ISSN 1996-3343

Asian Journal of Applied Sciences

http://knowledgiascientific.com

Knowledgia SCIENTIFIC
A Place to Publish Outstanding Research

Asian Journal of Applied Sciences 5 (8): 563-572, 2012
ISSN 1996-3343 / DOI: 10.3923/ajaps.2012.563.572

Numerical Simulation of Flow Inside a Modified Turbocharger Centrifugal Compressor

Layth H. Jawad, S. Abdullah, R. Zulkifli and W.M.F.W. Mahmood
Department of Mechanical and Materials Engineering, Faculty of Engineering and Built Environment, The National University of Malaysia, 43600 Bangi, Selangor, Malaysia

Corresponding Author: Layth H. Jawad, Department of Mechanical and Materials Engineering, Faculty of Engineering and Built Environment, The National University of Malaysia, 43600 Bangi, Selangor, Malaysia

ABSTRACT

In order to increase the performance of the conventional turbocharger compressor's to boost the pressure into engine. The flow inside a small centrifugal compressor with both single and modified 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 double splitters on the performance of a turbocharger compressor. The simulation was undertaken using Computational Fluid Dynamics (CFD) analyses on aerodynamic flow field and to predict the mass flow rate and aerodynamic characteristics. The mesh generator of CFD Code was used to generate a polyhedron mesh. According to the results of the simulation, the outlet mass flow rate for a specific speed increased for the compressor with double splitters and all the performance characteristics also increased. It was observed that the turbocharger compressor performance was heavily affected by the impeller design.

Key words: Turbocharger, centrifugal compressor, numerical simulation, flow characteristics, CFD simulation

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. A turbocharger compressor is an effective device for improving the performance of diesel engines. The conventional turbocharger compressor is constrained by surge or choke conditions. The improvement of centrifugal compressor performance and the enlargement of the stable operating ranges are becoming crucial 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 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 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 (Yang *et al*., 2011). 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 (Galindo *et al*., 2009). The application of CFD to turbocharger compressor characteristic predictions over a range of speeds between 100,000 and 200,000 rpm, to develop an efficient methodology for analysing the turbocharger compressor performance, Also to compare the computation versus rig measurements (Baris, 2011). 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 reduction in compressor mass flow, the results show that there is a distinct stall frequency at the given compressor speed (Guo *et al*., 2007). 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 (Jiang *et al*., 2006). 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 (Galindo *et al*., 2006).

The contribution to the design methodology and performance assessment of LSVD (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 Engeda (2003). 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 Engeda (2007).

The effect of the piping systems on the surge characteristics to the design of the compressor 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 Tamaki (2008). 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. To downsize the engine displacement also to increase the power output and reducing fuel consumption (Galindo *et al*., 2010). 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 (Cukurel *et al*., 2010; Higashimori *et al*., 2004). 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 flow inside blade to blade passage is very affected by the throat area at the inlet of the impellers, so in the conventional type there is splitter blade with full blade to avoid the blockage of flow. The hypothesis in this work we proposed to use a second splitter with the impellers but the splitters were not continued to the leading edge due to if the splitters continued to the leading edge, it would produce sufficient blockage to cause choking at high speed. To achieve further performance improvement of the turbocharger compressor, by boosting the outlet pressure and mass flow rate.

MATERIALS AND METHODS

The turbocharger compressor studied was a centrifugal compressor model GT1749V Trim55. 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 are given in Table 1.

Table 1: Geometric features of a turbocharger centrifugal compressor

Turbocharger compressor	GT1749V Trim 55 (Conventional) single splitter	Modified(doublesplitter)
Axial width of impeller in meridional view (mm)	39.257	39.257
Inner diameter at compressor inlet (mm)	12	12
Outer diameter at compressor inlet (mm)	36.25	36.25
Impeller outer diameter (mm)	49	49
Impeller width at trailing edge (mm)	5	5
Diffuser exit diameter (mm)	55	55
Number of blades	12	18
Number of splitters	6	12
Throat length at leading edge (mm)		
Impeller	9.760	9.760
Splitter 1	15.963	13.505
Splitter 2	-	15.273
Angular offset (pitch fraction or location)		
Splitter 1	0.5	0.667
Splitter 2	-	0.333
Leading edge cut off of splitter 1 (%)		
Hub	35	25
Shroud	32	20
Leading edge cut off of splitter 2 (%)		
Hub	-	45
Shroud	-	42

