CFD Letters www.cfdl.issres.net Vol. 5 (4) – December 2013

Numerical Investigation on the Effect of Impeller Trimming on the Performance of a Modified Compressor

Layth H. Jawad[1, 2*], S. Abdullah[2], R. Zulkifli[2], and W.M.F.W. Mahmood[2]

[1]*Foundation of Technical Education, Baghdad, IRAQ*
[2]*Deprtment of Mechanical & Materials Engineering, Faculty of Engineering & Built Environment, The National University of Malaysia, MALAYSIA*

Received: 17/07/2013 – Revised 02/11/2013 – Accepted 02/11/2013

Abstract

This study was intended to predict the performance of a modified centrifugal compressor used in turbo charger numerically. In order to increase the performance of the conventional turbocharger compressor's to boost the pressure in the engine, by the impeller trimming. The compressor impeller was trimmed at the impeller hub and shroud layers, thus generating different trim geometries, and the effect of the trimming on the compressor impeller performance was studied. The flow inside these geometries of a small modified centrifugal compressor with double splitters used as an automotive turbocharger was simulated. Further, the performance of the centrifugal compressor depended on the flow in the impeller and splitters, but the splitters were not continued to the leading edge. If the splitters continued to the leading edge, it would produce sufficient blockage to cause choking at high speed. This research aimed to study and simulate the effect of impeller trimming on the performance of a turbocharger compressor. The simulation was undertaken using CFD analyse on the aerodynamic flow field and to predict performance in terms of pressure ratio, efficiency, mass flow rate and aerodynamic characteristics. According to the results of the simulation, it was observed that the fluid flow into a compressor has been indicated a better understanding. In addition the compressor performance was heavily affected by the impeller modification. Conclusively, it was observed the performance it was greatly affected by the double splitter design and the pressure ratio and air mass flow rate were increased.

Keywords: Computational Fluid Dynamic; Turbocharger; Centrifugal Compressor; Double splitter

1. Introduction

A turbocharger is one of the most effective techniques for increasing the volumetric efficiency and performance of an internal combustion engine as it increases the air density inside the cylinders. The conventional turbocharger compressor is constrained by surge or chokes conditions. The improvement of turbocharger compressor performance and the extension of the stable operating

* Corresponding Author: Layth H. Jawad
Email: Laihasan@yahoo.com Telephone: +601 8935908
 PII: S2180-1363(13)54174-X

ranges are becoming critical for the viable future of low emission diesel engines. In this 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 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 on a line with a constant number of revolutions.

Furthermore, the influences of the different diffuser meridian channel width ratios on the compressor performance show a remarkable significance in terms of improving the efficiency of the whole machine [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 40-67 Hz range (corresponding to characteristic pulsation when boosting an internal combustion engine) which increases the surge margin [2]. The application of CFD to turbocharger compressor characteristic predictions over a range of speeds in order to develop an efficient methodology for analysing the turbocharger compressor performance, Also 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 increases with reduction in compressor mass flow, the results show that there is a distinct stall frequency of the given compressor speed [4]. An analytical model of 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 were studied and discussed and 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 was studied and test several centrifugal compressors for turbochargers combined with the different piping systems and investigate 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 uses the Method of Characteristics to determine the flow conditions at compressor inlet and outlet. In order to increase the power output and reducing fuel consumption by downsize the engine displacement [10]. The complex shock waves within the diffuser throat and impeller inlet, respectively, within high-speed compressors. These flow phenomena do 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 a surge and chokes on a compressor performance map. Much effort has been spent to define performance maps of a new high speed compressor facility at Purdue University [13]. 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. In view of the above, the work reported in this paper deals with numerical investigations on centrifugal compressor impeller trimming with different configurations, CFD simulations and flow behaviour of each impeller configuration were performed. To study the effect of impeller trimming in order to achieve further performance improvement of the turbocharger compressor, by boosting the outlet pressure and mass flow rates.

