Numerical Simulation of Flow inside a Vaned Diffuser
Of a Modified Centrifugal Compressor

Layth H. Jawad, Shahrir Abdullah, Rozli Zulkifli and Wan Mohd F. Wan Mahmood
Department of Mechanical and Materials Engineering, Faculty of Engineering and Built Environment,
Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Selangor, MALAYSIA

Abstract

In this paper, the flow inside a small modified centrifugal compressor with a vaned diffuser used in an automotive turbocharger application was simulated. It has been known that the performance of the centrifugal compressor depends on the interaction between the compressor impeller and the vaned diffuser. The modified compressor comprises two splitters, which are not extended to the leading edge. If the splitters extend to the leading edge, it would produce a sufficient blockage to cause choking at high speed. This research aimed to study and simulate the effect of a vaned diffuser on the performance of a modified turbocharger compressor. The simulation was undertaken using CFD analysis to predict the aerodynamic flow field and characteristics. The mesh generator of a CFD code was used to generate a polyhedral mesh. Steady state analysis was carried out for the stage with the mixing plane approach. According to the results of the simulation, the vaned diffuser flow is characterized by a subsonic flow and there is no choking in between impellers exit and vane inlet. It was also observed that the outlet diffuser velocity is lower than outlet impeller velocity. At the outlet of the vaned diffuser, the total pressure was found to decrease, and the static pressure increase.

Keywords: Numerical Simulation; Centrifugal Compressor; Vaned diffuser; Flow Characteristics; CFD.

Introduction

In the past decades, interest has been progressively more 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 conventional turbocharger compressor is constrained by surge or choke condition. The improvement of turbocharger compressor performance and the extension of the stable operating ranges are becoming critical for the viable future of low emission diesel engines. In the case of the centrifugal compressor, 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 throughout the compressor into its inlet. In general, 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 (MGT) centrifugal compressor [1]. In addition, the stall flow phenomenon inside a turbocharger centrifugal compressor with a vaneless diffuser simulated numerically, and it was found that the amplitude of the static pressure oscillation at this frequency in the diffuser is increased with reduction in compressor mass flow. The results show that there is a distinct stall frequency at the given compressor speed [3]. 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 [5]. 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 [4].

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 [6]. The effect of impeller exit width trimming were studied and discussed and the effect on overall performance, blade loading and impeller diffusion was examined experimentally by [7]. The effects of different piping systems on the surge features were studied and tested on a number of turbocharger compressors by [8]. Therefore, the simulation was carried out to study surge phenomena and steady working conditions by using Characteristic's method to find out the flow conditions of the compressor. Stable working conditions and surge phenomena were simulated with the Method of Characteristics to determine the flow conditions at compressor inlet and outlet. The output power was found to increase and fuel consumption decrease when the engine displacement was downsized [9]. The complex shock waves within the diffuser throat and impeller inlet, respectively, were found high-speed compressors. This flow phenomenon does not occur in low speed compressors and are very significant in the design of these compressors [13,14]. 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 necessity for high efficiency and compact size has led to the extensive use of vaned diffusers in centrifugal stages. The diffuser is usually subjected to a strong three-dimensional non uniform unsteady flow discharged from the impeller. In order to achieve higher diffuser performance and to expand the operating range, it is believed that it may be essential to take into account the influence of a vaned diffuser on the impeller aerodynamic flow and characteristics. To this aim, in the past years considerable attention has been paid to vaned diffuser flow. The flow inside blade to blade passage is very affected by the throat area at the inlet of the vaned diffuser. The present work is to computationally study the flow inside a vaned diffuser which was

modified from a conventional vaneless type centrifugal compressor.

