor was proposed to predict the compressor performance such as outlet pressure, efficiency and losses. The model provides a valuable tool for evaluating the system performance as a function of various operating parameters [6]. The compressor performance map is described experimentally for characterization of the automotive turbocharger, and a mathematical tool has been developed for marking out surge operation points from stable compressor points [5]. The contribution to the design methodology and performance assessment of low solidity vaned diffusers to understand the pressure recovery phenomena in each of the three types of diffusers, and the effect of design parameters on performance was studied by [7]. The effect of impeller exit width trimming was studied and discussed along with the effect on overall performance on the basis of experimental data for two impellers. One with a low-flow coefficient and the other with a high-flow coefficient, blade loading and impeller diffusion was examined by [8].

The effect of the piping systems on the surge characteristics to the design of the compressor was studied and tested by several centrifugal compressors for turbochargers combined with the different piping systems, including investigating the changes of surge characteristics, surge lines, which connect surge points on the performance map by [9]. Stable working conditions and surge phenomena were simulated and boundary was used as the Method of Characteristics to determine the flow conditions at compressor inlet and outlet. The increasing of power output and reducing fuel consumption, by downsize the engine displacement [10,26]. The complex shock waves within the diffuser throat and impeller inlet, respectively, within high-speed compressors. This flow phenomena does not occur in low speed compressors and are very significant in the design of these compressors [11,12]. Many researchers have indicated that suitable treatments can extend the stable operating range of a turbocharger centrifugal compressor, but the performance is still insufficient under the majority of conditions.

The aerodynamic performance of centrifugal compressors is bound by surge and chokes on a compressor performance map. Much effort has been spent to define performance maps [13,23-25], to determine the performance of a new high speed compressor. The barrier imposed by the surge line, which separates the regions of stable and unstable operation, is of particular interest due to its close proximity to the maximum efficiency operating point. The initiation of unstable operation has been studied by many researchers [14]. When the mass flow through the compressor is below the surge line, unstable operation occurs in the form of rotating stall or surge.

One of the important aspects of designing efficient compressors is properly matching the impeller and diffuser. The importance of this topic was presented by [17], who showed that the presence of diffuser vanes considerably increases the pressure at the exit of the impeller, proving that there is a coupling between the impeller and diffuser. The diffuser used in the compressor must be designed for the mass flow rate, flow velocities, and flow angles at the exit of the impeller. This can be very challenging due to the highly unsteady nature of compressors and the jet-wake flow at the exit of the impeller. The performance of the impeller and diffuser is highly coupled, as shown by [18]. The potential field generated by the diffuser acting on the impeller exit flow field is not only driven by geometry but is also dependent on the unsteady diffuser loading. The potential field is the effect the geometry that it has on the pressure in the airflow. This diffuser loading is in turn a function of the rotating impeller potential field and the highly three-dimensional velocity field produced by the impeller. The diffuser potential field acting on the impeller has been studied using CFD by [19]. Within the impeller passage, there are high and low momentum regions known as the jet and wake, which have been studied by [20]. These zones not only vary along the circumference of the impeller, but also along the span, from the hub to the shroud. As flow emerges from the impeller, the blade forces on the fluid are lost, and the jet and wake regions undergo rapid mixing within the vaneless space. The design of the vaneless space has a significant impact on the overall performance of the compressor, including operating range, stability limits, viscous loss, and diffuser separations. Some of the earlier models based on low-speed compressors showed complete mixing within the vaneless space. Investigations on high-speed machines, such as in [21], have shown that the mixing process is not sufficient to produce a uniform flow field at the leading edge of the diffuser. The flow at the inlet of the diffuser is made up of shocks, partial separations, varying momentum and incidence angles. The interaction between the diffuser and impeller makes it difficult and impossible to predict the behavior of these flow irregularities and to study one isolated from the other respectively. The degree of these impeller-diffuser interactions is highly dependent on the radial gap between the impeller exit and diffuser inlet along with the diffuser vane geometry. Due to this dependence, it is important for the designer to understand the effects of these geometries in order to properly match these components. The angle of the diffuser vanes is also an important aspect of the performance due to the varying incident angles occurring at the diffuser inlet because of the unsteady flow at the exit of the impeller. There are several studies that have been conducted on the effects of radial gap and vane angle such as the work of [22]. In the light of the facts given above, the work reported in this paper deals with numerical inves-

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

tigations on interaction of vaned diffuser with modified centrifugal compressor impeller where CFD simulations and flow behavior of each conventional and modified centrifugal compressor impeller with vaned diffuser configurations were performed. To study the effect of the vaned diffuser to achieve high quality flow and further performance improvement of the turbocharger compressor by boosting the outlet pressure and mass flow rates.

2. DESIGN & SPECIFICATIONS

The compressor studied was a centrifugal compressor with vaned diffuser stage model. The inflow and the outflow of the fluid zone were as shown in Fig. 1. The main geometry features and dimensions of a conventional compressor and the modified compressor vaned diffuser are given in Table 1.

