International Journal of Engineering Works
ISSN-p: 2521-2419
ISSN-e: 2409-2770
Vol. 6, Issue 11, PP. 399-405, November 2019
https://www.ijew.io/
https://doi.org/10.34259/ijew.19.611399405

Numerical Analysis of Flow through High Pressure Ratio Centrifugal Compressor Impeller and Effect of varying Diffuser Exit Width on Performance

Amjid Khan[1], Tariq Ullah[2]
[1,2] Centre for Advanced Studies in Energy, NUST, ISB, Pakistan
17teeamjid@uspcase.nust.edu.pk, 17teetariq@uspcase.nust.edu.pk
Received: 07 October, Revised: 15 October, Accepted: 20 October

***Abstract*—**This paper carries out a comprehensive numerical investigation of turbocharger high-pressure ratio centrifugal compressor impeller. The aim is to study the effect of varying mass flow rate on the pressure ratio and efficiency from stall to choke using (3D) numerical simulations. The transonic SRV2 compressor developed by DLR (German Aerospace Center) has been used as the test case in this study. Numerical simulations have been performed using Reynolds Averaged Navier-Stokes (RANS) based k-ε model to predict turbulence. Y-plus is kept 35 for the structured mesh near the boundaries. In first part, calculations were carried out for design speed of 50,000 1/min to study the suitability of ANSYS CFX in the design procedure and compared the results with experimental data and four other (3D) solvers. The numerical simulations showed that ANSYS CFX over predicts the experimental data by 9% in this compressor.

The second part describes the effect of vaneless diffuser exit width on performance parameters of centrifugal compressor at design high rotational speed, which shows that decreasing vaneless diffuser exit width increases pressure ratio, isentropic efficiency and operating range from stall to choke.

***Keywords*—** Centrifugal compressor, Diffuser Exit width, Numerical simulations, Entropy generation, Transonic, Pressure, Efficiency, Performance, Pressure ratio

I. Introduction

In this age of renewable energy, energy efficiency is the most important thing to make contribution toward power consumption and power generation. The aim of this research paper is to improve the performance of centrifugal compressor in order to reduce power consumption. The extremely huge use of high-pressure centrifugal compressors in automotive turbochargers and micro-gas turbines make it necessary to improve its performance by increasing operating range and reducing losses. For these applications, usually unshrouded impellers with splitter blades are used. To circumvent huge stresses with rise in weight unshrouded compressors are used [1].

In this case, vaneless diffuser compressor (SRV2-O) designed and fabricated by German Aerospace center (DLR) is used to achieve the goal of high operating range using numerical simulations. Using conventional techniques and laser velocimetry at DLR test rig, this compressor was investigated experimentally at design speed (50,000 1/min) and design mass flow rate (2.55 kg/s). The total pressure ratio at these design conditions is 5.65:1 and performance map is shown in Fig. 1 [2].

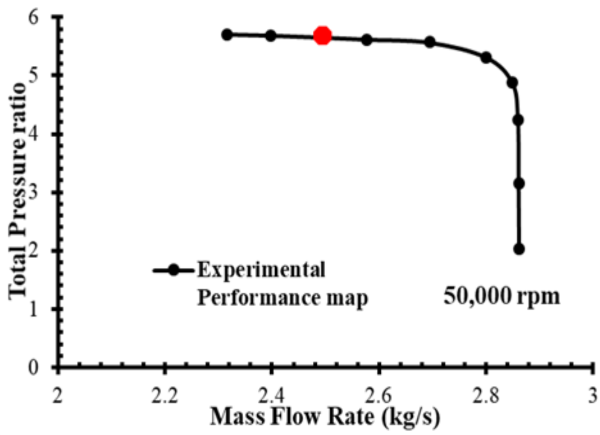

Figure 1: Experimental Performance Map

Over the last few decades, many 3D solvers have been developed and made commercially available. The computing power have been increased so many times, which made it possible to use it for industrial purposes for fluid dynamics analysis and reduce the experimental prices by designing the most optimized model using numerical simulations. Many 3D solvers like VISIUN, STAGE3D, TASCFlow and FLOWSIM have been used to predict the performance of centrifugal compressor and all of them have given quite reasonable results [3]. The first purpose of this paper is the applicability and suitability of commercial ANSYS-CFX mostly used nowadays to validate the experimental results and compare these results with other computational fluid dynamics commercial solvers mentioned above. It is worth mentioning that ANSYS-CFX and ANSYS-FLUENT gives almost similar results.