2. Design & specifications

The turbocharger compressor studied was a centrifugal compressor model GT1749V Trim55.The inflow and the outflow of the fluid zone was as shown in Figure 1. The main geometry features and dimensions of a conventional compressor and the modified compressor are given in Table 1.

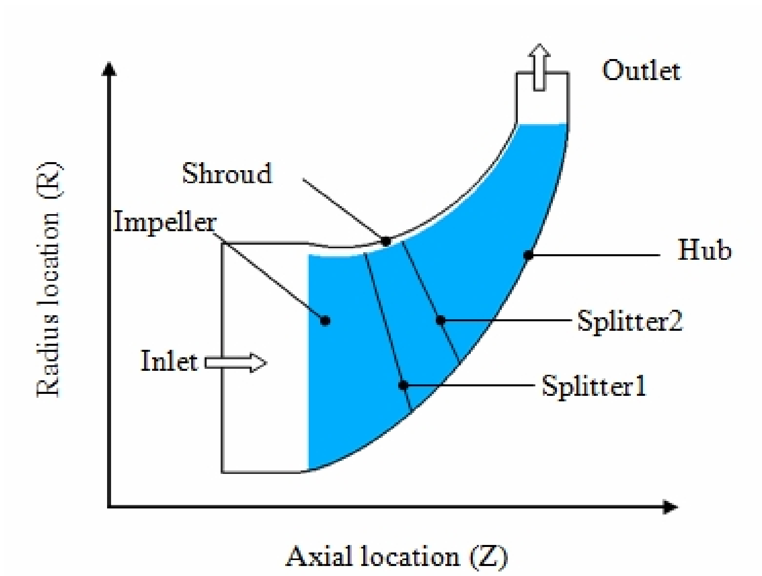


Figure 1: Meridional Plane View of a Full Impeller Blade.

TABLE 1: GEOMETRIC FEATURES OF CENTRIFUGAL COMPRESSOR

Turbocharger Compressor	(Conventional) Single Splitter	Modified1 (Double Splitter)	Modified2 (Double Splitter)	Modified3 (Double Splitter)
Axial width of impeller in meridional view	39.25 mm	39 mm	35 mm	30 mm
Inner radius at compressor inlet	12 mm	8 mm	8 mm	8 mm
Outer radius at compressor inlet	36.25 mm	35 mm	35 mm	35 mm
Impeller outer radius	49 mm	49 mm	49 mm	49 mm
Impeller width at trailing edge	6 mm	6 mm	6 mm	6 mm
Diffuser inlet radius	55 mm	55 mm	55 mm	55 mm
Number of blades	12	18	18	18
Number of splitters	6	12	12	12

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. 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.

Figure 4 shows the radius location of impeller trimming configurations for the hub and shroud layers with respect to axial location. Figure 5 shows the modified impeller Angles Distribution (Theta & Beta) With Respect to Radius Location for the Impeller Configurations at

Span 0 (Hub Layer) and Span 1 (Shroud Layer). Figure 6 shows the impeller Position Angles Distribution With Respect to Radius Location for the Conventional and Modified Types at Span 0 (Hub Layer) and Span 1 (Shroud Layer).

