Numerical Investigation of Performance Improvement of a Modified Centrifugal Compressor with second Splitter and Vaned Diffuser

تحقيق عددي لتحسين أداء ضاغط الطرد المركزي المطور مع تقسيم ثاني وناشر ذي ريش

Layth H. Jawad[1], Mahir H. Majeed[1] and Boraire Ahmed[1]
[1]*Department of Mechanical Techniques,*
Technical Institute of Karbala,
Al-Furat Al-Awsat Technical University, IRAQ
Email:laihasan@yahoo.com

Abstract

The performance of a centrifugal compressor for the turbocharger device is heavily affected by impeller design due to the flow dynamics in the impeller. Furthermore, modification and improvement of a centrifugal impeller is a challenging task for turbomachinery engineers . Hence, this study is aimed to further increase the compressor performance by introducing extra splitters in the impeller design. A series of impellers, having vaned diffuser stage was used for the performance evaluation of different configurations. A commercial CFD code, was used to simulate the flow and the primary grid generation. The influence of vaned diffuser meridional outlet section with a different width ratio of the modified centrifugal compressor has been studied by comparing the compressor stage and impeller performance at different width ratio. The comparative study of these results indicated noticeable compressor stage performance differences between the conventional and modified types. The results showed that the fluid dynamics within the modified compressor has indicated that the pressure ratio and volume flow rate were increased. Obviously, it was observed that the efficiency was increased and both the total pressure ratio and static pressure for 0.5 width ratio are increased. Overall, the numerical results obtained from CFD simulation could produce a highly reliable compared with experimental results for estimation on the performance and efficiency of the modified centrifugal compressor.

الخلاصة

أداء ضاغط الطرد المركزي لجهاز الشاحن التوربيني يتأثر بشدة بتصميم المكرة بسبب ديناميكية الجريان في المكرة. وعلاوة على ذلك، تطوير وتحسين مكرة الطرد المركزي مهمة صعبة لمهندسين التوربوماشينيري. ولهذا ، الهدف من هذه الدراسة هو لزيادة أداء الضاغط عن طريق إدخال تقسيم إضافي في تصميم المكرة. تم استخدام سلسلة من طور المكرات مع ناشر ذي ريش لتقييم الأداء لانواع مختلفة. كود CFD التجاري، استخدم لمحاكاة الجريان وتوليد الشبكة الرئيسية. وقد تمت دراسة تأثير مقطع مخرج الناشر الطولي لنسبة عرض مختلفة لضاغط الطرد المركزي المطور ومقارنة طور الضاغط وأداء المكرة لنسبة العرض المختلفة. مقارنة نتائج هذه الدراسة أشارت بشكل ملحوظ إلى فروقات اداء طور الضاغط بين النوع التقليدي والمطور. وأظهرت النتائج أن ديناميكية الجريان داخل الضاغط المطور قد أشارت إلى أن نسبة الانضغاط ومعدل حجم التدفق كانت متزايدة. ومن الواضح، لوحظ أن الكفاءة كانت متزايدة وكل من نسبة الانضغاط الكلي والضغط الاستاتيكي لنسبة العرض 0.5 كانت متزايدة. بشكل عام، النتائج العددية التي تم الحصول عليها من محاكاة CFD يمكن ان تقدم درجة عالية من الموثوقية مقارنة مع النتائج العملية لتقدير الأداء وكفاءة ضاغط الطرد المركزي المطور.

Keywords: Centrifugal Compressor; Vaned Diffuser; CFD.

Nomenclature

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
W_1 Inlet Width Diffuser
W_2 Outlet Width Diffuser

1. Introduction

Nowadays, the interest has more increasingly been devoted to the development of turbocharger because of their compact size, large capacity, high performance, and ability to improve volumetric efficiency. The turbocharger is 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 on a line with a constant number of revolutions.

Moreover, 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 40-67 Hz (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 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 increases with the 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 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 the boundary was used as the Method of Characteristics to determine the flow conditions at compressor inlet and outlet. To downsize the engine displacement to increase the power output and to reduce fuel consumption [10 and 26]. The complex shock waves within the diffuser throat and impeller inlet, respectively, within high-speed compressors. This flow phenomenon does not occur in low speed compressors and are very significant in the design of these compressors [11 and 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 [13, 24 and 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 behaviour 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 investigations on the interaction of vaned diffuser with modified centrifugal compressor impeller where CFD simulations and flow behaviour of each conventional and modified centrifugal compressor impeller with vaned diffuser configurations were performed. In this paper, the study is focused on the effect of impeller modification with different meridional passage width ratios of vaned diffuser on the compressor performance to achieve high quality flow and further performance improvement of the turbocharger compressor.

2. Specifications & Design

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 Figure 1. under the same geometric conditions except for the meridional passage width,keeping the diffuser meridional passage inlet W_1 unchanged,changing the outlet width $W2$ selecting, a different width ratio of the diffuser meridional passage W_2/W_1=0. 5, 0.7, 0.8, 1 and 1.2 respectively. The main geometry features and dimensions of the modified compressor vaned diffuser and conventional compressor are given in Table 1.
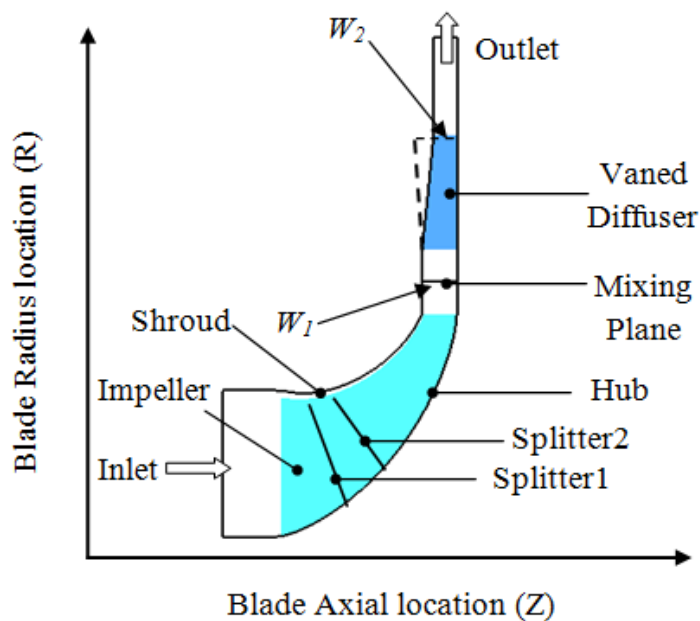

Figure 1: Impeller and Vaned Diffuser on the Meridional View

Table 1: Centrifugal Compressor and Vaned Diffuser Features.

Centrifugal Compressor	Conventional Type	Modified Type
Impeller Axial Width	39.25 mm	30 mm
Compressor Inner Radius at Inlet	12 mm	8 mm
Compressor Outer Radius at Inlet	36.25 mm	35 mm
Outer Radius of Impeller	49 mm	49 mm
Width of Impeller at Trailing Edge	6 mm	6 mm
Blades Number	12	18
Splitters Number	6	12
Shroud Tip Clearance (% span)	95%	95%
Offset of Splitter1 angular	50%	67%
Offset of Splitter2 angular offset	/	33%
Diffuser Vanes Number	/	19
Inner Radius of Vane	/	61 mm
Outer Radius of Vane	/	82 mm
Inner Axial Width of Vane	/	6 mm
Outer Axial Width of Vane	/	4 mm
Outer Flow angle of Vane	/	30°
Inlet Width Diffuser (W_1)	/	6mm