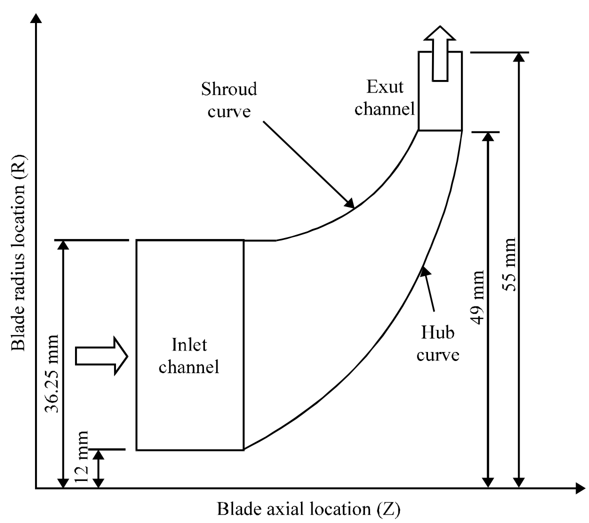

Fig. 1: Meridional plane view of a full impeller blade

Figure 2 shows a conventional turbocharger centrifugal compressor wheel comprising of six main impeller blades and six splitter blades and Fig. 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 geometry as shown in Fig. 4. All the surface geometry, inlet, exit and periodic boundaries, were defined via Computer-aided Design (CAD) as Initial Graphics Exchange Specification (IGES) parts.