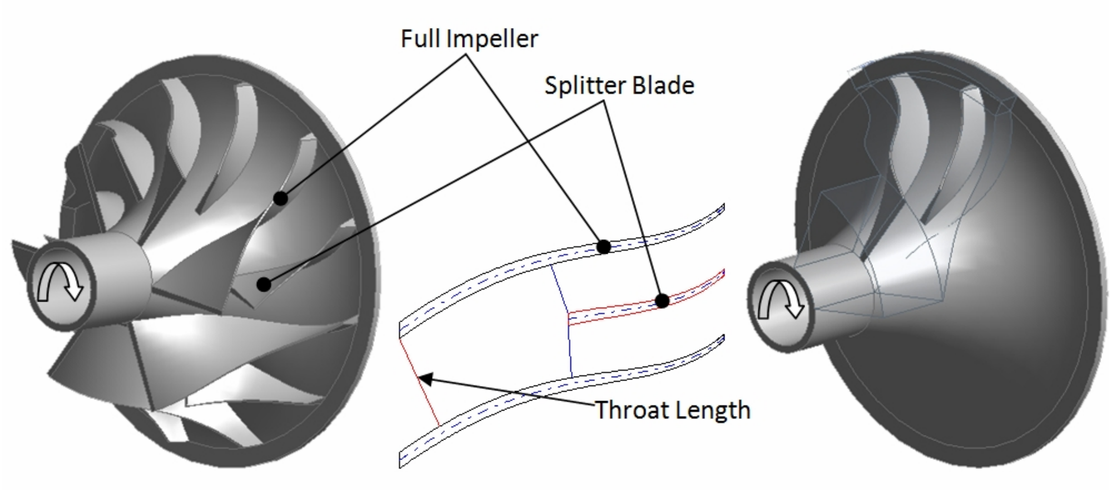


Figure 2: Conventional Centrifugal Compressor.

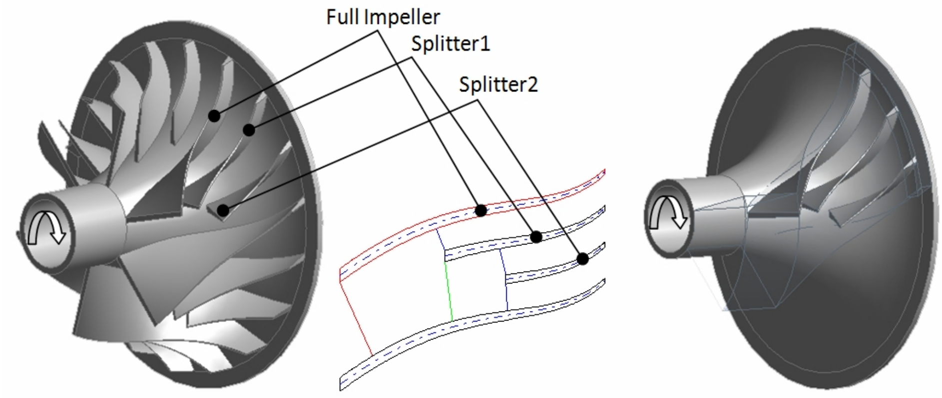


Figure 3: Modified Centrifugal Compressor.