Figure 2 shows a conventional centrifugal compressor wheel comprising of 6 main impeller blades and 6 splitter blades, Figure 3 shows the geometry of the modified compressor wheel vaned diffuser stage comprising of six main impeller blades, 12 splitter blades and 19 diffuser vanes. The CFD computations for the 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.
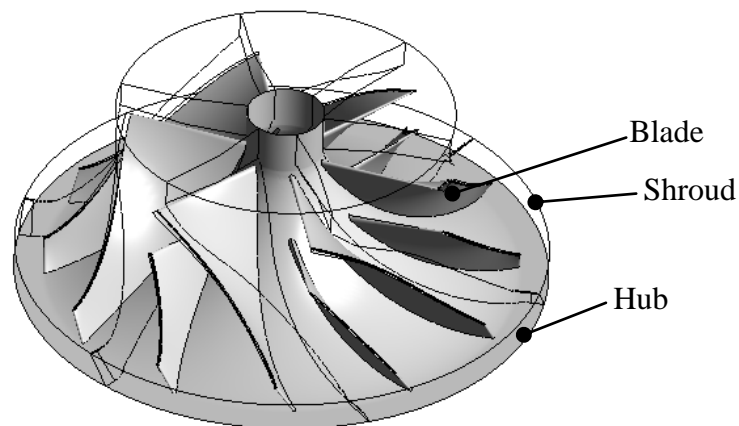

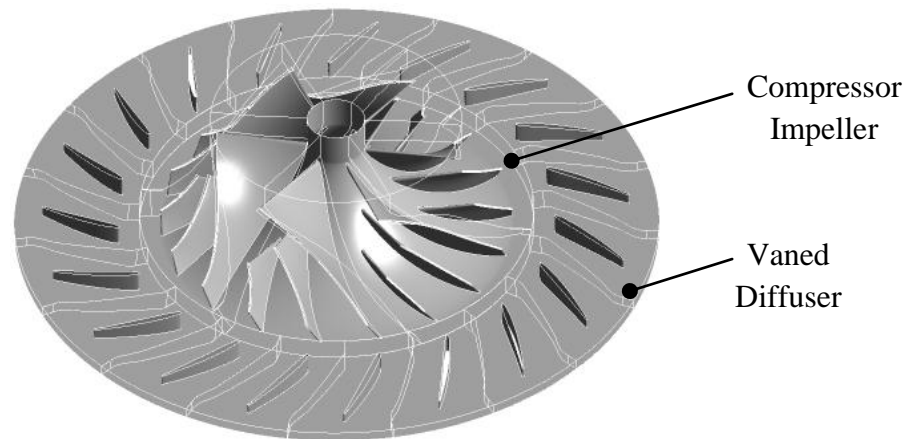Figure 2: Three Dimensional View of Conventional Centrifugal Compressor

Figure 3: Three Dimensional View of Centrifugal Compressor Stage

3. CFD Methodology
3.1 Grid Generation

A commercial CFD code 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 a 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 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 finite element 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 splitters 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 fine grid elements were created in the impeller tip clearance region, around the impeller, and at the hub and shroud walls. Sufficient mesh quality checks were performed by keeping the parameters like mesh angle and determinants within acceptable limits.

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.

3.2 Modelling of Fluid Flow

The Parametric computations were performed on a three dimensional-Turbulent CFD to obtain the performance of backswept impeller configurations. 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). 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 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. Experimental Facility

The facility used in this work is a turbocharger test bench presented in Figure 4. A Mitsubishi Pajero turbo diesel engine is used for feeding the turbocharger's turbine. An eddy current dynamometer loads the engine and allows controlling the engine speed. This set is used just as an engine-like pulsating hot flow generator able to lead the turbocharger to any operating condition. In the compressor side a control valve is installed. The testing procedure consists in the measurement of compressor constant speed lines from maximum flow up to surge by closing the controlling valve. In order to properly define the exact moment and conditions where the compressor begins to surge, the methodology proposed is followed: compressor's outlet pressure spectrum is computed and analysed in the low frequency band, where peaks associated to surge pulsations are expected to appear. An additional, externally controlled valve (CV) permits controlling pulsation amplitude and mean flow through the compressor.

Measurement procedure for locating surge line in pulsating conditions is as follows: manual valves are set fixed at a given position according to the required pulsation level, while the controlling valve is set fully open; once the air-stand is started and the turbine is set to attain the desired compressor speed. The rotating disk is switched on and the controlling valve is progressively closed in order to arrive to an operating point close to surge line (which has been previously determined in the steady tests). At this moment the recording system is triggered and the controlling valve is slowly and progressively closed up until deep surge is developed. The turbocharger used for the experimentation was a Mitsubishi Turbo Technologies model TD04-10T 4 turbocharger. Main instrumentation is summarized in Figure 4.14. In the compressor circuit, a hot-film flow meter , range 0-700 kg/s, with 0.9% of measured error and 0.5 s of time response, two pressure transducers at the compressor inlet and outlet (absolute pressure range 300 psig), and 2 K-type thermocouples (range 20-1300 C) were used. Turbocharger speed was measured by means of an eddy current sensor (range 0-400k RPM).
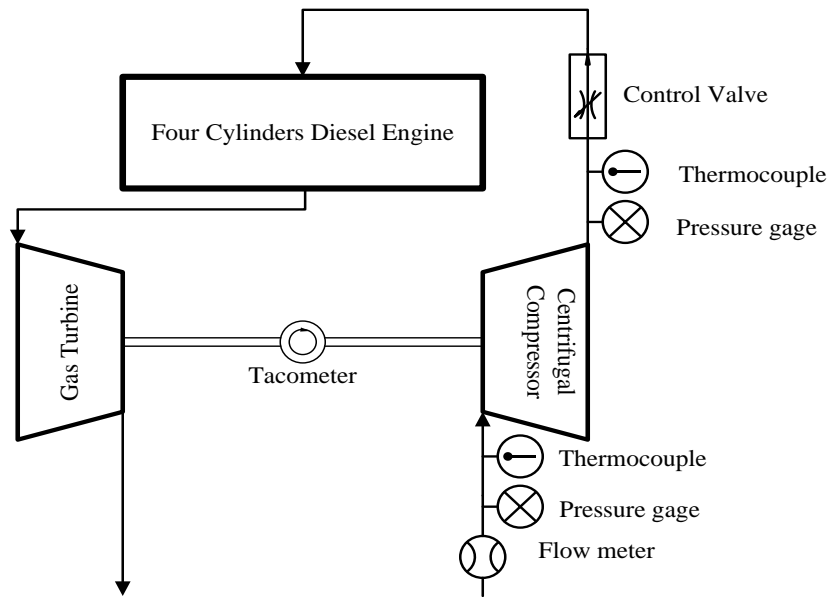
Figure 4: Turbocharger Test Bench

5. Results
5.1 Effect of the Impeller Modification

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. Figure 5 shows the impeller pressure ratio of the modified centrifugal compressor plotted against the mass flow rate. The mass flow plot shows that for the operating range studies here, choke does not occur in the impeller. This is a characteristic of the design of the particular centrifugal compressor studied. Only at high speed case seems to be approaching choke. Surge and flow reversals on the other hand appears to occur in the impeller, because the speed line characteristics show the typical constant pressure ratio at a low mass flow rate characteristic that symbolic of approaching the surge region. These lines can be used to determine the operating line based on the mass flow rate desired and the surge margin required. It is clearly seen that the performance of the modified type was better than the performance of the conventional type, the reasons behind that the range of pressure ratio was increased in the same conditions.
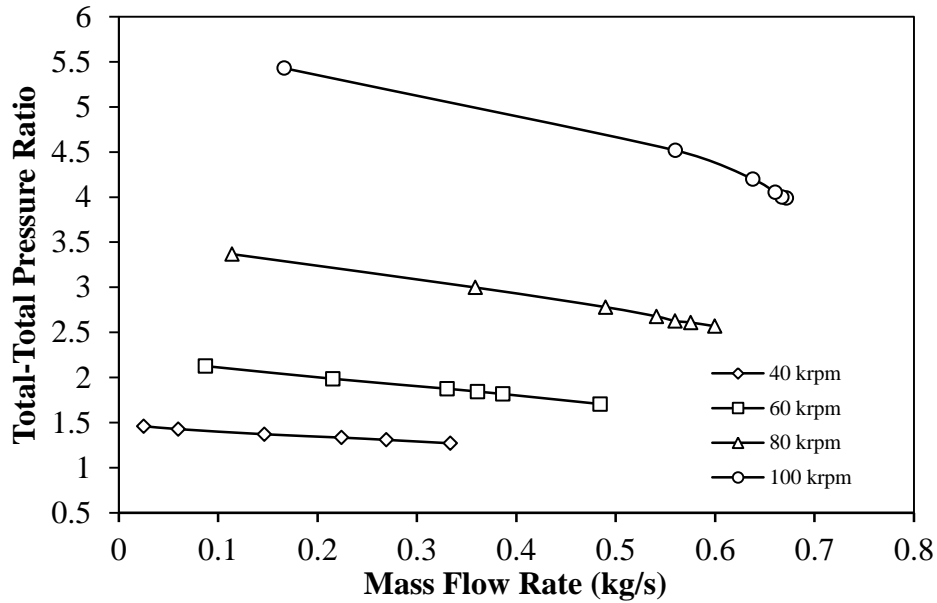
Figure 5: Total-Total Pressure Ratios Versus Mass Flow Rate for Modified Type