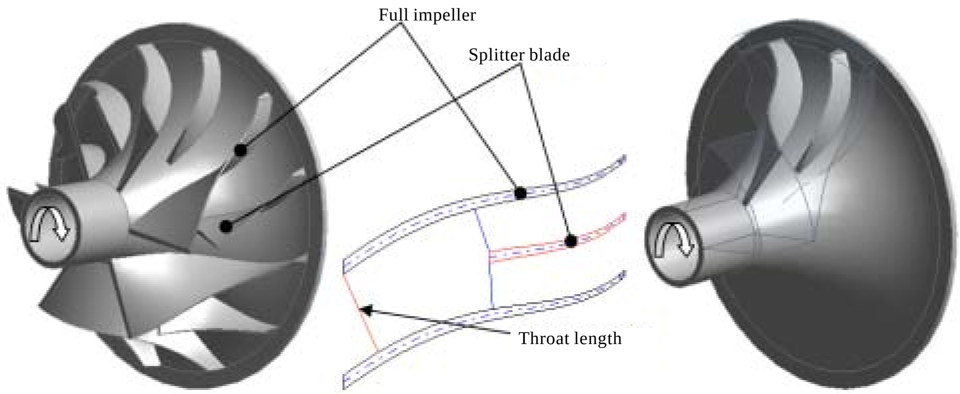

Fig. 2: Conventional turbocharger centrifugal compressor

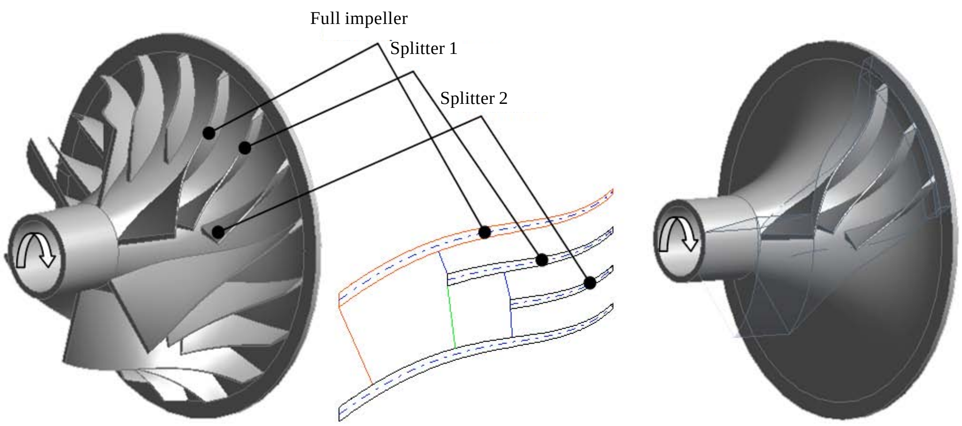

Fig. 3: Modified turbocharger centrifugal compressor

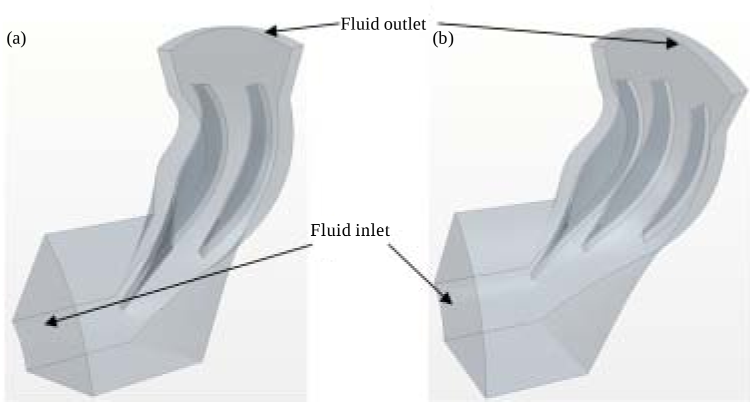

Fig. 4(a-b): CFD geometry for computations of (a) Conventional design and (b) Modified design

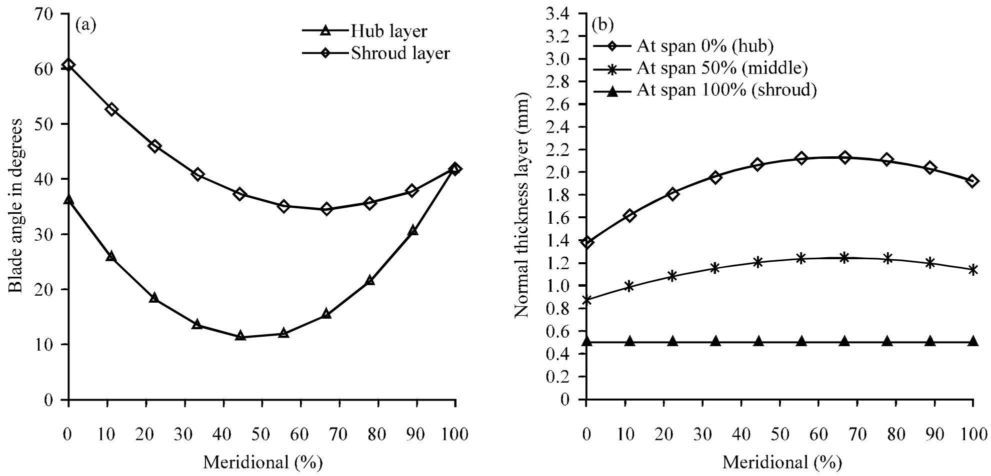

Fig. 5(a-b): (a) Blade angle (axile) distribution and (b) Normal thickness of the blades with respect to the meridional plane

Figure 5 shows the blade angle (from axial) distribution for the hub and shroud layers with respect to the meridional plane and the normal thickness of the blade distribution for different layers.

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 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 (Peric, 2004; Mendonca *et al.*, 2008). 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 (Peric, 2004).

Figure 6 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.

Fluid flow modelling: STAR CCM+(2011) uses a compressible implicit coupled algorithm. 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.

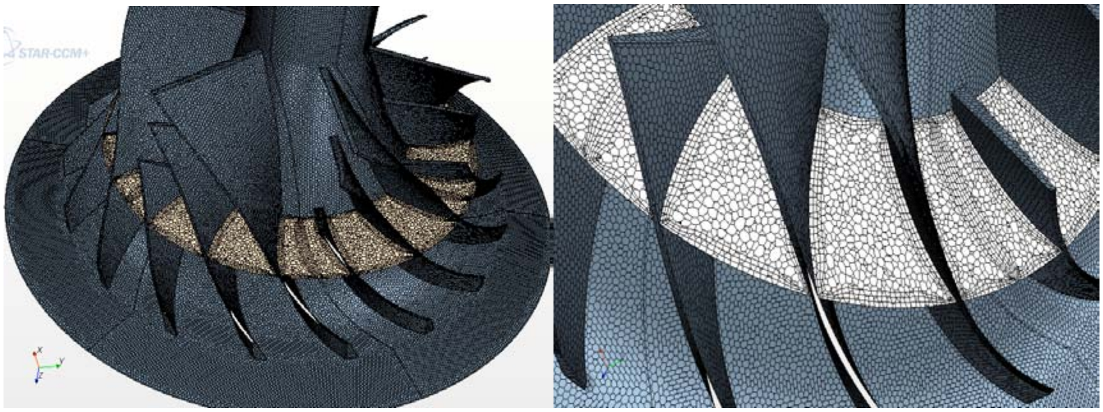

Fig. 6: Volume polyhedral mesh

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 601,292 cells, which at a specific speed takes approximately four hours to run.