The second part of this paper is variation in the diffuser exit width and its effect on the performance of transonic centrifugal compressor. Diffuser exit width can be varied in two ways; first by reducing diffuser exit width from shroud side and second by decreasing diffuser exit width from hub side. From literature it has been clarified that decreasing exit width from hub side has too minor effect on the performance of centrifugal compressor, while reducing exit width from shroud side can increase pressure ratio, efficiency and operating range [4]. The aim of this research is working on variation of diffuser exit width from shroud side and studying its effect on pressure ratio, efficiency and operating range. This phenomena of variation in diffuser exit width is called pinching.

It has been studied that as diffuser-exit width has been reduced, it results increase in efficiency and pressure ratio at low rotational speed. At high rotational speed, the result was different as it given us high efficiency but lower pressure ratio [5]. The other advantange of reduced diffuser-exit width is that it reduce secondary flow losses caused by tip-leakage flow. At reduced diffuser exit width, shroud forces the flow towards centre, accelerate it and decrease losses in diffuser section, which results in high pressure ratio of the stage. When the diffuser-exit width is reduced it decreases stall phenomena chances as it provide greater margin to critical flow angle at impeller exit. It also make it possible to increase the pressure ratio by increasing the diffuser length [4].

The numerical simulated results showed valuable increase in the isentropic efficiency then the original standard compressor as diffuser exit width is reduced 20% and 25%, while in case of 10% and 15% reduction in diffuser exit width has lower isentropic efficiency then experimental test case compressor [6]. The wake fluid area has also been reduced on the blade suction-side as the diffuser exit is reduced. In case of vaned diffuser performance of centrifugal compressor has improved significantly as diffuser exit width is reduced to optimized level [7].

Numerical simulations have been performed by reducing the blade exit width by 5% and 10% and results pronounced great increment in isentropic efficiency and static pressure rise with reduced losses in diffuser [5].

A. *Specifications of Simulated Case*

A centrifugal compressor test case has been studied to understand effect of diffuser exit width, having high pressure ratio, mass flow coefficient and specific speed. The test case was designed and built by DLR (German Aerospace Center).
As shown in the Fig. 2, impeller with 13 full and 13 splitter blades. Splitter blades leading edges is at 26% of full blade chord. The exit diameter of test case compressor is 112mm and nominal tip speed is 586m/s at 50000 1/min. A vaneless diffuser is connected after the impeller as depicted in Fig. 2 [3].
The compressor (SRV2-O) test case data is shown in table:

Table 1: Specifications of SRV2-O compressor [3]

Parameter	**Value**	**Unit**
Inlet ambient pressure, P_{t1}	101325	[Pa]
Inlet ambient temperature, T_{t1}	288.15	[K]
Design Shaft speed, n	50000	[1/min]
Design mass flow rate, m	2.55	[kg/s]
Number of full/splitter blades	13/13	-
Tip speed of Impeller	586	[m/s]
Pressure ratio of Impeller	6.1	-
Impeller efficiency	84%	-

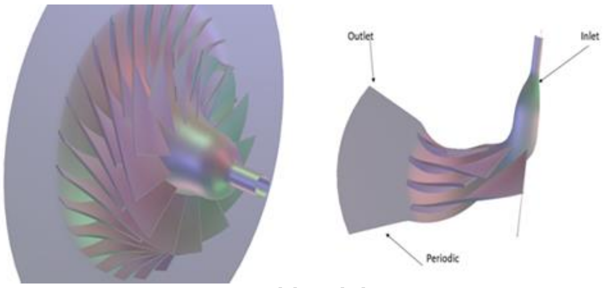

Figure 2: Model in BladeGen

II. NUMERICAL SETUP

A. *Turbulence Model*

The first step is to model 3D geometry for compressor. Based on available test case data, 2D blade shape has been modified to 3D model using Commercial modeler ANSYS BladeGen. Only single passage of impeller having single pair of blades (full and splitter) has been modeled to reduce computational time for simulation due to smaller number of mesh elements as illustrated in Fig. 4. For CFD simulations ANSYS CFX 15.0 is used to model flow field for test case (SRV2-O) compressor under steady state conditions. K-epsilon model used as turbulence model as it gives better results for planar shear layer and recirculating flows as is the case of centrifugal compressor. According to the K-epsilon turbulence model criteria, the value of Y-plus is set to 35.