The results are compared with experimental work data which was presented for validation. Figure 6 shows the predicted compressor characteristic at the four speeds investigated versus the rig measurements. The calculations at low pressure ratio points for all speeds correspond very closely in predicting mass-flow to the measurements. At the high speed line, the 60,000 and 80,000 RPM predictions are in good agreement with the experiments, whereas at the highest speed, 100,000 RPM deviation from the measurements of 3-5% is observed. At the higher pressure ratios, and especially near the surge line, the deviation is up to 4.5% over-prediction at any given speed.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 figure 6 .
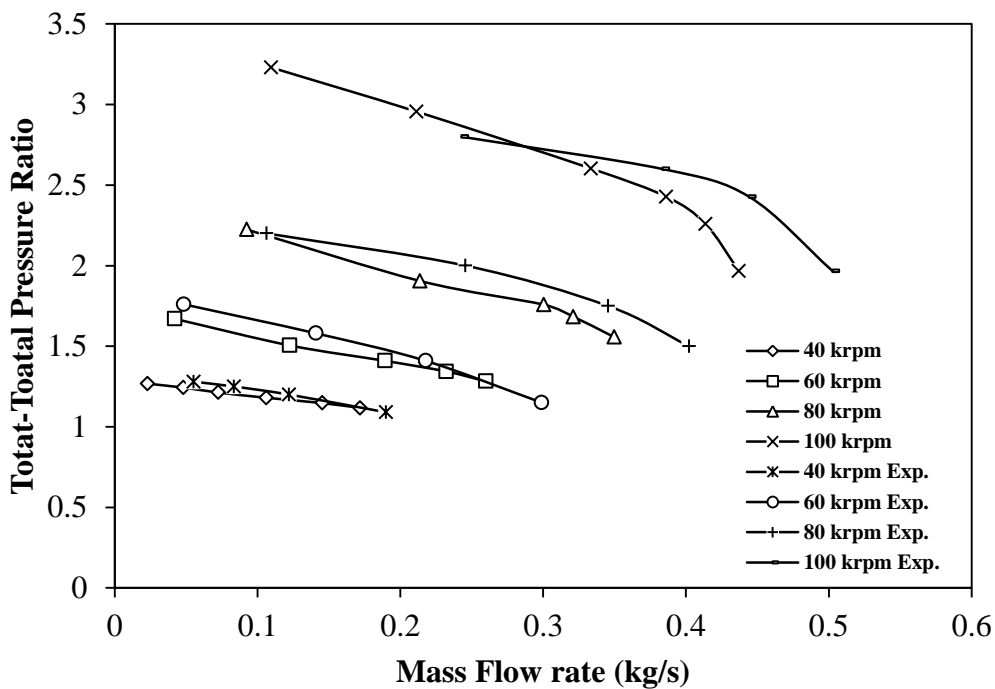

Figure 6: Comparison of Experimental With Numerical Work for Conventional Type

The variation of the head and work coefficents are shown in Figures 7 and 8 respectivly . Looking at the head coefficient increases with the decrease the flow coefficient. While it is clearly seen that the head coefficient line for modified type is higher than conventional type. Figure 8 shows that the work coefficient is higher for the modified type; it then goes down when the flow coefficient increases. This true for most of the operation range except for a relatively high flow coefficient where the work coefficient decreases more rapidly for higher rotational speed.
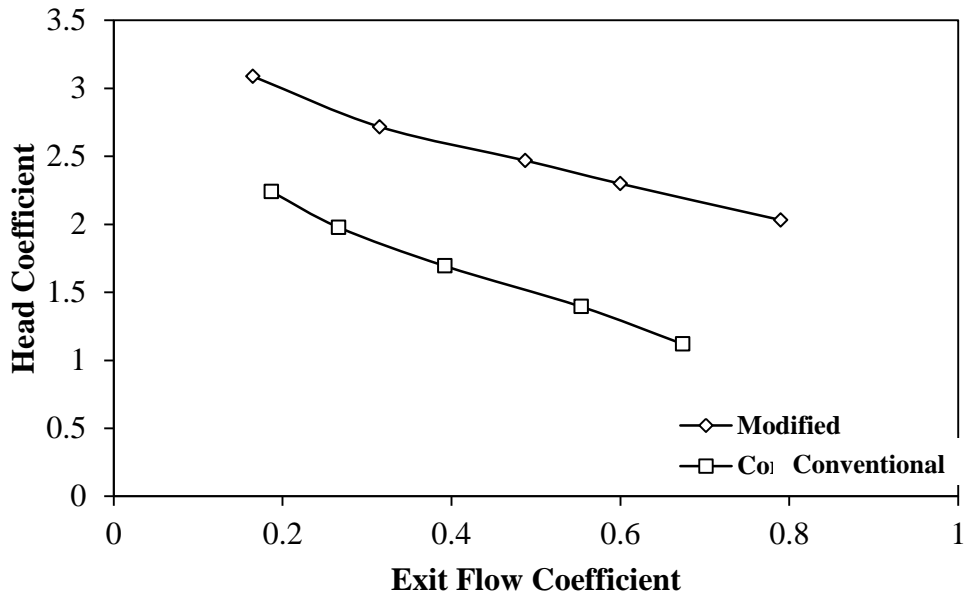
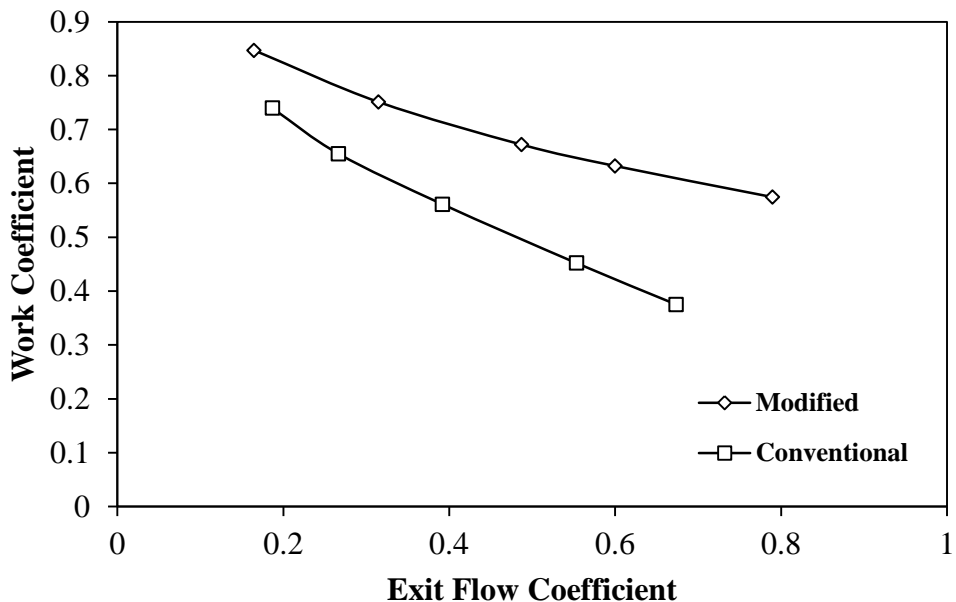Figure 7: Head Coefficient With Respect to Exit Flow Coefficient