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 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 rate (0.0825308 kg sec^{-1}) and the pressure ratio (1.30) of the modified design is higher than the mass flow rate (0.05291 kg sec^{-1}) and 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 Fig. 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 rate and to the measurements (Galindo *et al*., 2009).

Figure 8 shows the cell relative velocity magnitude for different streamwise layers from the leading edge to the trailing edge of the blades of the conventional and modified compressor impeller. 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 conventional and 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 vanes diffuser is close to Mach 1 which means the space area ratio it is a very important factor to modify in order to remove any choking of the flow.

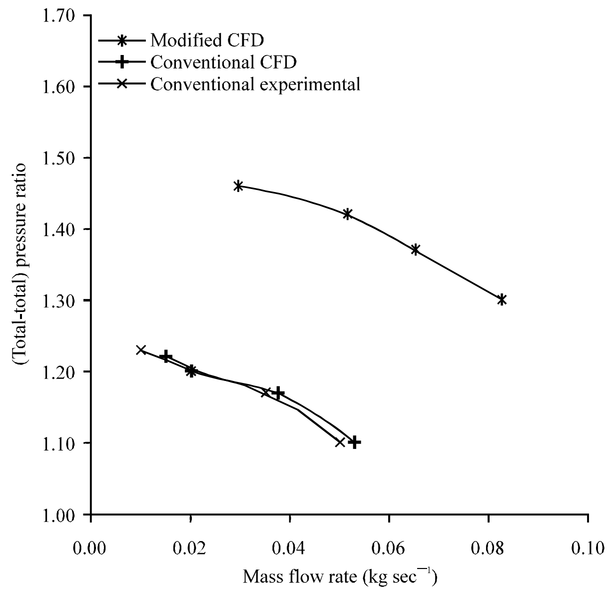

Fig. 7: The relationship between the mass flow rate and (T-T) pressure ratio for modified and conventional type