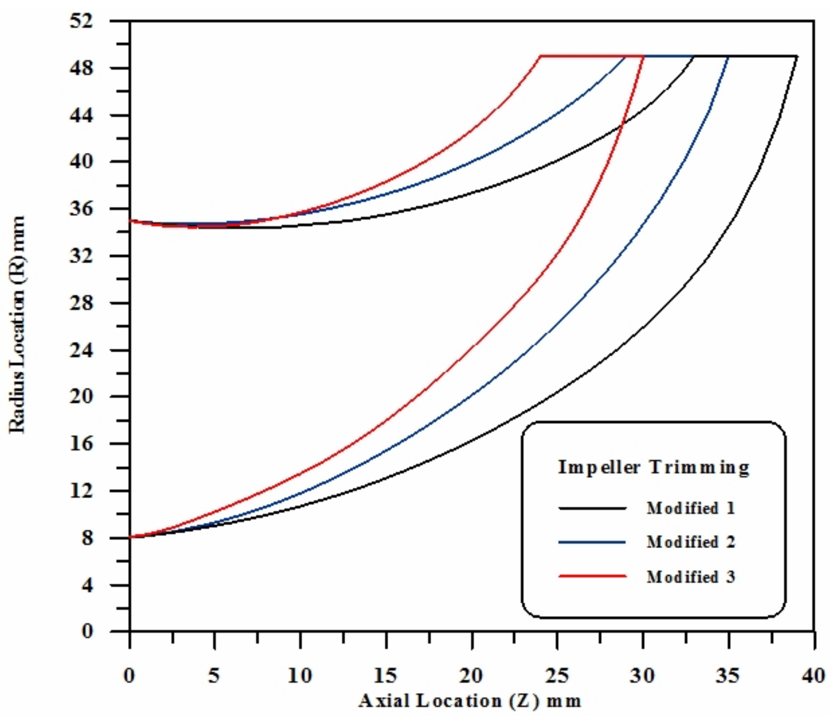


Figure 4: Radius Location With Respect to the Axial Location for Different Configurations of Impellers Trimming.

3. CFD Methodology

3.1. 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 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 the 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 polyhedral, as validated for flow and thermal solutions [15-16]. The polyhedral cell mesh consists of about 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 with 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 [15]. Figure 5 shows polyhedral meshes 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.

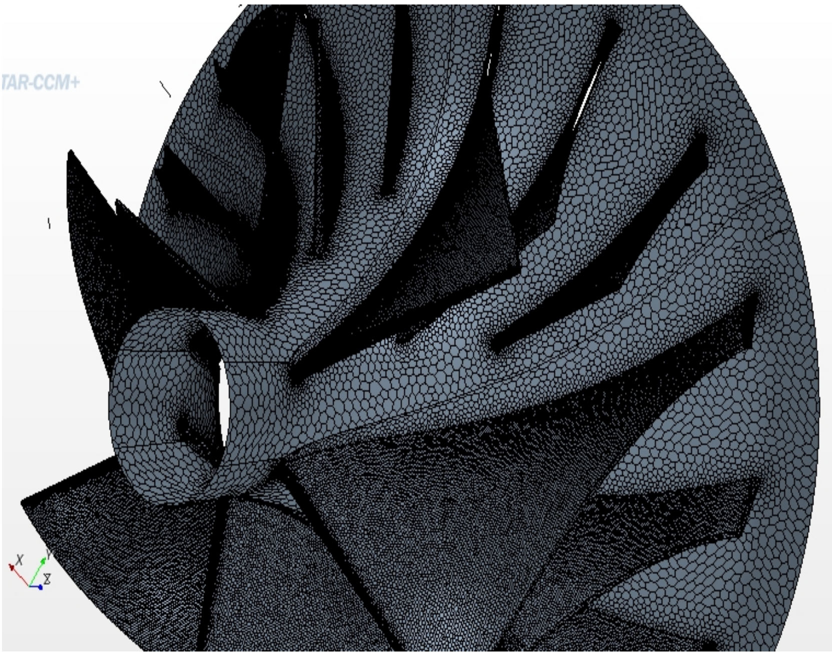


Figure 5: Volume Polyhedral Mesh.

3.2. Modelling of fluid flows

A compressible implicit coupled algorithm used in [17]. The fluid zone comprises one area enveloping all the rotating parts (blades and hub) and the other area the stationary parts (shroud, inlet and outlet). The left and right boundaries are defined as periodic. 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 exit static pressure is modified in stages and a new analysis is run to determine the mass flow rates. The exit pressure is adjusted (4) from the surge to the choke limit to give a constant speed compressor performance curves. The models contain exactly 559,445 cells, which at a specific speed take approximately 1993.357666 (s) to run on a 96–core Linux machine.

4. Results and discussion

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 magnitude should decrease 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 mass flow rates and the pressure ratio of the modified designs are higher than the mass flow rate and pressure ratio of the conventional design because of the effects of the second splitter and impeller trimming on the performance of compressor as shown in Figure 6. Moreover, the effects of impeller trimming on the efficiency for the impeller configurations are clearly shown in Figure 7. 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 rates and to the measurements [2].