Figure 2 shows a conventional centrifugal compressor wheel comprising of 6 main impeller blades and 6 splitter blades, and Fig. 3 shows the geometry of the modified compressor wheel vaned diffuser stage comprising of 6 main impeller blades, 12 splitter blades and 24 diffuser vanes. The CFD computations for the conventional and modified designs were performed on the geometries. All the surface geometry, inlet, exit, and periodic boundaries, were defined via computer-aided design (CAD) as Initial Graphics Exchange Specification (IGES) parts.

Table 1 Geometric Features of the Vaned Diffuser Centrifugal Compressor

Centrifugal Compressor	(Conventional) Single Splitter	Modified (Double Splitter)
Axial Width of Impeller in Meridional View	39.25mm	30mm
Inner Radius at Compressor Inlet	12mm	8mm
Outer Radius at Compressor Inlet	36.25mm	35mm
Impeller Outer Radius	49mm	49mm
Impeller Width at Trailing Edge	6mm	6mm
Number of Blades	12	18
Number of Splitters	6	12
Tip Clearance at Shroud	9% Span	9% Span
Splitter1 angular offset (pitch fraction or location)	50%	67%
Splitter2 angular offset (pitch fraction or location)		33%
Number of Diffuser Vanes		24
Vane Inner Radius		61mm
Vane Outer Radius		82mm
Vane Inner Axial Width		6mm
Vane Outer Axial Width		4mm
Vane Outer Flow angle		30

3. CFD METHODOLOGY

3.1 Grid Generation

ANSYS CFX was used to carry out the numerical study. The fluid domain was extracted for one impeller blade passage with the blade itself in the middle of the domain. The surface mesh is generated by using an hexahedral mesh on the impeller, splitter and vane surfaces. The surface repair tools have sufficient control to allow the analysis by choosing among the components to include and exclude in the meshing. This is to control the size of mesh in various parts by using surface curvature or by defining local refinement zones. Once these surface mesh control settings are defined, the tool retains the association with the imported CAD parts. This makes parametric modelling of the components very easy. The finite elements mesh is generated by using hexahedral, as validated for flow and thermal solutions [15,16]. The hexahedral element mesh consists of 8 nodes, agglomerated from the underlying automatically generated hexahedral mesh. Hexahedral meshes offer significant advantages over traditional mesh types. The computational grid, for the modified impeller blade passage with two splitter comprising exactly 442035 structured hexahedral elements in multi-block environment, for conventional impeller blade passage with single splitter comprising exactly 327788 hexahedral elements, and for the vaned diffuser passage comprising 363300 hexahedral elements were generated using TurboGrid. Sufficiently

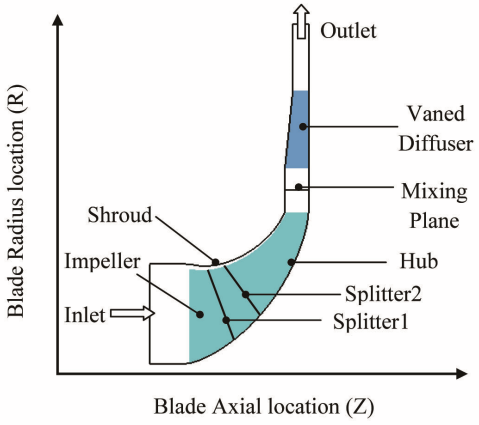


Fig. 1 Meridional view of the impeller, vaned diffuser, hub and shroud

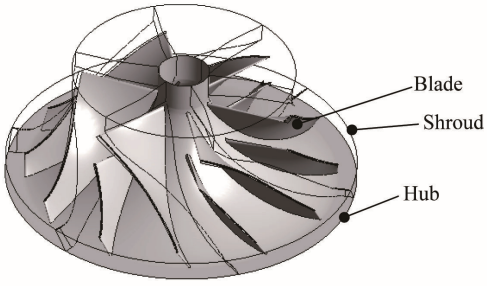


Fig. 2 Isometric 3D view of conventional centrifugal compressor (blade, hub and shroud)

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

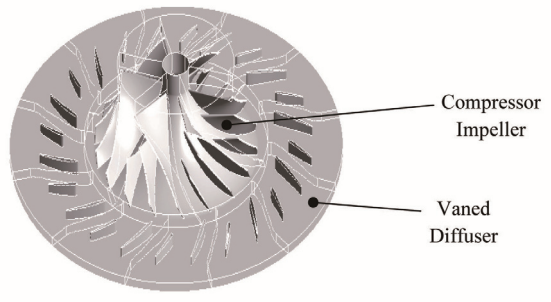


Fig. 3 Isometric 3D view of modified vaned diffuser centrifugal compressor stage

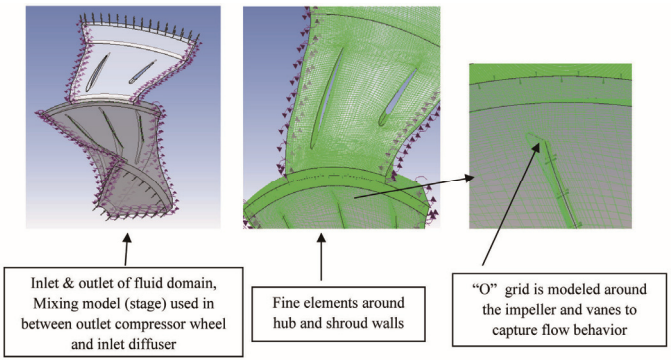


Fig. 4 Details of computational grid

fine grid elements were created in the impeller tip clearance region, around the impeller, and at the hub and shroud walls, as shown in Fig. 4. Sufficient mesh quality checks were performed by keeping the parameters like mesh angle and determinants within acceptable limits.