Figure 8: Work Coefficient With Respect to Exit Flow Coefficient

5.2 Effect of the vaned diffuser

The effect of the vaned diffuser is to shorten the flow path compared to the long flow path in the vaneless diffuser. Also, the vanes can be used to direct the flow more in the radial direction, to increase the rate of diffusion, and to decrease the losses. However, figure 9 shows the total-total Polytropic efficiency and total-total pressure ratio with respect to width ratio of the vaned diffuer, considering the best performance was at 0.5 width ratio, and the worst one was at 1.2 width ratio. Moreover, it is clearly seen that the width ratio does have an important impact on the efficiency and pressure ratio. The reason might be due to its own characteristics of this type of centrifugal compressor.
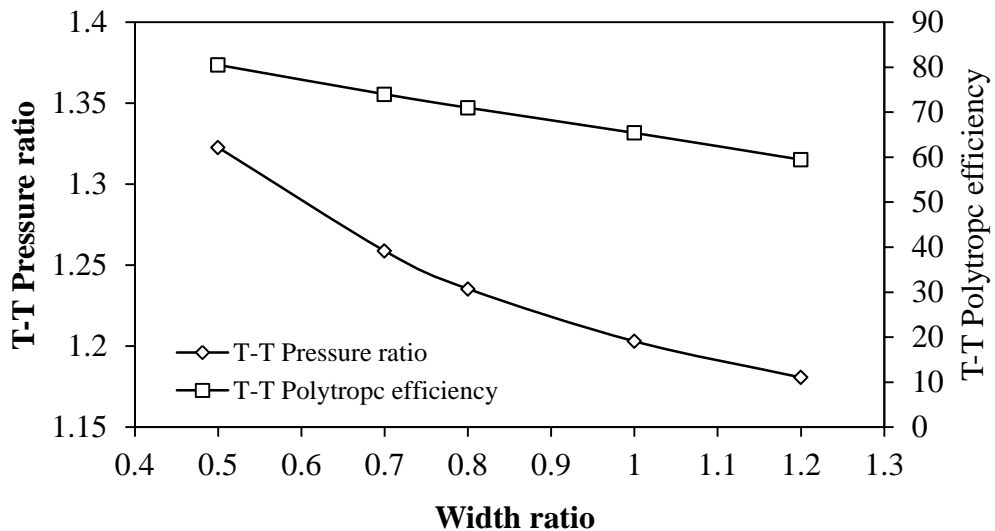

Figure 9: Total-Total Polytropic Efficiency and Total-Total Pressure at Different Width Ratio

Figure 10 shows the absolute Mach number difference across the streamwise location, from the inlet impeller to the outlet vaned diffuser; it is visibly seen that Mach number is higher at the space between the impeller exit and vane inlet; therefore, it is very important to take into account the impact of a space ratio in the design.
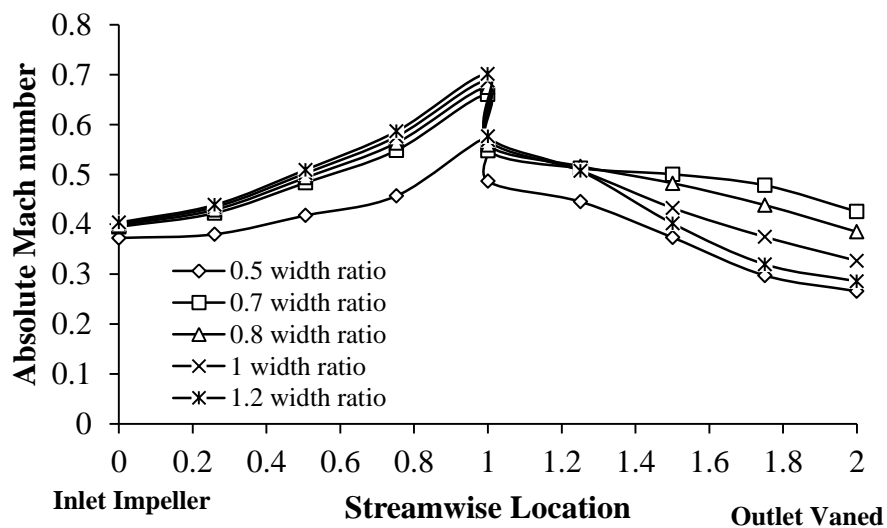

Figure 10: Absolute Mach number with Streamwise Location from Inlet to Outlet

Figure 11 shows the meridional velocity difference across the trailing edge (TE) Spanwise of the vaned diffuser configurations for a different width ratio it is clearly seen that velocities is increasing at 0.2 spanwise between hub and shroud, and then it is decreasing at a shroud, the reason due to narrow passage between shroud and vaned tip. Obviously, the meridional velocity at 0.5 width ratio is higher than the others' types, the reason behind that due to an outlet narrow passage between shroud and vaned tip.
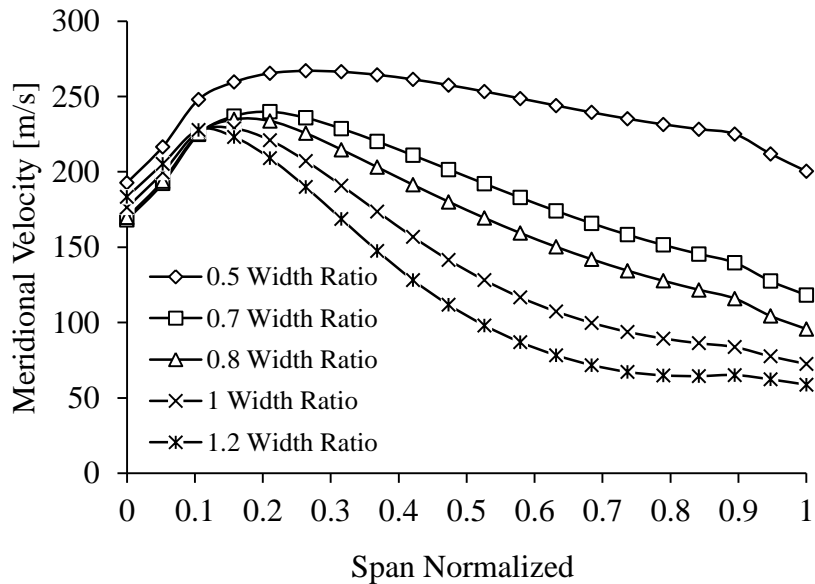

Figure 11: Meridional Velocity (Cm) versus Span Normalized at Trailing Edge (TE) of Vaned Diffuser for Different Width Ratio

The effect of absolute Mach number difference across the Spanwise location at trailing edge of the vaned diffuser as shown in Figure 12, it is visibly seen that Mach number is higher at 0.5 width ratio than others' types of width ratios.
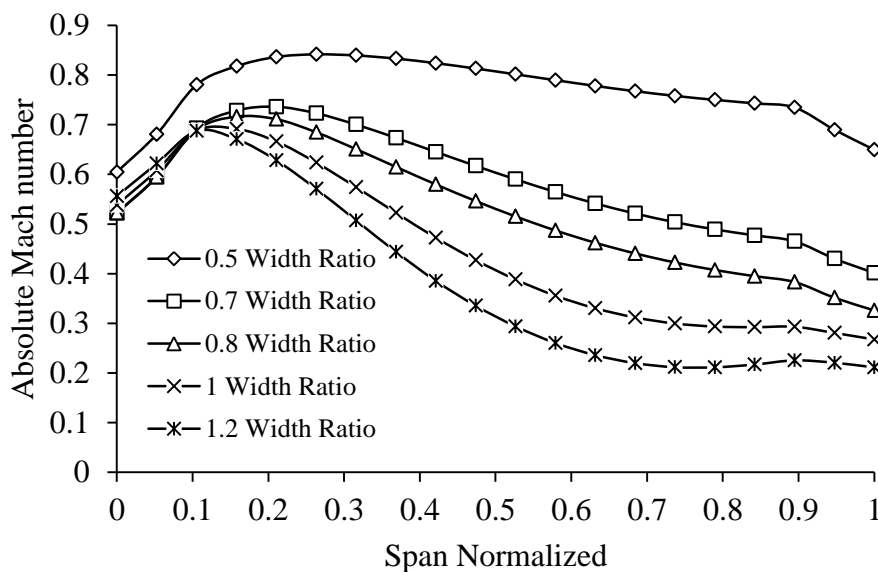