B. *Grid Generation (Turbogrid)*

After modelling centrifugal compressor in BladeGen, an H-grid mesh topology was created in ANSYS TurboGrid module. TurboGrid is a meshing scheme for turbomachinery configurations, it selects automatic and refined mesh quality for most complex geometries creating structured mesh. For the better mesh quality, compressor computational domain is divided to H-grid topology with splitter blades arrangement. Design tip clearance for this case is variable tip clearance of 0.5,0.3mm and diffuser exit width of 7.06mm. Mesh is generated using H-grid with 25 number of elements at the inlet and outlet each. A grid independency test executed using 3 distinct grid sizes for complete compressor. The grid with total number of mesh elements 560788 and total number of nodes 504345, has been found adequate as alteration in pressure ratio and isentropic efficiency was negligible in this range of elements as depicted in Fig. 3.

C. *Computational Solver*

In order to study inlet, impeller and diffuser, boundary conditions were specified in CFD model and stage mixing approach is used. First order finite volume discretization is used for turbulence numerics and high resolution as advection scheme. The simulations of compressor were performed using commercial ANSYS CFX flow solver. The time scale factor 1 is used for calculations over 1000 iterations for steady state 3D centrifugal compressor analysis.

To analyse diffuser exit width effect, numerical simulations using Reynolds Averaged Navier-Stokes (RANS) based K-ε Model for turbulence modelling.

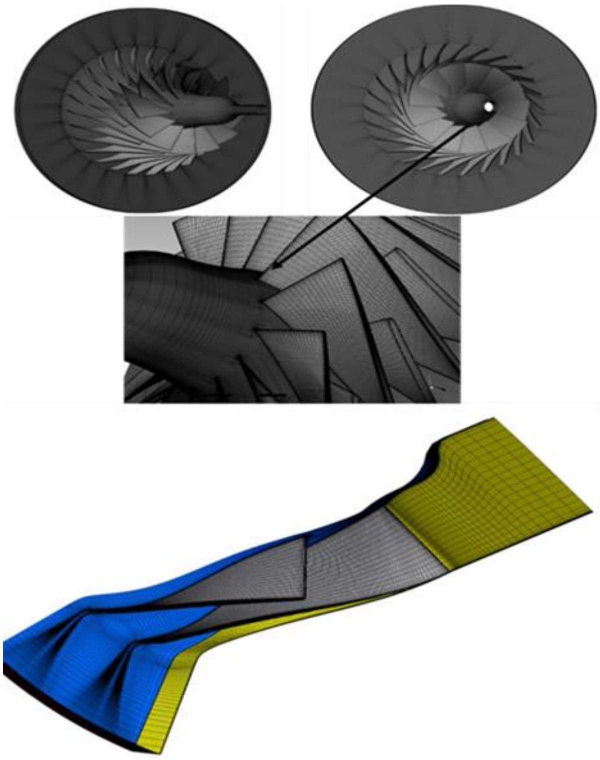

Figure 3: Computational Mesh (TurboGrid)

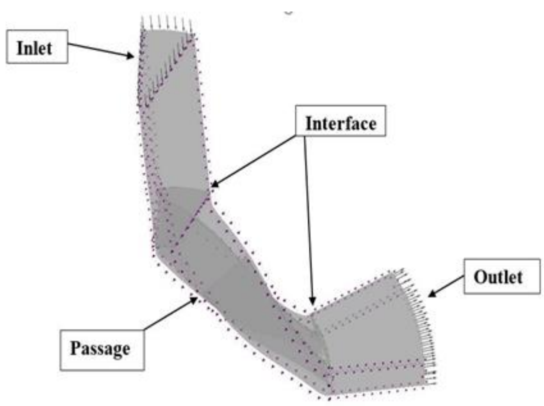

Figure 4: Computational Domain

D. Boundary conditions:

The compressor inlet conditions for all simulated cases; total inlet pressure of 101.325 kPa and total inlet temperature is 288.15 K respectively with 5% medium turbulence intensity, according to standard ambient conditions used by DLR (German Aerospace Center) for turbocharger configuration. The convergence criterion is set to 1x10-5 for all residuals while for monitoring convergence, mass flow rate and isentropic efficiency has been recorded for choke margin. As at choke point minor variation in mass flow rate cause substantial variation in performance, so to check choke limit, static pressure is defined at outlet while mass flow rate is monitored, and choke point is found by slightly reducing static pressure at outlet. The stage or mixing plane interfaces have been defined between diffuser, impeller and inlet. The rotational periodicity has been defined at boundaries of single passage assuming flow symmetry in all passages [2]. The total number of flow passages are 13 and each passage consist of 1 main and 1 splitter blade as shown in Fig. 4.