3.2 Effect of Grid Independence

Grid size plays an important role in both convergence and accuracy of the solution. A coarse mesh is initially used to quickly examine the solver settings and boundary conditions. The grid generated should be appropriate to capture the complex flow phenomena like boundary layers, flow separation, leakage flows and secondary vortices in the blade passage. To capture all these flow characteristics, the number of cells should be large enough. But, as the number of cells increases, the computation time also increases rapidly. The grid size is, therefore, a compromise between computational time and accuracy of the results. Finer grids, in general, make the solution independent of the grid size and yield more accurate results but always require lager computational resources and time. Thus, a compromise between the grid size on one hand and convergence and accuracy on the other hand is required. Hence, a grid independence study was carried out to ensure that the numerical solutions are grid-independent. Hence, a fine grid size of elements was used for the CFD simulations reported in this paper.

3.3 Fluid Flow Modelling

The fluid zone comprises of one area enveloping all the rotating parts (blades and hub) and the other area of the stationary parts (shroud, inlet, outlet and vaned diffuser parts). The left and right boundaries are defined as periodic. The key simplifying assumption in the analysis of the flow inside centrifugal compressor has always been that the inherently (unsteady) flow in the machine, can be treated as steady when viewed in the rotating reference frame of blade. Designs of all of the most efficient machines now in service have been made by applying steady flow assumption. The assumptions of this work were 3D steady state and fully turbulent. Turbulence is modelled using the k-ω-SST model. This model is a zonal combination of k-ω near the wall, nominally in the boundary layer, and k-ε away from the walls. When the near-wall mesh is compatible with the wall-function approach, this model behaves predominantly as a high-Reynolds number k-ε formulation. All surfaces are treated as adiabatic. Total pressure and total temperature are applied to the inflow inlet boundary. The outflow outlet condition is set to static pressure. The exit static pressure is modified in stages and a new analysis is run to determine the mass flow rates. The exit pressure is adjusted from the surge to the choke limit to give constant speed compressor performance curves.

4. RESULTS

The numerical method used by the solver part of the software requires an iterative process in order to obtain a solution. In general, the residual (RMS) magnitude should decrease to the residual target (1 E-4) as the solution converges. When the magnitude of the residuals for all the quantities falls below the convergence level, the solver will stop iterating, and the results will be exported for post-processing. The pressure ratio of the modified vaned diffuser centrifugal compressor stage is lower than the pressure ratio of the conventional and modified vaneless diffuser design because of the effects of the vaned diffuser on the conversion of the kinetic energy into static pressure. Moreover, the effects of vaned diffuser on the Polytropic efficiency and all performance parameters are clearly shown in Table 2. The results are compared with experimental work data which was presented by Galindo *et al.* [2] for validation. It is found that the behaviour of low pressure ratio point for a corresponding specific speed very closely to the experimental work data as shown in Fig. 5.

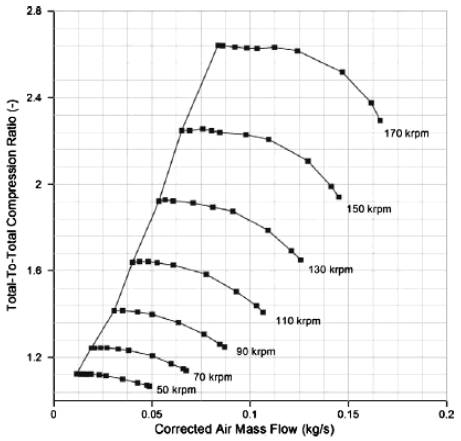
Figure 6 shows the Meridional velocity difference across the trailing edge (TE) Spanwise of the centrifugal compressor impeller configurations it is clearly seen that Meridional velocity leaving the modified impeller vaned diffuser type is higher than the Meridional velocity of conventional type and modified vaneless diffuser due to the space between impeller exit and vanes inlet.

Figure 7 shows the relative Mach number difference across the trailing edge (TE) Spanwise of the centrifugal compressor impeller, it is clearly seen that relative Mach number is increasing for the modified vaned diffuser centrifugal compressor type.