Figure 12: Absolute Mach Number Versus Spanwise Location at Trailing Edge (TE) of Vaned Diffuser

The absolute Mach number difference across the streamwise location, from inlet impeller to outlet vaned diffuser as shown in Figure 13 it is visibly seen that Mach number is higher at the space between the impeller exit and vaned inlet; therefore, it is very important to take into account the impact of a space ratio in design.
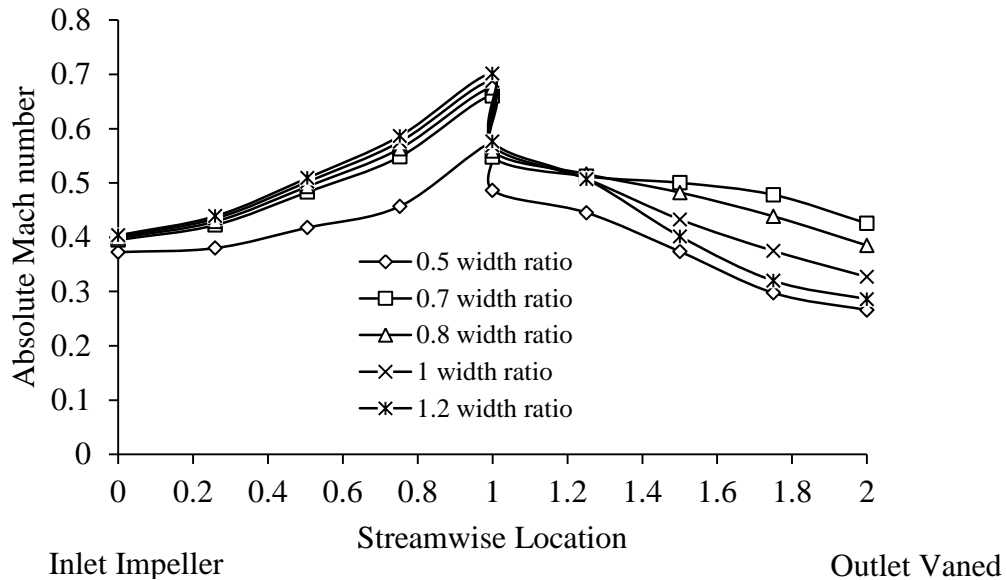

Figure 13: Absolute Mach Number Versus Streamwise Location from Inlet to Outlet

The effect of static pressure difference across the streamwise location of the modified vaned diffuser centrifugal compressor for a different width ratio as shown in Figure 14; it is clearly seen that the static pressure is decreasing at space between exit impeller and inlet vaned diffuser; furthermore, it is obviously seen that static pressure is increasing through vaned diffuser due to the kinetic energy which is converted into static pressure. Conclusively, the static pressure for 0.5 width ratio is higher than others' types of width ratios.
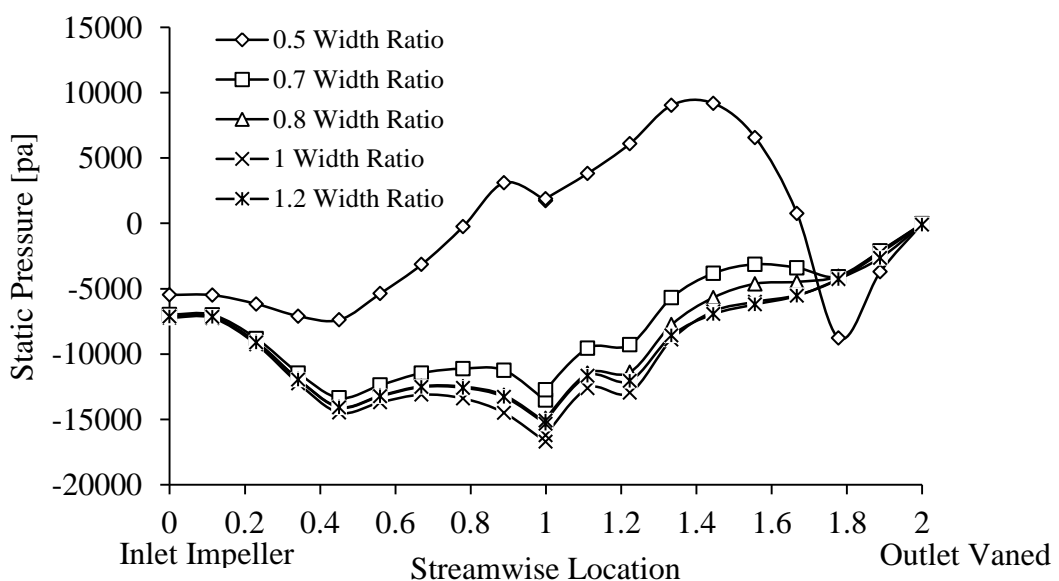

Figure 14: Static Pressures (Ps) Versus Streamwise Location

The velocity vector magnitude at the mid Spanwise location for impeller vaned diffuser configurations from the impeller inlet and vaned exit as shown in Figure 15. It can be seen that the flow conditions in diffuser near the outlet vary obviously over different width ratio, when the width ratio is 1.2, as a result of meridional passage, a backflow vortex exists near the pressure surface next to the diffuser outlet. It greatly increased the flowing loss; therefore, the efficiency and pressure ratio is low; when the width ratio is 0.8, the backflow vortex and its influence region decrease a lot, when the width ratio is 0.5, the flow field becomes uniform. Therefore, the flow losses decrease while the efficiency and pressure ratio becomes higher.