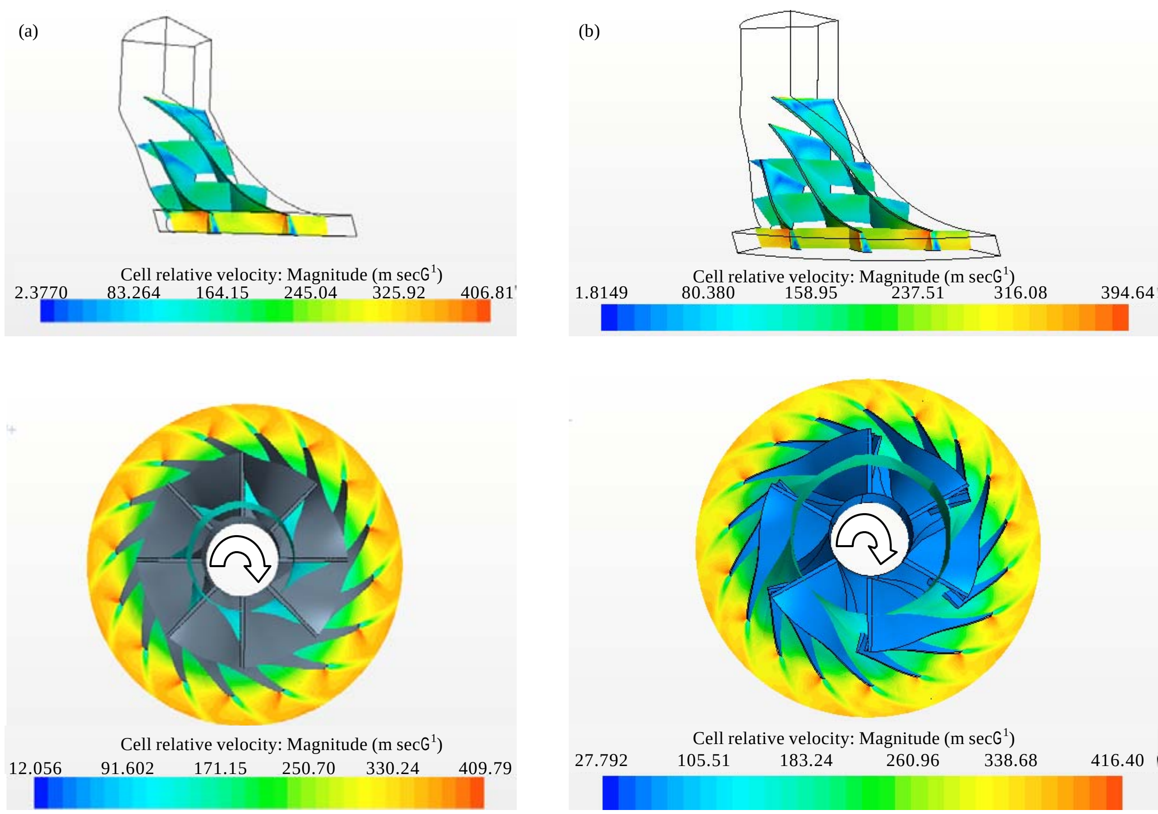

Fig. 8(a-b): Cell relative velocity (a) Conventional and (b) Modified design

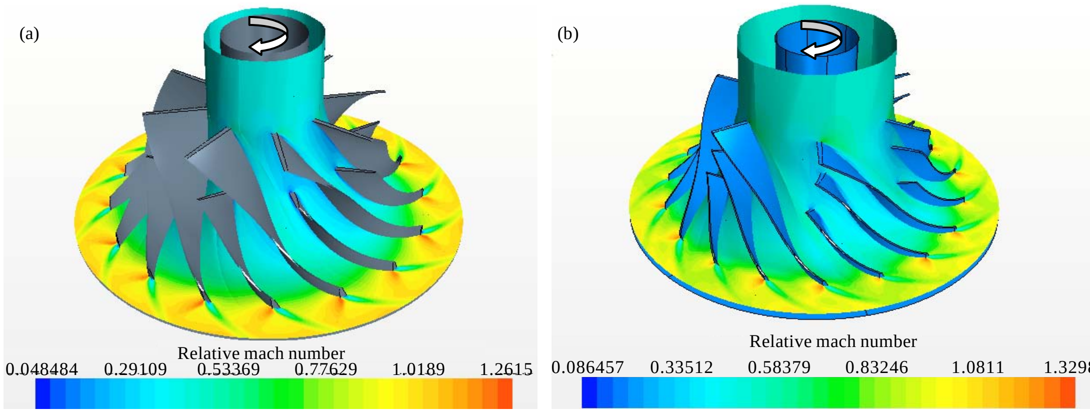

Fig. 9(a-b): Relative mach number for (a) Conventional and (b) Modified design

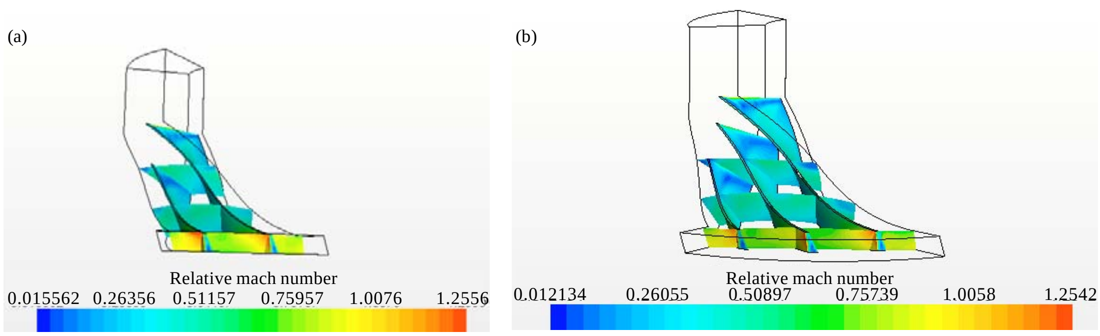

Fig. 10(a-b): Relative mach number for different streamwise layers for (a) Conventional and (b) Modified design

Figure 10 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 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.

Figure 11 shows the relative velocity vector at the mid span directed from the upstream point to downstream in the impeller duct. The fluid flow is high quality without apparent backflow or vortex. In the impeller throat area, speeds gradually increase from the inlet to the outlet.