Design and Specifications

The conventional turbocharger compressor studied was a centrifugal compressor model GT1749V Trim55 with a vaneless diffuser. The compressor modifications were carried out by adding second splitters and a vaned diffuser NACA shape model (0012). The inflow and the outflow of the fluid zone were as shown in Figure 1. The main geometry features and dimensions of a conventional compressor and the modified compressor are given in Table 1.
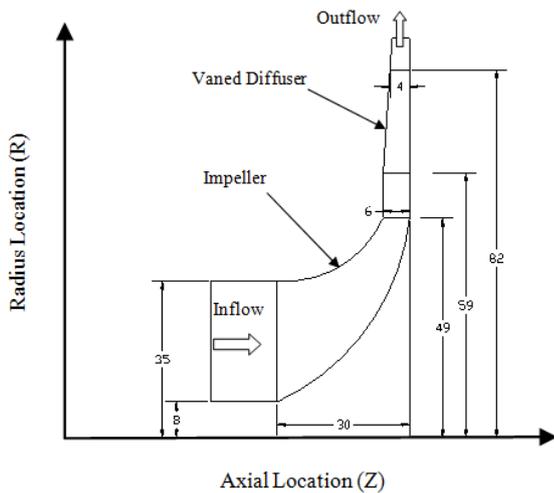
Figure 1: Meridional Plane View of Full Impeller   Blade Units in mm.

Table 1: Geometric Features of a Turbocharger Centrifugal Compressor

Turbocharger compressor	Conventional Type	Modified Type
Axial width of impeller in meridional view	39.257 mm	30 mm
Inner Radius at compressor inlet	12 mm	8 mm
Outer Radius at compressor inlet	36.25 mm	35 mm
Impeller outer diameter	49 mm	49 mm
Impeller width at trailing edge	6 mm	6 mm
Diffuser Inlet Radius		59 mm
Diffuser Outlet Radius		82mm
Number of blades	12	18
Number of splitters	6	12
Number of Diffuser Vanes	0	18

Figure 2 shows a conventional turbocharger centrifugal compressor wheel comprising of six main impeller blades and six splitter blades, and Figure 3 shows the geometry of the modified turbocharger compressor wheel comprising of six main impeller blades and twelve splitter blades with an eighteen-vane diffuser. The CFD computations for the modified design were performed on the geometry as shown in Figure 3. All the surface geometry, inlet, exit, and periodic boundaries, were defined via computer-aided design (CAD) as Initial Graphics Exchange Specification (IGES) parts.
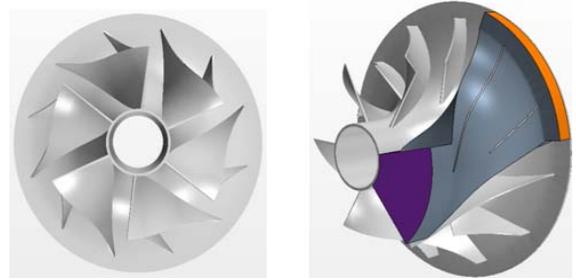

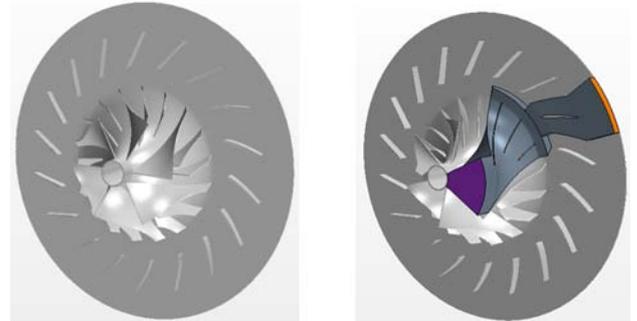Figure 2: Conventional Turbocharger Centrifugal Compressor.

Figure 3: Modified Turbocharger Centrifugal Compressor.

CFD Methodology:

Surface and Volume Mesh Generation

The surface mesh is generated by using a re-triangulation mesh on the impeller and splitter surfaces. The surface repair tools have a sufficient control to allow the analysis to choose which components to include and exclude in the meshing. This is to control the size of the triangulations 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 volume mesh is generated by using a polyhedral, as validated for flow and thermal solutions [10-11]. The polyhedral cell mesh consists of from 12-16 faces, agglomerated from the underlying automatically generated tetrahedral mesh. Polyhedral mesh meshes offer significant advantages over traditional mesh types. As with tetrahedral and unlike hexahedral meshes, they can be automatically generated. Polyhedral meshes exhibit far less numerical diffusion compared to tetrahedral meshes because of the greater likelihood of face alignment to the flow. Gradient calculations are more accurate due to the greater number of face neighbours. Cell counts are typically a third of the equivalent tetrahedral meshes for similar fineness of resolution. All these mean that polyhedral meshes run faster, are more accurate and converge more robustly than tetrahedral meshes [10].