0.5 Width ratio 0.7 Width ratio 0.8 Width ratio

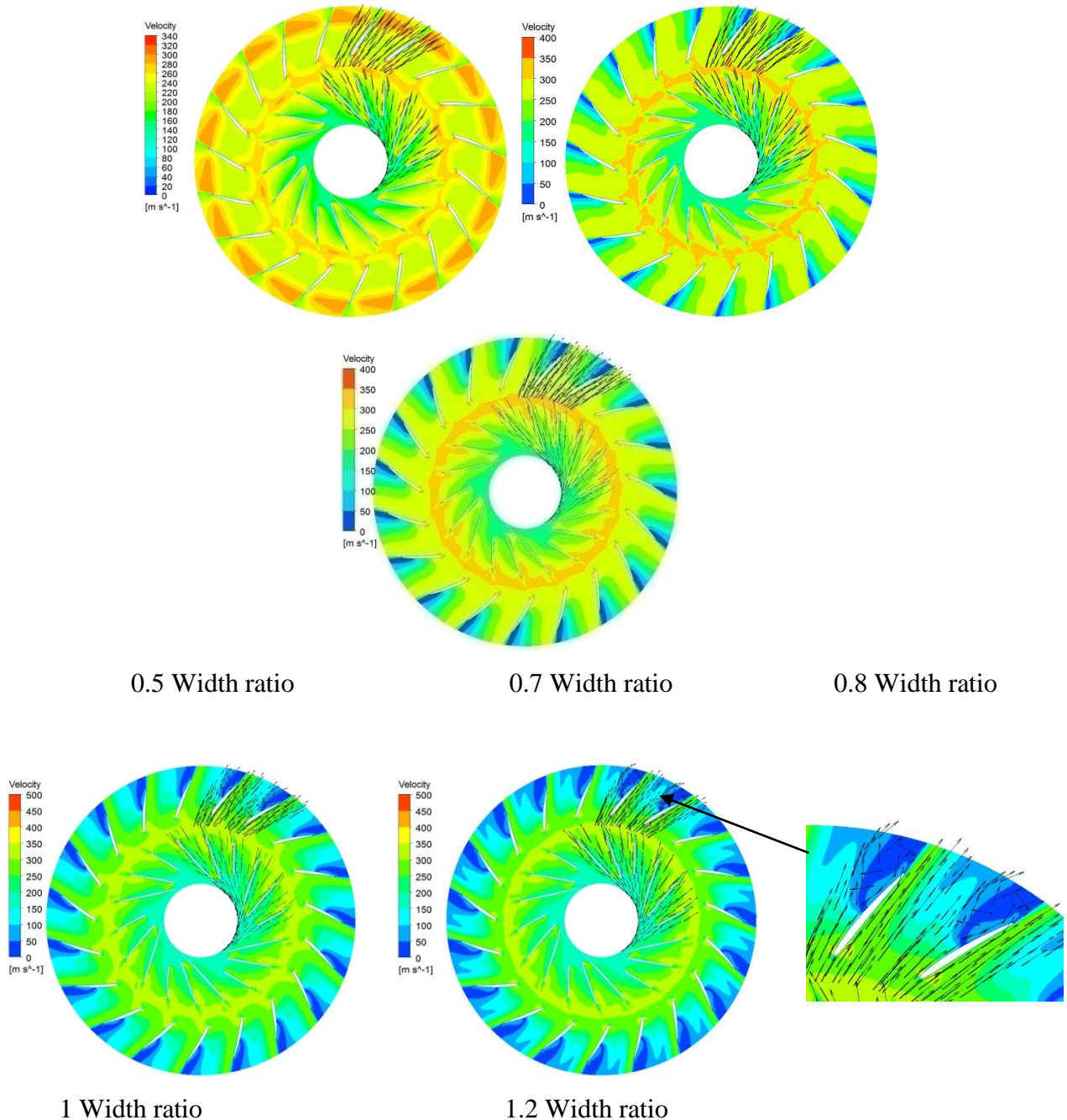1 Width ratio 1.2 Width ratio

Figure 15: Velocity Vectors at 50% Span for Different Width ratio

In addition, the contour of a relative Mach number at the mid Spanwise location for impeller and vaned diffuser configurations from the impeller inlet and vaned exit, as an example of CFD computations as shown in Figure 16. It can be seen no choke of flow at the inlet of the modified type because of the uniform distribution of the throat area between 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. It can be seen 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 for all configurations of width ratios. It is clearly seen that the effect of flow vaned angle to increase and decrease the backflow vortex.

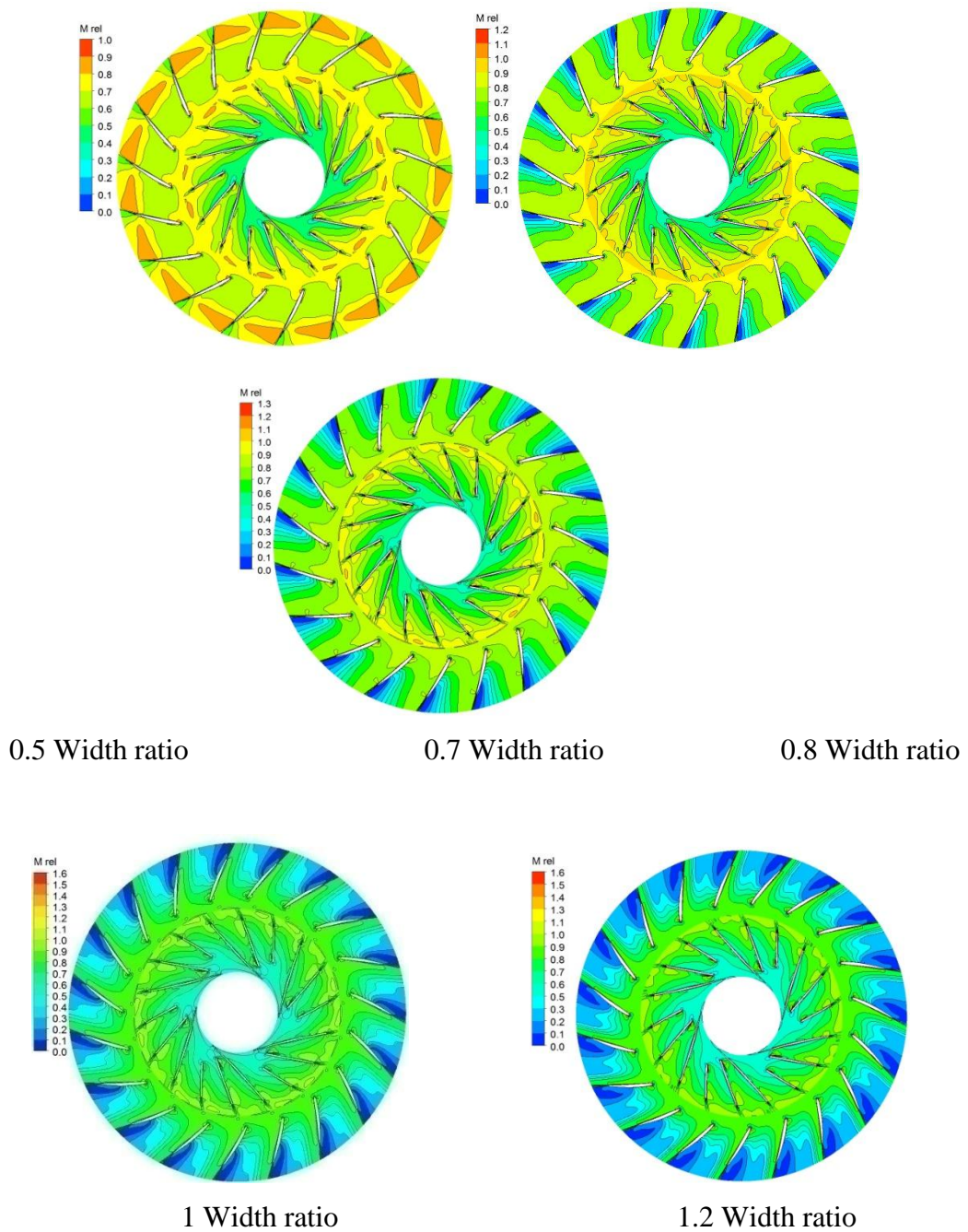| 0.5 Width ratio | 0.7 Width ratio | 0.8 Width ratio |

| 1 Width ratio | 1.2 Width ratio |

Figure 16: Contour of Relative Mach number at 50% Span for Different Width ratio

Moreover, the contour of static pressure difference on the meridional surface of the impeller vaned diffuser from the inlet to outlet location for different configurations of width ratios as shown in Figure 17 it is obviously seen that the effect of the outlet band width of the distribution of static pressure, it can be seen evidently the effect of vaned diffuser to increase the static pressure, the reason behind that due to the kinetic energy which is converted into static pressure.