Figure 8 shows the cell relative velocity magnitude for impeller configurations from the leading edge to the trailing edge of the impellers of the modified types. The high value of the velocity close to the trailing edge of the blades indicates the flow would be sonic in the space area between the impellers and the diffuser.

Figure 9 shows the relative Mach number at the mid span position for the modified impellers 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 the 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 vane diffuser is close to Mach one which means the space area ratio it is a very important factor to modify in order to remove any choking of the flow.

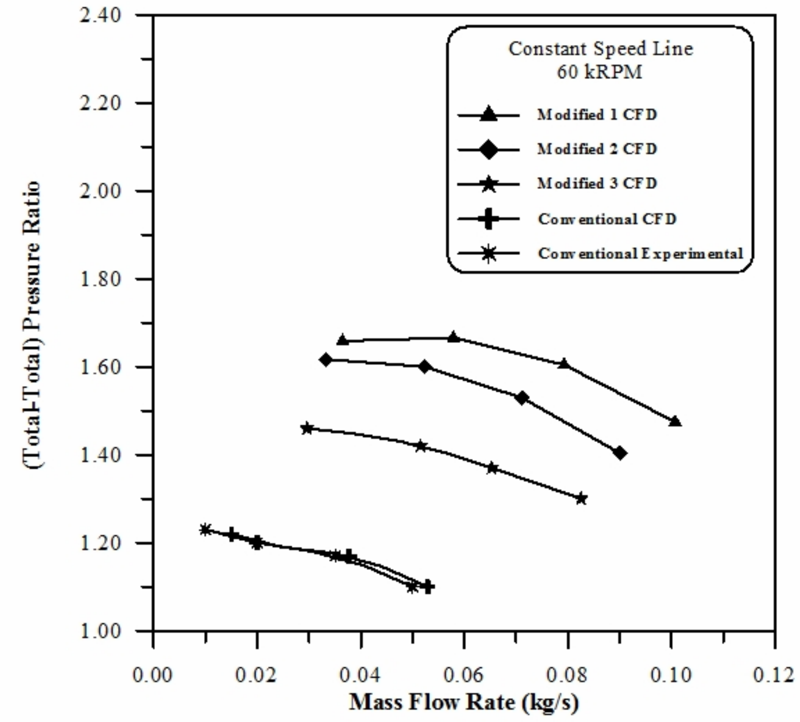


Figure 6: (T-T) Pressure Ratio with Mass Flow Rate for Impeller configurations

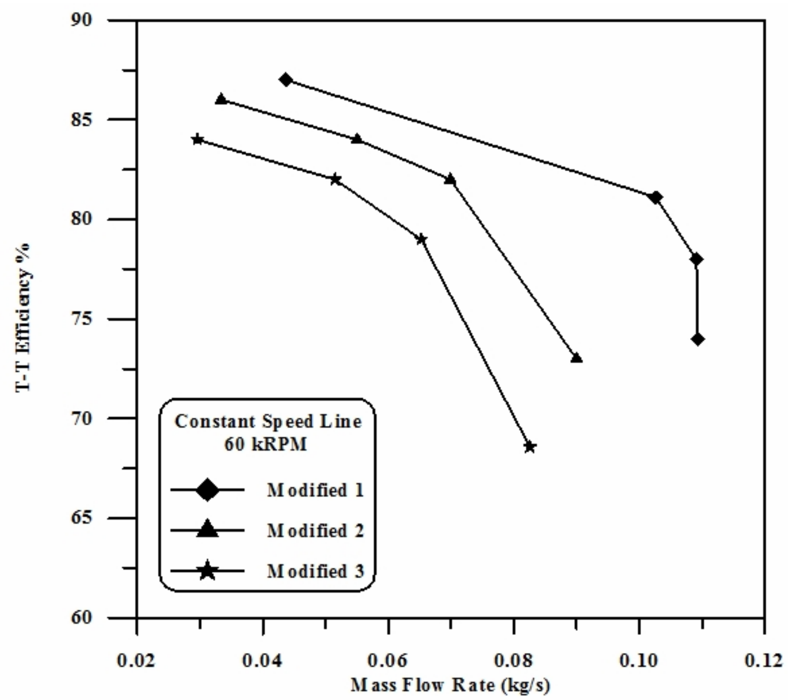
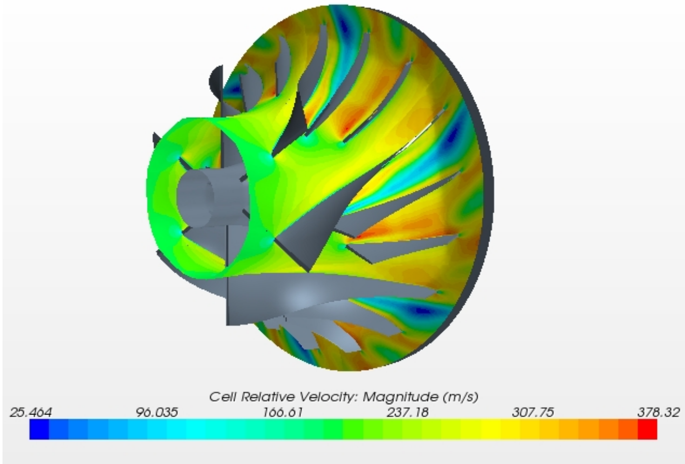
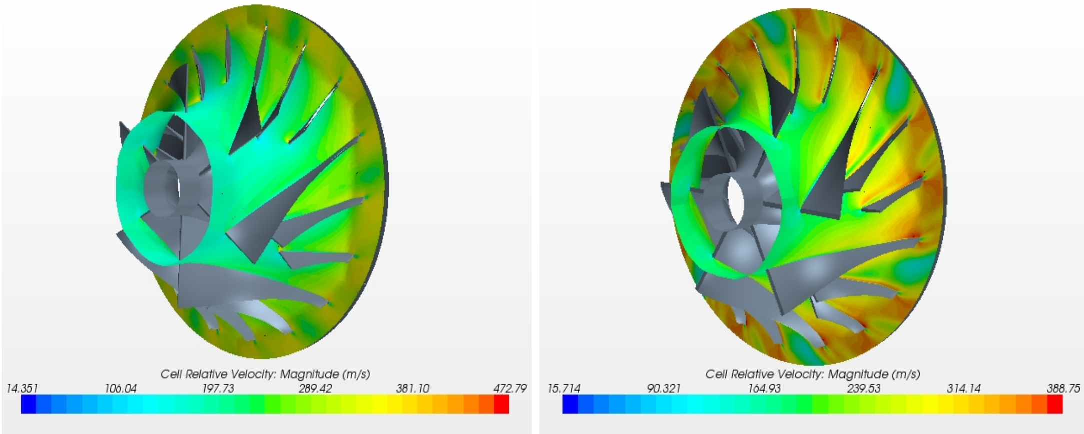


Figure 7: (T-T) Efficiency with Mass Flow Rate for Different Impeller configurations