Figure 4 shows the polyhedral mesh through the rotational plane, and also shows a schematic of the normal to wall extrusion layers in the boundary layer of the blade to blade passage.
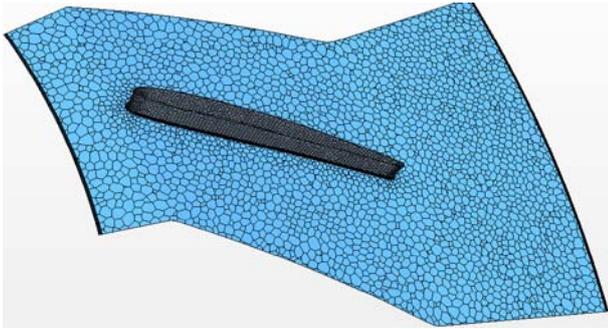
Figure 4: Volume Polyhedral Mesh.

Fluid Flow modelling

STAR CCM+ uses a compressible implicit coupled algorithm [12]. The fluid zone comprises one area enveloping all the rotating part (compressor wheel) and the other area the stationary part (Vaned diffuser). The left and right boundaries are defined as periodic. The mixing-plane interface type between outlet impeller and inlet vaned diffuser are available on an indirect interface topologies only. It is used to transfer "mixed-out" flow field data in a conservative manner between two regions of the same continuum. 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 at the inflow inlet boundary. The outflow outlet condition is set to static pressure. The models contain exactly 2,015,281 cells, which at a specific speed take approximately 7072.5068 (s) to run on a 96–core Linux machine.

Results

The numerical method used by the solver part of the software requires an iterative process in order to obtain a solution. The mass flow rate (0.0825308 kg/s) and the pressure ratio (1.30) of the modified design is higher than the mass flow rate (0.05291kg/s) and the pressure ratio (1.10) of the conventional design because of the effect of the second splitter on the performance of the turbocharger compressor as shown in Figure 5. The results are compared with experimental work data for validation. It is found that the calculations for the low-pressure ratio point for a specific speed correspond very closely to the predicted mass flow rate and to the measurements [2].
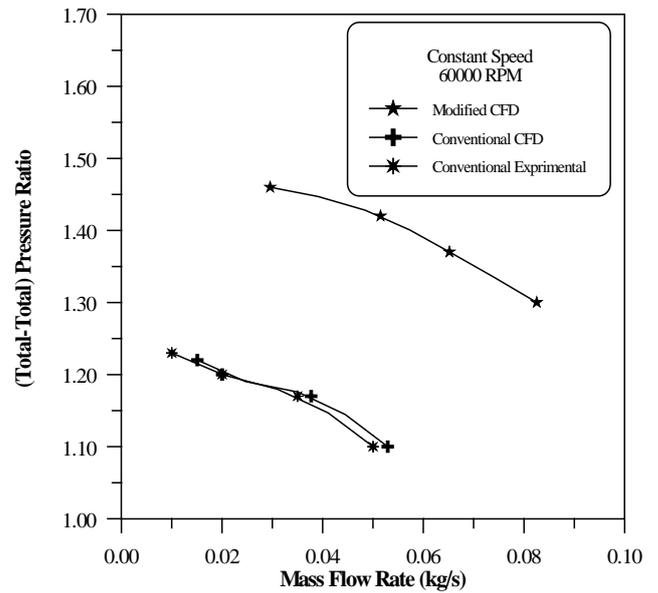

Figure 5: Shows the Relationship between the Mass Flow Rate and (T-T) Pressure Ratio for Modified and Conventional Type.