0.5 Width ratio   0.7 Width ratio   0.8 Width ratio

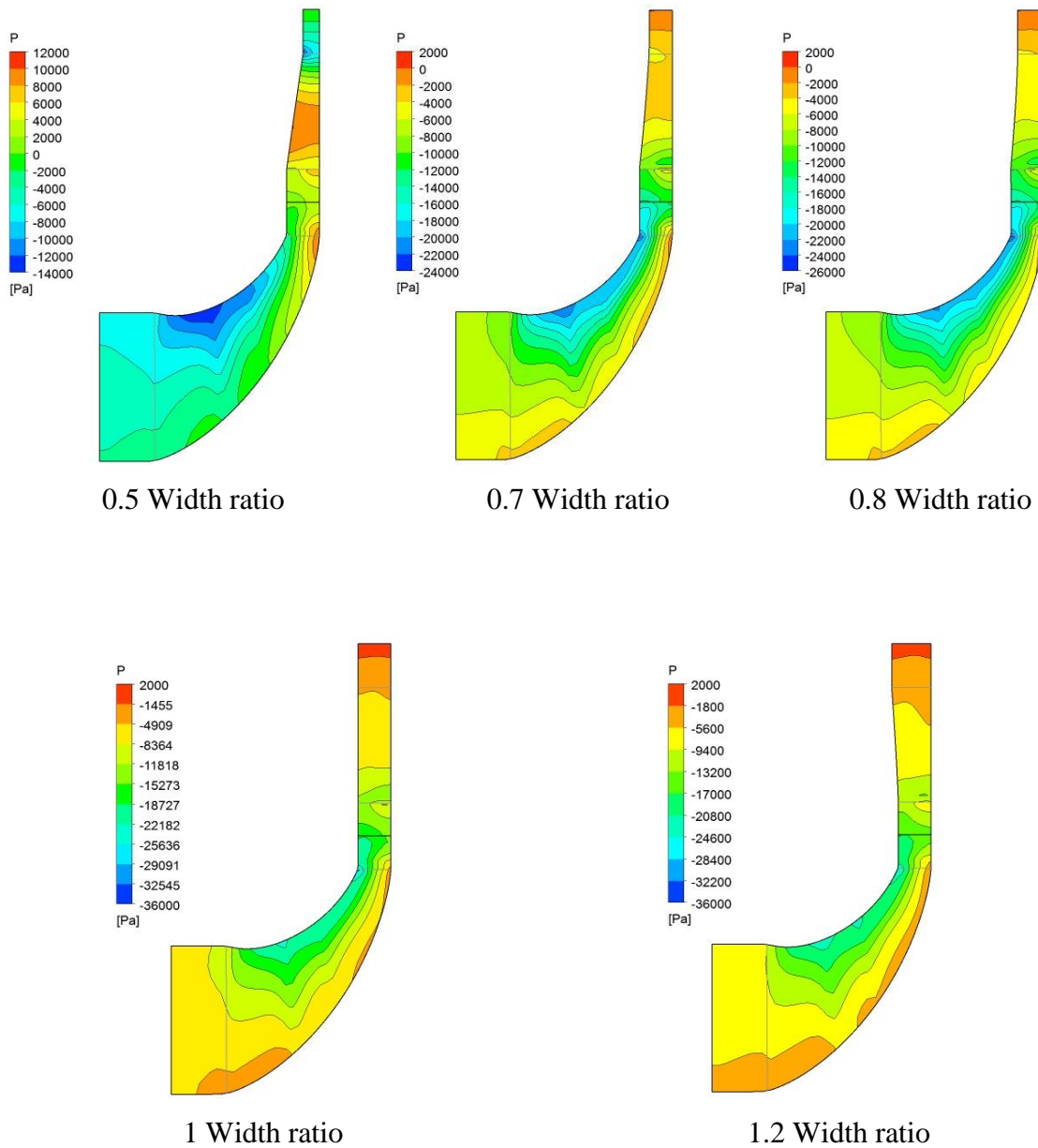1 Width ratio   1.2 Width ratio

Figure 17: Contour of Static Pressure on Meridional Surface for Different Width ratio

Figure 18 shows the relative Mach number on a meridional surface of the impeller vaned diffuser from the inlet to outlet location for different configurations of width ratios. It is clearly seen that the relative Mach number of 0.5 width ratio type is lower than the relative Mach number of other's types at outlet flow and at the space between the impeller exit and vaned outlet. It is necessary to convert the high kinetic energy to a static pressure through a diffuser provided downstream of the impeller.

| 0.5 Width ratio | 0.7 Width ratio | 0.8 Width ratio |

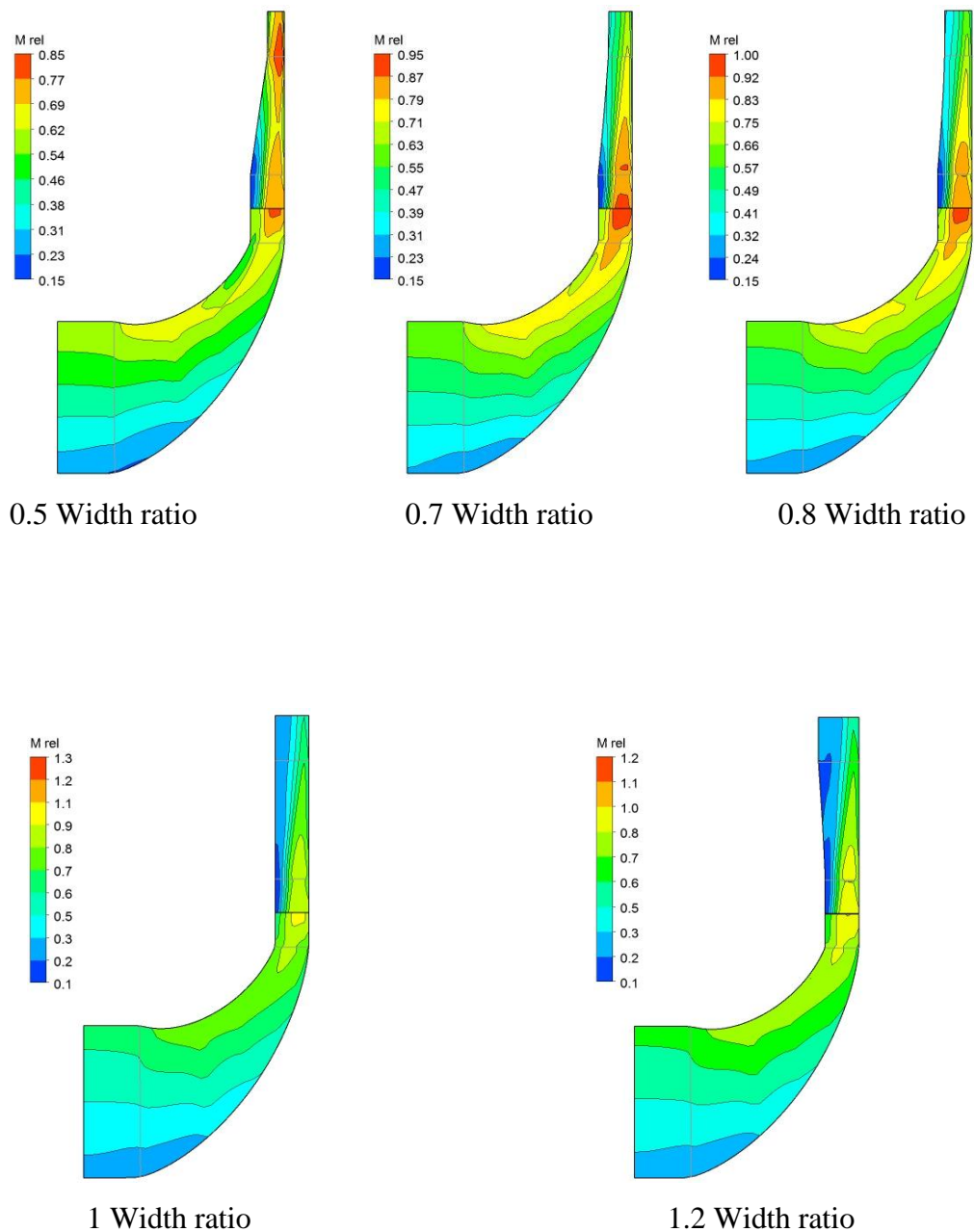| 1 Width ratio | 1.2 Width ratio |

Figure 18: Contour of Relative Mach number on Meridional Surface for Different Width ratio

Figure 19 shows the contour of meridional velocity from Leading Edge (LE) of impeller to Trailing Edge (TE) of vaned diffuser for different configurations of a width ratio. It is clearly seen that the effect of outlet 0.5 width ratio to decrease the meridional velocity along a meridional length of vaned diffuser. Moreover, the diffuser is used to reduce this velocity while at the same time it increases the static pressure. It can be seen that the high velocity is located in the space between the impeller exit and vane inlet. As earlier mentioned that the space area ratio which is a very important factor to modify.