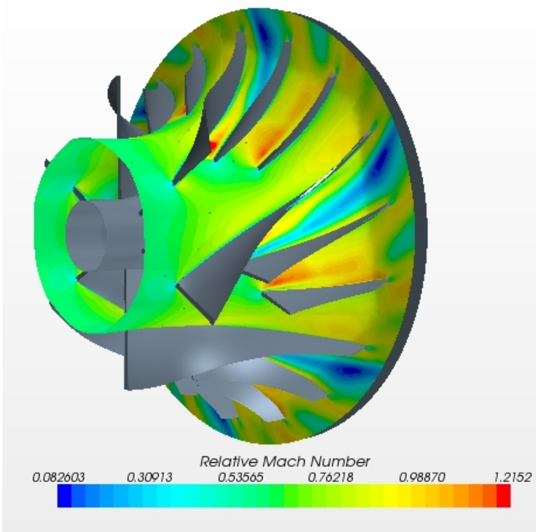


a. Modified 1

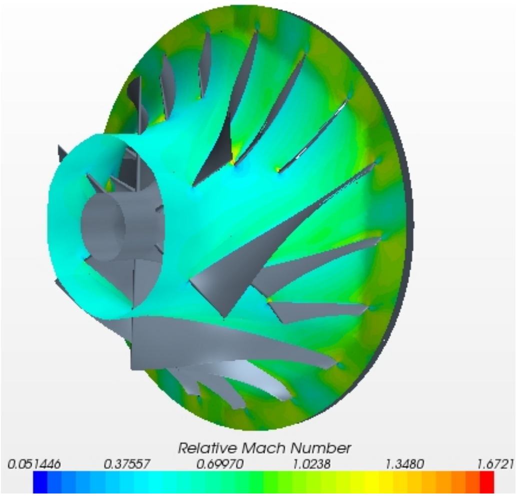


b. Modified 2 c. Modified 3

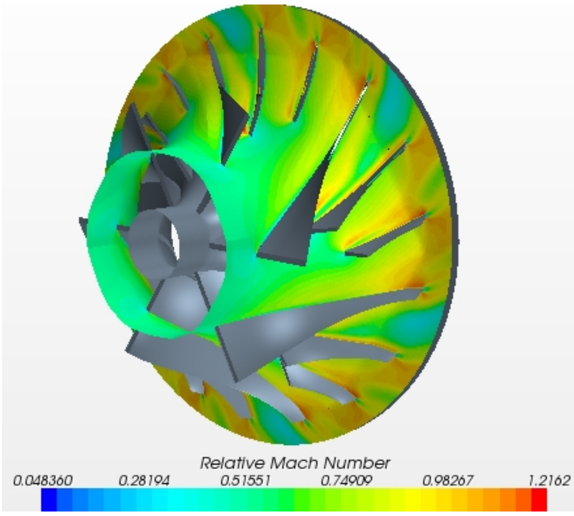
Figure 8: Contour of Cell Relative velocity for Different Impeller Configurations



a. Modified 1



b. Modified 2



c. Modified 3

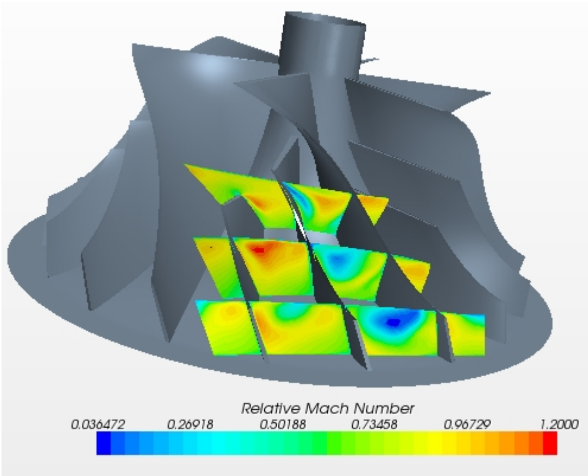
Figure 9: Relative Mach number of Different Modified Types.

Figure 10 shows the relative Mach number for different streamwise layers from (Leading Edge-Trailing Edge) of the modified compressor impeller configurations. The high Mach due to the high outlet velocity therefore leads to choking 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.