Figure 6 shows the velocity magnitude for different streamwise and mid span layers from the leading edge to the trailing edge of the vaned diffuser. The high value of the velocity close to the leading edge of the vanes indicates the flow would be sonic in the space area between the impellers and the vaned diffuser.
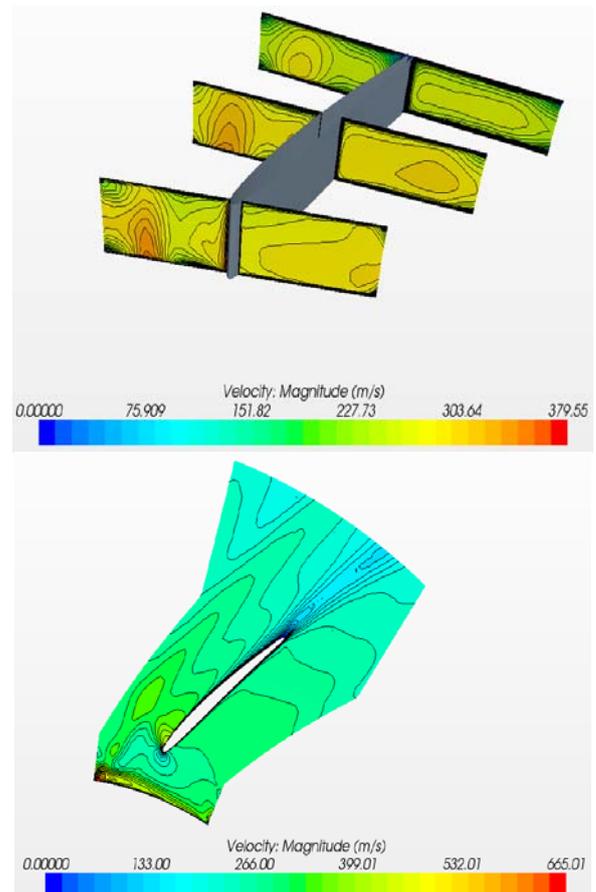

Figure 6: Velocity Magnitude for Different Streamwise and Mid Span Layers.

Figure 7 shows the relative Mach number at the mid span position for the vaned diffuser. There is no choke of flow at the inlet of the vanes because of the uniform distribution of the throat area between the vane to vane passages. We can see the flow in the space area between the trailing edge of the impeller, and the leading edge of the vanes' diffuser is a subsonic flow, which means the space area ratio it is a very important factor to remove any choking of the flow.
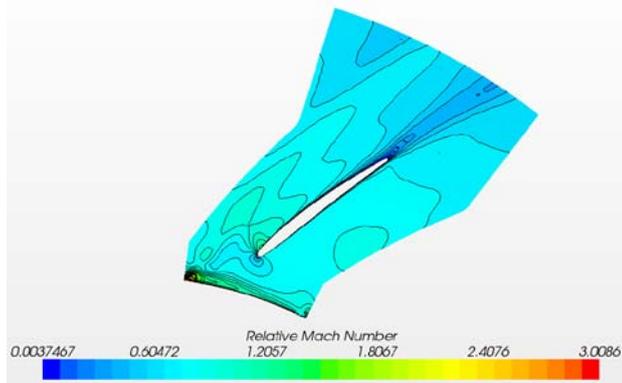

Figure 7: Relative Mach number for at Mid Span Layer 50%.

Figure 8 shows the relative Mach number for different streamwise layers from (Leading Edge-Trailing Edge) of the conventional and modified compressor impellers. The high Mach due to the high outlet velocity, therefore, leads to choke, and 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 increase the static pressure.
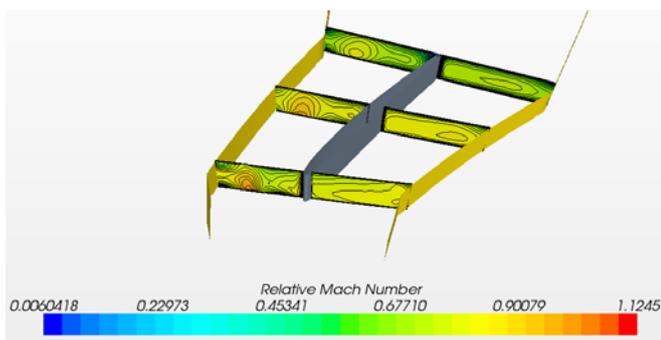

Figure 8: Shows Relative Mach number for Different Streamwise Layers.

Figure 9 shows the relative velocity vector at the mid span directed from the upstream point to be downstream in the impeller duct. The fluid flow is great without apparent backflow or vortex. In the trailing area, speeds gradually decrease from the inlet to the outlet.
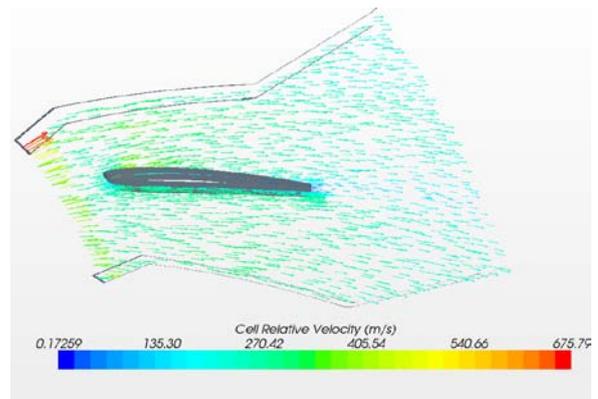

Figure 9: Cell Relative Velocity Vectors at Mid Span Layer 50%.