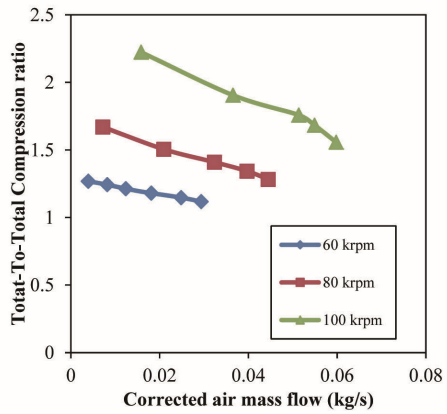
https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

Table 2 Performance parameters of centrifugal compressor

Performance Parameters	Conventional Type	Modified Type Without Vaned Diffuser	Modified Type With Vaned Diffuser
Rotation Speed	60k RPM	60k RPM	60k RPM
Mass Flow Rate	0.3931kg/s	0.5844 kg/s	0.5134 kg/s
Total Pressure Ratio	1.575	1.7055	1.3215
Total Temperature Ratio	1.1605	1.1889	1.177
Total Isentropic Efficiency.	86.7788	85.5087	56.3125
Total Polytropic Efficiency.	87.4277	86.2327	57.9522
Absolute Mach Number (Mabs) at Outlet	0.8433	0.8972	0.5863
Relative Mach Number (Mrel) at Outlet	0.9236	0.9155	
Diffuser Static Pressure Rise Coefficient (Cp)			0.2239



(a) Experimental results



(b) Numerical results

Fig. 5 Comparison of experimental work [2] with numerical work

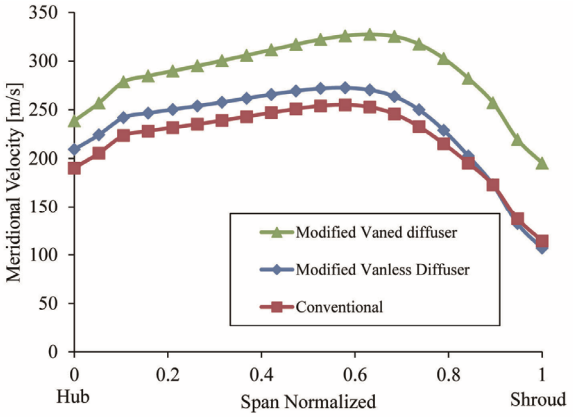


Fig. 6 Relationship between the meridional velocity (Cm) and span normalized at trailing edge (TE) for different compressor configurations

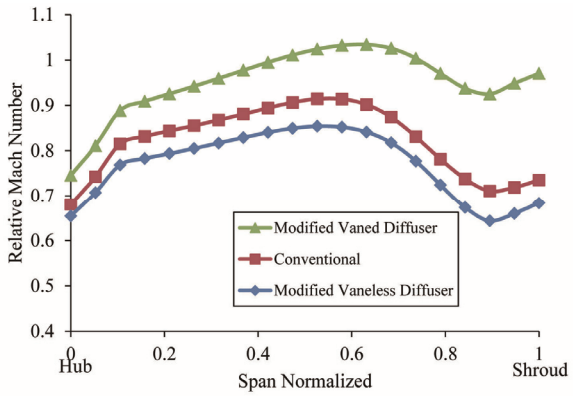


Fig. 7 Relationship between the relative Mach number (Mrel) and span normalized at trailing edge (TE) for different compressor configurations

Figure 8 shows Mach number difference across the Spanwise location at trailing edge of the vaned diffuser, it is visibly seen that Mach number is higher at 30% span nearby hub layer.

Figure 9 shows the static pressure difference across the stream wise location of the modified vaned diffuser centrifugal compressor, it is clearly seen that the static pressure is decreasing at space between exit impeller and inlet vaned diffuser, moreover it is obviously seen that static pressure is increasing through vaned diffuser due to the kinetic energy which is converted into static pressure.

Figure 10 shows the absolute Mach number difference across the streamwise location of the modified vaned diffuser centrifugal compressor it is evidently seen that absolute Mach number is at peak at the space between exit impeller and inlet vaned diffuser. However the effect of compressibility, when the absolute velocity of a compressible fluid reaches the speed of sound in the fluid, separation of flow causes excessive pressure losses. It is clearly seen, that the value of absolute Mach number in the vaned diffuser streamwise location wasn't exceeding the value at which shock waves occur.

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

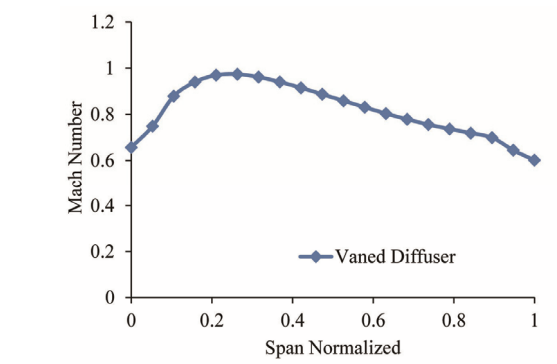


Fig. 8 Relationship between the mach number with spanwise location at trailing edge of vaned diffuser

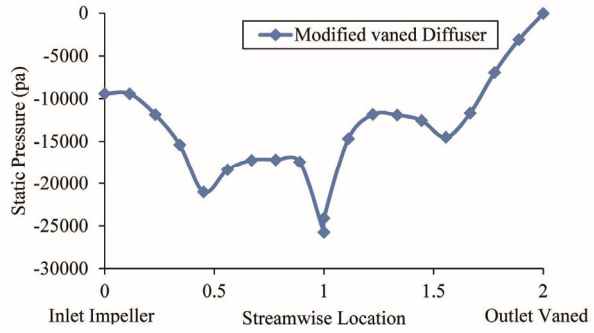


Fig. 9 Relationship between the static pressures (Ps) with streamwise location (inlet-outlet)