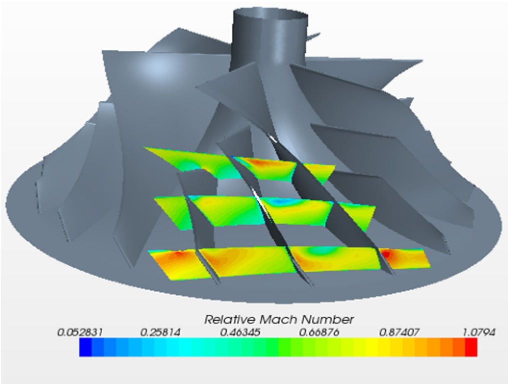
The numerical analysis was carried out including just the impeller flow passage. As we mentioned earlier that the flow is heavily affected by the impeller design, so the results show the effect of second splitter and impeller trimming on the performance of a compressor to increase the mass flow rate and pressure ratio. 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 the turbocharger centrifugal compressor and enable us to solve many problems easier and much faster.

5. Conclusion

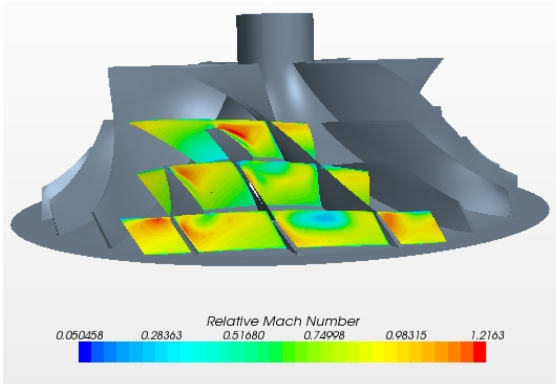
Steady state CFD simulations have been conducted in order to study the effect the impeller trimming on the compressor performance. The Parametric computations were performed on a three dimensional Turbulent CFD to obtain the performance of backswept impeller configurations at a specific speed. The analysis of the flow characteristics was also performed to obtain a better understanding of the blade to blade compressor behaviours. The results also show the potential of a trim of impeller and double splitter to improve the performance of a centrifugal compressor and also increase the outlet mass flow rate and pressure ratio. The relative velocities flowing out of the impeller were quite low, but with a high impeller tip speed, the relative velocity leaving the impeller could reach values above Mach one. Obviously the performance was heavily affected by the impeller design because of the nature of the flow in the impeller.



a. Modified 1



b. Modified 2



c. Modified 3

Figure 10: Relative Mach number of Different Streamwise layers for Different Modified Types.

References

[1] Y. Yang, RongXie, Lu-yuan Gong and Yang Hai (2011). 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 (2009). On the effect of pulsating flow on surge margin of small centrifugal compressors for automotive engines. *Experimental Thermal and Fluid Science, 33:1163–1171.*

[3] O. Baris (2011). Automotive turbocharger compressor CFD and extension towards incorporating installation effects. *Proceedings of ASME Turbo Expo 2011: Power for Land, Sea and Air GT2011.*

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

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

[6] W. Jiang, Jamil Khan, and Roger A. Dougal (2006). Dynamic centrifugal compressor model for system simulation. *Journal of Power Sources 158 (2006) 1333-1343.*

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

[8] A. Engeda (2007). 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.*

[9] H. Tamaki (2008). Effect of piping systems on surge in centrifugal compressors. *Journal of Mechanical Science and Technology 22 (2008) 1857-1863.*

[10] 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 52 (2010) 1288_1297.*

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

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

[13] R. Shook, W. Oakes, 1994. "The Aerodynamic Performance of a High Speed Research Centrifugal Compressor Facility*," AIAA Paper 1994-2798, pp. 1-9.*

[14] J. Gravdahl and F. Willems, 2004. "Modelling of Surge in Free-Spool Centrifugal Compressors: Experimental Validation," *Journal of Propulsion and Power, Vol. 20, pp. 849-857.*

[15] M. Peric, (2004). Flow simulation using control volumes of arbitrary polyhedral shape. *ERCOFTAC Bulletin 62.*

[16] F. Mendonça, Clement J, Palfreyman D and Peck A, (2008). Validation of Unstructured CFD Modelling Applied to the Conjugate Heat Transfer in Turbine Blade Cooling. *ETC_8-198, European Turbomachinery Conference, Graz.*

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