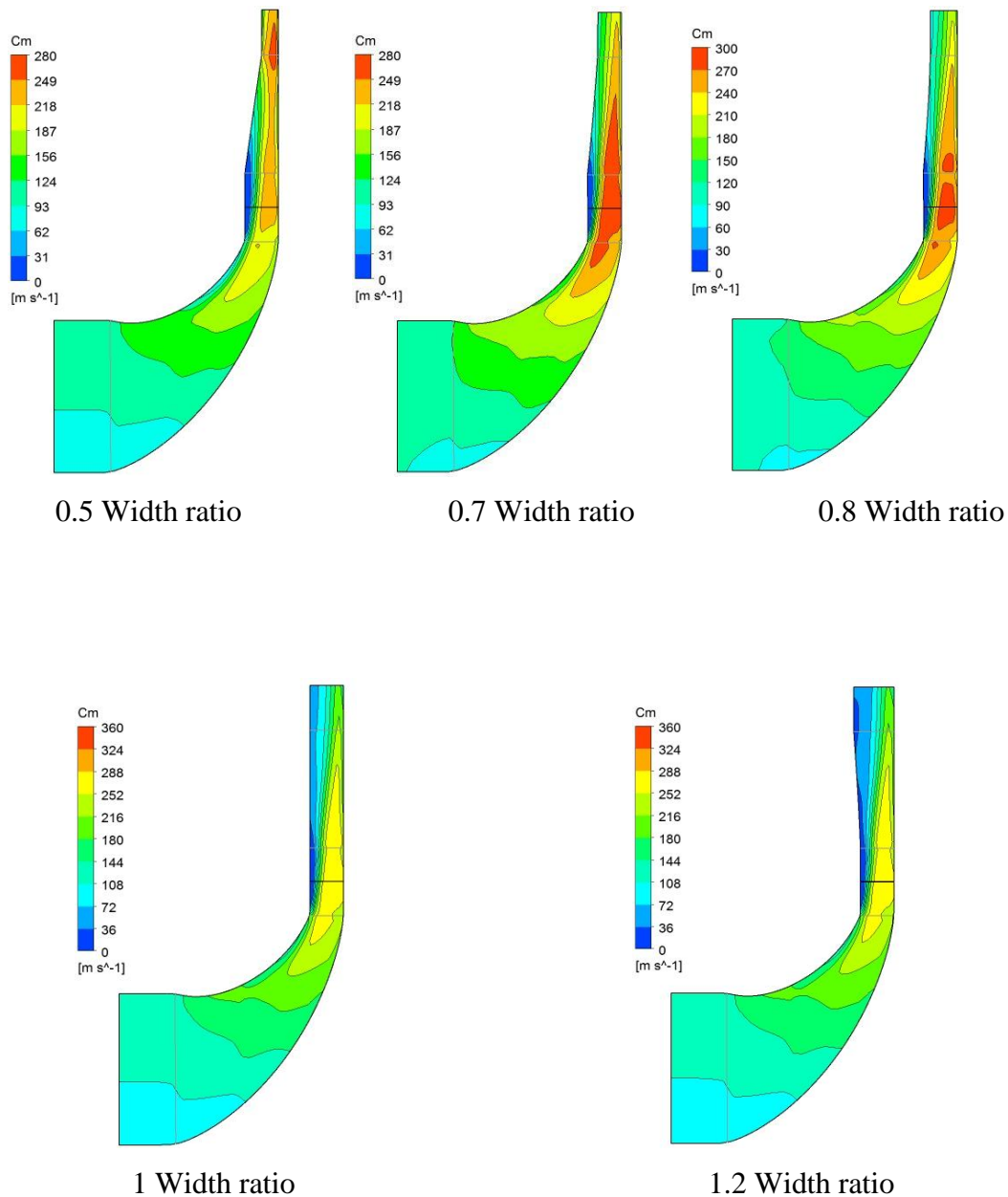| 0.5 Width ratio | 0.7 Width ratio | 0.8 Width ratio |

| 1 Width ratio | 1.2 Width ratio |

Figure 19: Contour of Meridional Velocity on Meridional Surface for Different Width ratio

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

5. Conclusions

The result is of relevance in internal combustion engine boosting applications: larger mass flow and compression ratio can be increased without experiencing a surge. Compressor surge margin is a major concern in current automotive engines because the downsizing trends. However, the matching between compressor impeller and vaned diffuser is difficult because the surge limit obtained in steady conditions. Computational fluid dynamics (CFD) play a greater part in the aerodynamic design of turbomachinery than it does in any other engineering application. For many years in 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 effects of the different meridional passage width ratio of vaned diffuser on the centrifugal compressor performance are systemically simulated and analyzed. The analysis of the flow characteristics was also performed to obtain a better understanding of the blade to blade vaned diffuser compressor flow behaviours, at a certain speed it was 60k RPM. The results show the range of the best width ratio is determined. The following conclusions are obtained; firstly, reducing the width ratio can enhance the performance of centrifugal compressors. When the width ratio is 0.5, the Polytropic efficiency of the impeller coupled with the vaned diffuser reaches the highest value 73.52%, and the total pressure ratio gets to the highest value 1.35. secondly, as the width ratio decrease, the flow in the radial direction of the diffuser meridional passage is squeezed. The backflow and vortex near the pressure surface gradually disappear. The flow tends to be uniform; hence the efficiency and total pressure ratio are both improved. 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 Streamline plots are generated for better understanding of fluid flow through the centrifugal compressor stage. Conclusively, the aerodynamic results show that the best width ratio range of diffuser meridional passage is 0.5 and 0.7. Obviously, the performance was significantly affected by outlet meridional width passage vaned diffuser.

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.

[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. M. P.J. Shook, W. Oakes, 1994. "The Aerodynamic Performance of a High Speed Research Centrifugal Compressor Facility," AIAA Paper 1994-2798, pp. 1-9.

[14] J.T. 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] Mendonça F, 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] K.U. Ziegler, H.E. Gallus, and R. Niehuis, "A Study on Impeller-Diffuser Interaction-Part I: Influence on the Performance," Journal of Turbomachinery, vol. 125, 2003, pp. 173-182.170.

[18] K. Ramakrishnan, P.B. Lawless, and S. Fleeter, "High Speed Centrifugal Compressor Aeromechanics - Impeller Unsteady Aerodynamics," AIAA 2007-5020, Cincinnati, OH: AIAA, 2007, pp. 1-18.

[19] W.B. Barry, "An Investigation of Unsteady Impeller-Diffuser Interactions in a Centrifugal Compressor," Purdue University Thesis, 1991.

[20] K. Gallier, P.B. Lawless, and S. Fleeter, "PIV Characterization of High Speed Centrifugal Compressor Impeller-Diffuser Interaction," AIAA 2007-5019, Cincinnati, OH: AIAA, 2007, pp. 1-8.

[21] K.U. Ziegler, H.E. Gallus, and R. Niehuis, "A Study on Impeller-Diffuser Interaction-Part II: Detailed Flow Analysis," Journal of Turbomachinery, Vol. 125, 2003, pp. 183-192.

[22] S. Ibaraki, T. Matsuo, and T. Yokoyama, "Investigation of Unsteady Flow Field in a Vaned Diffuser of a Transonic Centrifugal Compressor," Journal of Turbomachinery, vol. 129, 2007, p. 686.

[23] Layth H. Jawad, S. Abdullah, R. Zulkifli1and W.M.F.W. Mahmood, (2012). Modelling of Centrifugal Compressor Impellers using Adaptive Neuro- Fuzzy Inference Systems (ANFIS). International Review of Mechanical Engineering (IREME) Vol. 6 n° 5, July 2012.

[24] Layth H. Jawad, S. Abdullah, R. Zulkifli1and W.M.F.W. Mahmood, (2012). Prediction of Centrifugal Compressor Performance by Using Adaptive Neuro-Fuzzy Inference System (ANFIS). International Review on Modelling and Simulations (IREMOS) Vol. 5 n° 4, August 2012.

[25] Layth H. Jawad, S. Abdullah, R. Zulkifli and W.M.F.W. Mahmood, (2012). Numerical Simulation of Flow inside a Modified Turbocharger Centrifugal Compressor. Asian Journal of Applied Sciences, Vol. 5 (8), pp. 563-572, 2012.