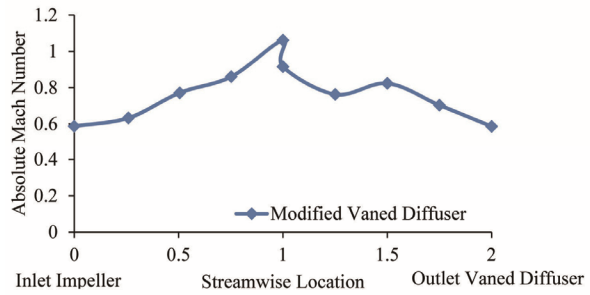


Fig. 10 Relationship between the absolute Mach number with streamwise location (inlet-outlet)

Figure 11 shows the meridional velocity difference across the Trailing edge (TE) Spanwise location of the modified Vaned diffuser centrifugal compressor, it is obviously seen that meridional velocity is at high value at 20% span.

Figure 12 shows the velocity vectors magnitude at mid spanwise location for impeller configurations from the hub layer to the shroud layer of the impeller. The high value of the velocity close to the trailing edge of the impellers indicates the flow would be sonic in the space area between the impeller and the diffuser.

Figure 13 shows the contour of relative Mach number at mid spanwise position for the conventional, modified vaneless diffuser and modified vaned diffuser as an example of CFD computations. There is no choke of flow at the inlet of the modified type because of the uniform distribution of the throat area between

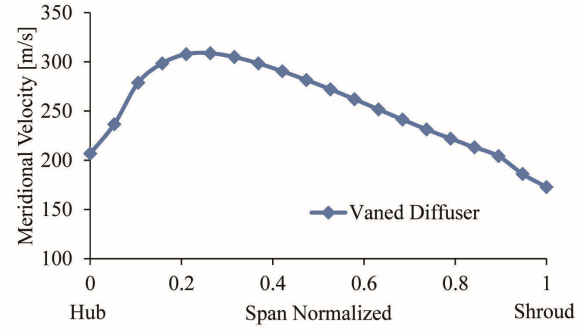
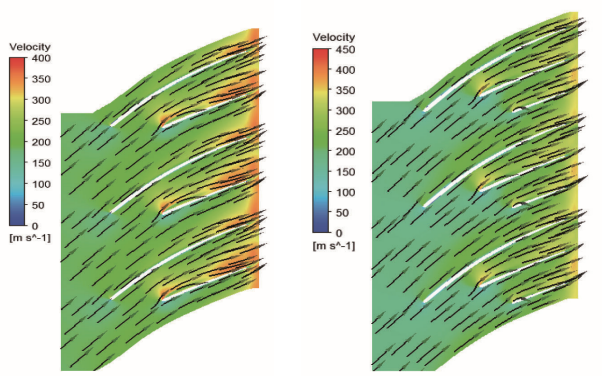
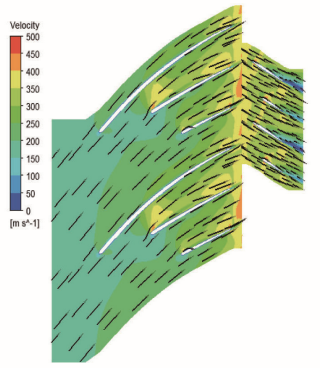


Fig. 11 Relationship between the meridional velocity (Cm) with span normalized at trailing edge (TE) of modified vaned diffuser centrifugal compressor type



(a) Conventional Type (b) Modified Vaneless Diffuser



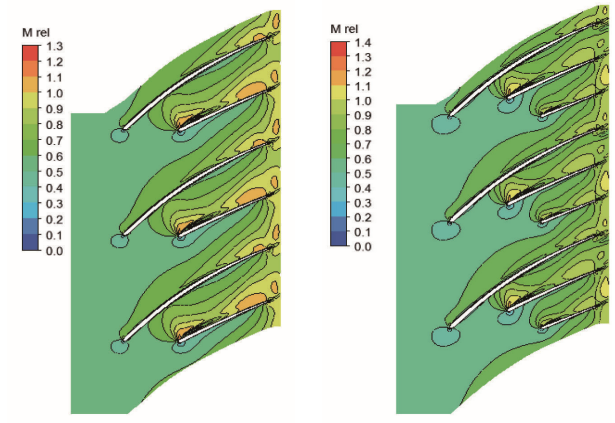
(c) Modified Vaned Diffuser

Fig. 12 Velocity vectors at 50% span position for the conventional, modified vaneless diffuser and modified vaned diffuser types

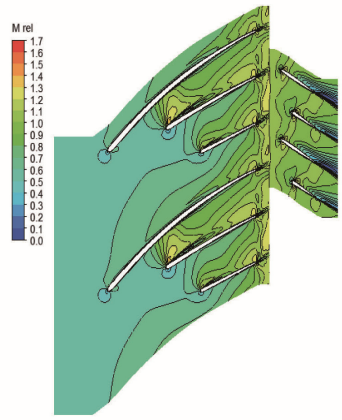
blade to blade passages. If we extend the leading edge of the splitters to the leading edge of the full blade, it will minimize the throat area and cause choking. We can see the flow in the space area between the trailing edge of the impeller, and the leading edge of the vanes diffuser which is close to Mach one meaning the space area ratio which is a very important factor to modify in order to remove any choking of the flow.