The numerical analysis was carried out including just the impeller flow passage and vaned diffuser passage. As we mentioned earlier that the flow is heavily affected by the impeller design, so the results show the effect of the vaned diffuser on the performance of compressor to convert the high outlet impeller velocity into static pressure and decrease the velocity from the inlet vanes to the outlet.

Conclusions

Steady state CFD simulations have been conducted in order to study the characteristics of the flow inside a vaned diffuser of a modified turbocharger compressor. The computations were performed on a three dimensional turbulent CFD to obtain the performance of a turbocharger backswept impeller at a specific speed. The analysis of the flow characteristics was also performed to obtain a better understanding of the blade to blade vaned diffuser behaviours. The results also show the potential of a double splitter to improve the performance of a centrifugal compressor and also increase the outlet mass flow rate. The relative velocities flowing out of the impeller were quite high, but with a vaned diffuser, the relative velocities leaving the diffuser were quite lower than velocities at the impeller exit. Obviously, the performance was greatly affected by the impeller and vaned diffuser design because of the nature of the flow in the impeller and vaned diffuser interaction.

References

[1] Y. Yang, RongXie,Lu-yuan Gong and Yang Hai. Study of Influence of Diffuser Meridian Channel Shape on Performance of Micro-Gas Turbine Centrifugal Compressor. Power and Energy Engineering Conference (APPEEC),2011 Asia-Pacific 978-1-4244-6255-1/11.

[2] J. Galindo,H. Climent, C. Guardiol, A. Tiseira. On the effect of pulsating flow on surge margin of small centrifugal compressors for automotive engines. Experimental Thermal and Fluid Science,2009,33:1163–1171.

[3] Q Guo,H Chen, X-C Zhu, Z-H Du, and Y Zhao. Numerical simulations of stall inside a centrifugal compressor. Power and Energy IMechE, 2007, Vol. 221 Part A: J.

[4] J. Galindo,J.R. Serrano , C. Guardiola, and C. Cervello. Surge limit definition in a specific test bench for the characterization of automotive turbochargers. Experimental Thermal and Fluid Science, 2006, 30, 449–462.

[5] Wei Jiang,Jamil Khan, and Roger A. Dougal. Dynamic centrifugal compressor model for system simulation. Journal of Power Sources,2006, 158, 1333–1343.

[6] Abraham Engeda. Experimental and numerical investigation of the performance of a 240 kW centrifugal compressor with different diffusers. Experimental Thermal and Fluid Science, 2003,28:55–72.

[7] Abraham Engeda. Effect of Impeller Exit Width Trimming on Compressor Performance. Proceedings of the 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows.ISAIF8, 2007, 00135.

[8] Hideaki Tamaki. Effect of piping systems on surge in centrifugal compressors. Journal of Mechanical Science and Technology, 2008, 22 , 1857~1863.

[9] J. Galindo, F.J. Arnau, A. Tiseira and P. Piqueras (2010). Solution of the turbocompressor boundary condition for one-dimensional gas-dynamic codes. Mathematical and Computer Modelling,2010,52 1288_1297.

[10] M. Peric, Flow simulation using control volumes of arbitrary polyhedral shape. ERCOFTAC Bulletin, 2004, 62.

[11] 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.

[12] STAR-CCM+, Release Version 6.02, CD-adapco, 2011, www.cd-adapco.com.

[13] B. Cukurel, P.B. Lawless, and S. Fleeter, Particle Image Velocity Investigation of a High Speed Centrifugal Compressor Diffuser, Spanwise and Loading Variations. Journal of Turbomachinery, 2010, vol. 132, pp. 1-9.

[14] H. Higashimori, K. Hasagawa, K. Sumida, and T. Suita, 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, 2004, vol. 126, pp. 473-481.