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 on the performance of compressor to increase the mass flow rate and pressure at the outlet. Modification of a previous design of centrifugal compressor impellers gives

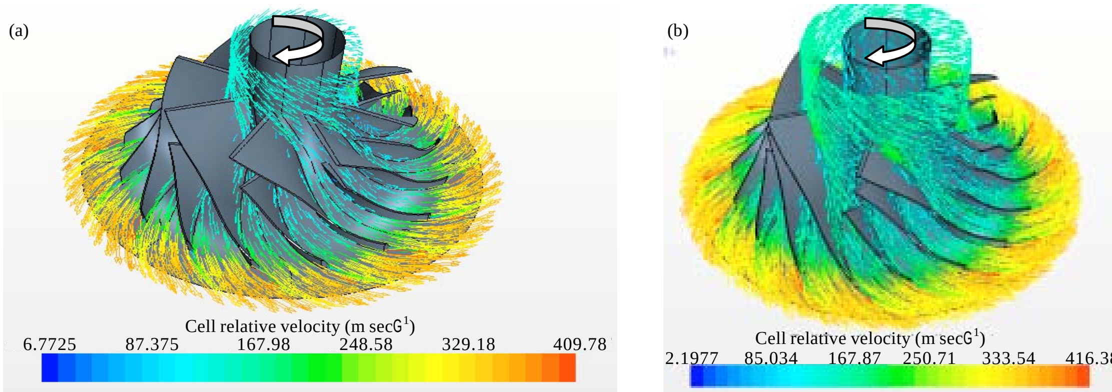

Fig. 11(a-b): Cell relative velocity vectors at mid span 50% for (a) Conventional and (b) Modified compressor at 60000 rpm

a better performance or a wide operating range. CFD models give a much deeper understanding of the flow inside turbocharger centrifugal compressor and enable us to solve many problems easier and much faster.

CONCLUSIONS

Steady state CFD simulations have been conducted in order to study the characteristics of the flow inside a modified turbocharger compressor. The parametric computations were performed on a 3D-Turbulent CFD to obtain the performance of a turbocharger backswept impeller at high speed (60,000) rpm. 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 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 low, but with a high impeller tip speed (60,000) rpm, the relative velocity leaving the impeller could reach values above Mach 1. Obviously the performance was heavily affected by the impeller design because of the nature of the flow in the impeller.

REFERENCES

Baris, O., 2011. Automotive turbocharger compressor CFD and extension towards incorporating installation effects. Proceedings of the Power for Land, Sea and Air, June 6-10, 2011, British Columbia, Canada.

Cukurel, B., P.B. Lawless and S. Fleeter, 2010. Particle image velocity investigation of a high speed centrifugal compressor diffuser, spanwise and loading variations. J. Turbomachinery, 132: 1-9.

Engeda, A., 2003. Experimental and numerical investigation of the performance of a 240 kW centrifugal compressor with different diffusers. Exp. Thermal Fluid Sci., 28: 55-72.

Engeda, A., 2007. Effect of impeller exit width trimming on compressor performance. Proceedings of the 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, July 2007, Lyon, France.

Galindo, J., F.J. Arnau, A. Tiseira and P. Piqueras, 2010. Solution of the turbocompressor boundary condition for one-dimensional gas-dynamic codes. Mathe. Comput. Modelling, 52: 1288-1297.

Galindo, J., J.R. Serrano, C. Guardiola and C. Cervello, 2006. Surge limit definition in a specific test bench for the characterization of automotive turbochargers. Therm. Fluid Sci., 30: 449-462.

Galindo, J., H. Climent, C. Guardiol and A. Tiseira, 2009. On the effect of pulsating flow on surge margin of small centrifugal compressors for automotive engines. Exp. Thermal Fluid Sci., 33: 1163-1171.

Guo, Q., H. Chen, X.C. Zhu, Z.H. Du and Y. Zhao, 2007. Numerical simulations of stall inside a centrifugal compressor. Power Energy, 221: 683-693.

Higashimori, H., 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. J. Turbomachinery, 126: 473-481.

Jiang, W., J. Khan and R.A. Dougal, 2006. Dynamic centrifugal compressor model for system simulation. J. Power Sources, 158: 1333-1343.

Mendonca, F., J. Clement, D. Palfreyman and A. Peck, 2008. Validation of unstructured CFD modelling applied to the conjugate heat transfer in turbine blade cooling. Proceedings of the 8th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, January 2008, Graz, Austria.

Peric, M., 2004. Flow simulation using control volumes of arbitrary polyhedral shape. ERCOFTAC Bull., 62: 25-29.

Tamaki, H., 2008. Effect of piping systems on surge in centrifugal compressors. J. Mechanical Sci. Technol., 22: 1857-1863.

Yang, Y., R. Xie, L.Y. Gong and Y. Hai, 2011. Study of influence of diffuser meridian channel shape on performance of micro-gas turbine centrifugal compressor. Proceedings of the Power Energy Engineering Conference, March 25-28, 2011, Wuhan, China.