Figure 14 shows the contour of static pressure difference on the meridional surface of the impeller from leading edge to trailing edge location of the conventional, and modified vaneless and modified vaned diffuser centrifugal compressor impeller, it is obviously

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

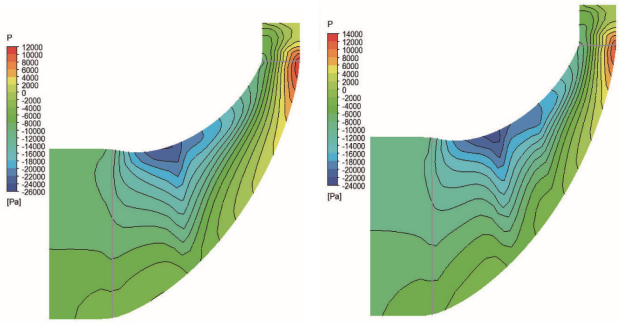


(a) Conventional Type (b) Modified Vaneless Diffuser

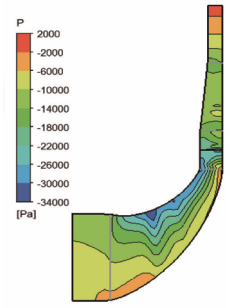


(c) Modified Vaned Diffuser

Fig. 13 Contour of relative mach number at 50% span position for the conventional, modified vaneless diffuser and modified vaned diffuser types



(a) Conventional Type (b) Modified Vaneless Diffuser



(c) Modified Vaned Diffuser

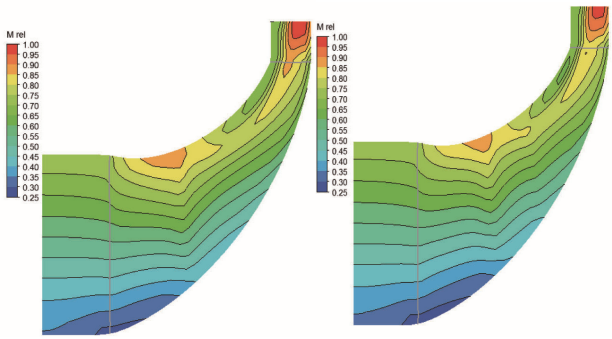
Fig. 14 Contour of static pressure on meridional surface for the conventional, modified vaneless diffuser and modified vaned diffuser centrifugal compressor types

seen that static pressure at the exit of vaned diffuser is increasing due to the kinetic energy which is converted into static pressure.

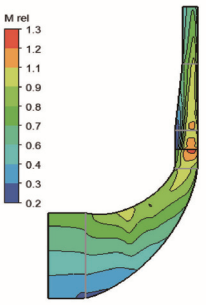
Figure 15 shows the relative Mach number on meridional surface from Leading Edge (LE) to Trailing Edge (TE) of the conventional, modified vaneless diffuser and modified vaned diffuser configurations. It is clearly seen that the relative Mach number of modified vaned diffuser type is lower than relative Mach number of others types at outlet flow due to the velocity which is decreased by vanes of diffuser. It is necessary to convert the high kinetic energy to a static pressure through a diffuser provided downstream of the impeller. The diffuser is used to reduce this velocity while at the same time it increases the static pressure.

Figure 16 shows the contour of meridional velocity from Leading Edge (LE) to Trailing Edge (TE) of the conventional, modified vaneless diffuser and modified vaned diffuser configurations. The high velocity is located at space between impeller exit and vanes inlet.

The numerical analysis was carried out including the impeller vaneless and vaned diffuser flow passage only. The results show the effect of vaned diffuser on the performance parameters of centrifugal compressor. The modification of a previous design of centrifugal compressor impellers gives a better performance or a wide operating range. CFD models give a much deeper understanding of the flow inside a vaned diffuser centrifugal compressor, enabling us to solve many problems easily and rapidly.



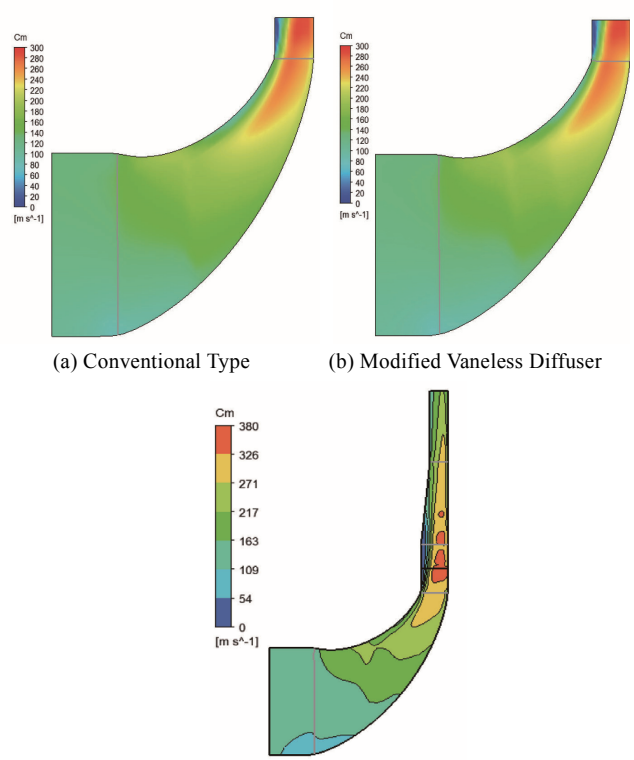
(a) Conventional Type (b) Modified Vaneless Diffuser



(c) Modified Vaned Diffuser

Fig. 15 Contour of relative mach number on meridional surface for the conventional, modified vaneless diffuser and modified vaned diffuser centrifugal compressor types

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press



(a) Conventional Type (b) Modified Vaneless Diffuser

(c) Modified Vaned Diffuser

Fig. 16 Contour of meridional velocity on meridional surface for the conventional, modified vaneless diffuser and modified vaned diffuser centrifugal compressor types

5. CONCLUSIONS

The result is of relevance in internal combustion engine boosting applications: Larger mass flow and compression ratio can be increased without experiencing surge. Compressor surge margin is a major concern in current automotive engines because the downsizing trend. However, the matching between compressor impeller and vaned diffuser is difficult because the surge limit obtained in steady conditions. Computational fluid dynamics (CFD) plays a greater part in the aerodynamic design of turbomachinery than it does in any other engineering application. For many years the design of a modern turbine or compressor has been unthinkable without the help of CFD and this dependence has increased as more of the flow becomes amenable to numerical prediction. The benefits of CFD range from shorter design cycles to better performance and reduced costs and weight.Steady state flow simulations have been conducted in order to analyse, the effect of vaned diffuser on the centrifugal compressor performance. The Parametric computations were performed on a 3D-Turbulent CFD to obtain the performance of backswept impeller configurations at a certain speed. The analysis of the flow characteristics was also performed to obtain a better understanding of the blade to blade vaned diffuser compressor behaviours. The results also show the potential of vaned diffuser to reduce the exit impeller velocity and to increase the static pressure; moreover, the compressor stage pressure ratio was 1.32, and the mass flow rate was 0.51kg/s, compared with vaneless diffuser the pressure ratio was 1.70 and mass flow rate was 0.58kg/s. An effort was made to model the flow from the inlet to the exit of a centrifugal compressor stage consisting of all the components in place using CFD tools. The vector plots, contour plots and Stream line plots are generated for better understanding of fluid flow through centrifugal compressor stage. Obviously, the performance was significantly affected by adding vaned diffuser due to the kinetic energy converted into static pressure.

NOMENCLATURES

CFD	Computational Fluid dynamics
CAD	Computer Aid Design
IGES	Initial Graphics Exchange Specification
TE	Trailing Edge
LE	Leading Edge
Pt	Total Pressure
Ps	Static Pressure
Tt	Total Temperature
Ts	Static Temperature
Mabs	Absolute Mach Number
Mrel	Relative Mach Number
Cm	Meridional Velocity
R	Blade Radius Location
Z	Blade Axial location
k-ω-SST	K-Omega Turbulence Model
k- ε	k-Epsilon Turbulence Model
3D	Three Dimensions

REFERENCES

1. Yang, Y., Xie, R., Gong, L.-Y. and Hai, Y., *Study of Influence of Diffuser Meridian Channel Shape on Performance of Micro-Gas Turbine Centrifugal Compressor. Power and Energy Engineering Conference (APPEEC)*, Asia-Pacific 978-1-4244-6255-1/11 (2011).
2. Galindo, J., Climent, H., Guardiol, C. and Tiseira, A., "On the Effect of Pulsating Flow on Surge Margin of Small Centrifugal Compressors for Automotive Engines," *Experimental Thermal and Fluid Science*, **33**, pp. 1163–1171 (2009).
3. Baris, O., "Automotive Turbocharger Compressor CFD and Extension Towards Incorporating Installation Effects," *Proceedings of ASME Turbo Expo 2011: Power for Land*, Sea and Air GT2011 (2011).
4. Guo, Q., Chen, H., Zhu, X-C., Du, Z-H. and Zhao, Y., "Numerical Simulations of Stall Inside a Centrifugal Compressor," *Proceedings of the Institute of Mechanical Engineers, Part A: Journal of Power and Energy*, **221**, pp. 683–693 (2007).

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press

5. Galindo, J., Serrano, J. R., Guardiola, C. and Cervello, C., "Surge Limit Definition in a Specific Test Bench for the Characterization of Automotive Turbochargers," *Experimental Thermal and Fluid Science*, **30**, pp. 449–462 (2006).
6. Jiang, W., Khan, Jamil and Roger A. Dougal "Dynamic Centrifugal Compressor Model for System Simulation," *Journal of Power Sources*, **158**, pp. 1333–1343 (2006).
7. Engeda, A., "Experimental and Numerical Investigation of the Performance of a 240Kw Centrifugal Compressor with Different Diffusers," *Experimental Thermal and Fluid Science*, **28**, pp. 55–72 (2003).
8. Engeda, A., "Effect of Impeller Exit Width Trimming on Compressor Performance," *Proceedings of the 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows*, ISAIF8-00135 (2007).
9. Tamaki, H., "Effect of Piping Systems on Surge in Centrifugal Compressors," *Journal of Mechanical Science and Technology*, **22**, pp. 1857–1863 (2008).
10. Galindo, J., Arnau, F. J., Tiseira, A. and Piqueras, P., "Solution of the Turbocompressor Boundary Condition for One-Dimensional Gas-Dynamic Codes," *Mathematical and Computer Modelling*, **52**, pp. 1288–1297 (2010).
11. Cukurel, B., Lawless, P. B. and Fleeter, S., "Particle Image Velocity Investigation of a High Speed Centrifugal Compressor Diffuser, Spanwise and Loading Variations," *Journal of Turbomachinery*, **132**, pp. 1–9 (2010).
12. Higashimori, H., Hasagawa, K., Sumida, K. and Suita, T., "Detailed Flow Study of Mach Number 1.6 High Transonic Flow With a Shock Wave in a Pressure Ratio 11 Centrifugal Compressor Impeller," *Journal of Turbomachinery*, **126**, pp. 473–481 (2004).
13. Shook, R. M. P. J. and Oakes, W., "The Aerodynamic Performance of a High Speed Research Centrifugal Compressor Facility," *AIAA Paper 1994-2798*, pp. 1–9 (1994).
14. Gravdahl, J. T. and Willems, F., "Modeling of Surge in Free-Spool Centrifugal Compressors: Experimental Validation," *Journal of Propulsion and Power*, **20**, pp. 849–857 (2004).
15. Peric, M., *Flow Simulation Using Control Volumes of Arbitrary Polyhedral Shape*, ERCOFTAC Bulletin 62 (2004).
16. Mendonça, F., Clement, J., Palfreyman, D. and Peck, A., "Validation of Unstructured CFD Modelling Applied to the Conjugate Heat Transfer in Turbine Blade Cooling," ETC_8-198, European Turbomachinery Conference, Graz (2008).
17. Ziegler, K. U., Gallus, H. E. and Niehuis, R., "A Study on Impeller-Diffuser Interaction—Part I: Influence on the Performance," *Journal of Turbomachinery*, **125**, pp. 173–182.170 (2003).
18. Ramakrishnan, K., Lawless, P. B. and Fleeter, S., "High Speed Centrifugal Compressor Aeromechanics — Impeller Unsteady Aerodynamics," *AIAA 2007-5020*, Cincinnati, OH: AIAA, pp. 1–18 (2007).
19. Barry, W. B., "An Investigation of Unsteady Impeller-Diffuser Interactions in a Centrifugal Compressor," Purdue University Thesis (1991).
20. Gallier, K., Lawless, P. B. and Fleeter, S., "PIV Characterization of High Speed Centrifugal Compressor Impeller-Diffuser Interaction," *AIAA 2007-5019*, Cincinnati, OH: AIAA, pp. 1–8 (2007).
21. Ziegler, K. U., Gallus, H. E. and Niehuis, R., "A Study on Impeller-Diffuser Interaction-Part II: Detailed Flow Analysis," *Journal of Turbomachinery*, **125**, pp. 183–192 (2003).
22. Ibaraki, S., Matsuo, T. and Yokoyama, T., "Investigation of Unsteady Flow Field in a Vaned Diffuser of a Transonic Centrifugal Compressor," *Journal of Turbomachinery*, **129**, p. 686 (2007).
23. Layth, H., Jawad, S., Zulkifli, Abdullah R. and Mahmood, W. M. F. W., "Numerical Study on the Effect of Impeller Trimming on the Performance of a Modified Turbocharger Centrifugal Compressor," *Archives Des Sciences*, **65**, pp. 324–335 (2012).
24. Layth, H., Jawad, S., Zulkifli, Abdullah R. and Mahmood, W. M. F. W., "Modelling of Centrifugal Compressor Impellers using Adaptive Neuro-Fuzzy Inference Systems (ANFIS)," *International Review of Mechanical Engineering (IREME)*, **6**, (2012).
25. Layth, H., Jawad, S., Zulkifli, Abdullah R. and Mahmood, W. M. F. W., "Prediction of Centrifugal Compressor Performance by Using Adaptive Neuro-Fuzzy Inference System (ANFIS)," *International Review on Modelling and Simulations (IREMOS)*, **5**, p. 1580 (2012).
26. Layth, H., Jawad, S., Zulkifli, Abdullah R. and Mahmood, W. M. F. W., "Numerical Simulation of Flow Inside a Modified Turbocharger Centrifugal Compressor," *Asian Journal of Applied Sciences*, **5**, pp. 563–572 (2012).

(Manuscript received November 14, 2012, accepted for publication March 28, 2013.)

https://doi.org/10.1017/jmech.2013.49 Published online by Cambridge University Press