III. RESULTS AND DISCUSSIONS

A. Validation results:

For numerical simulations following agreements were made similar for all Computational fluid dynamics solvers and codes, which are given below:

- Compressor stage geometry is same for all (3D) solvers
- Turbulence model is K- ε model
- Design mass flow rate 2.55kg/s
- Design speed of 50,000 1/min

Numerical calculations for performance map of compressor stage are carried out on VISIUN and ANSYS-CFX at design conditions and the total pressure ratio data extracted from stall to choke is validated with experimental data, which shows suitability of these solvers compared to experimental data as shown in Fig. 5. The impeller total pressure ratio has been calculated for all solvers including ANSYS-CFX and compared it with experimental data calculated by DLR [3] as depicted in Fig. 6.

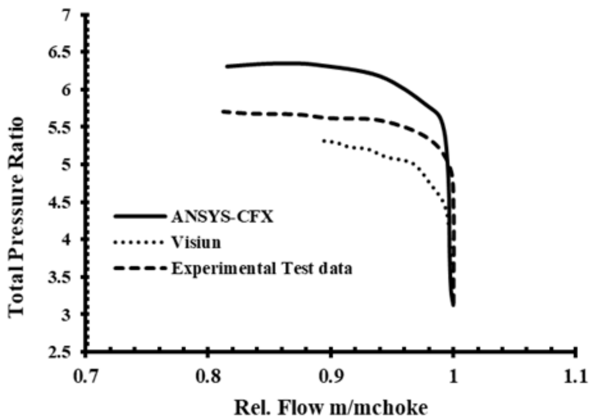

Figure 5: Compressor Stage Validation

The validation results of compressor stage show that ANSYS-CFX over predicts total pressure ratio by 9% while VISIUN shows lower pressure ratio value than experimental measured values. It depends on the compatibility of software which shows different results, but both the solvers showed almost similar results close to experimental test total pressure ratio.

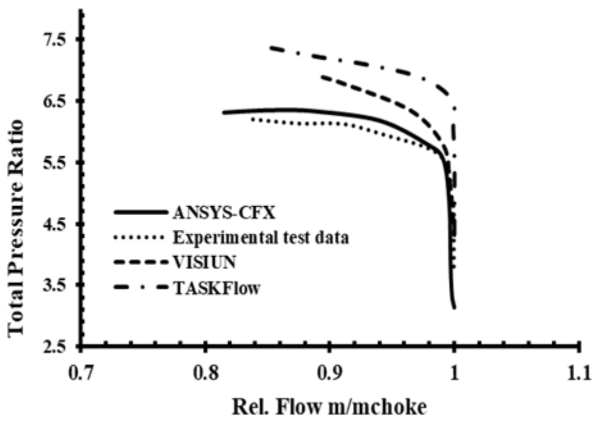

Figure 6:Validation of SRV2-O validation

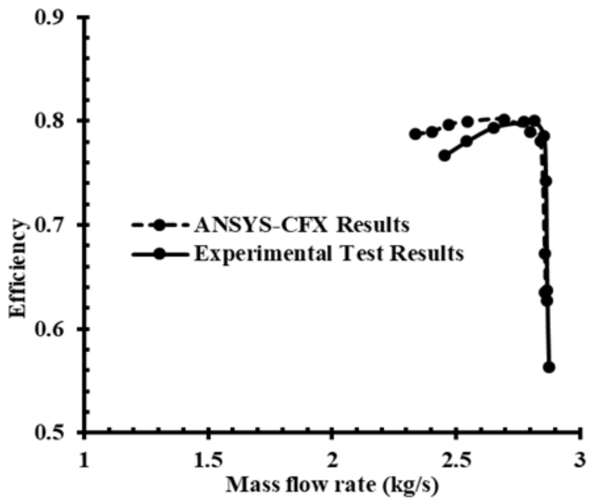

Figure 7: Efficiency Validation

Then the validation of impeller is executed using numerical simulations on ANSYS-CFX. The total pressure ratio and mass flow rate were extracted from results as depicted from Fig. 6. The prediction of total pressure ratio of ANSYS-CFX is appropriate compared to the design of DLR.

The data for efficiency has been extracted from ANSYS-CFX and compared it with experimental test results as illustrated in Fig. As the efficiency data from other solvers is not available, so the experimental data is only validated using ANSYS-CFX data. Which shows pretty similar results with test case data as shown in Fig. 7.

It is thus concluded from the current analysis that the computation results and simulations are capable of predicting the realistic results for the performance parameters i.e, pressure ratio and isentropic efficiency of centrifugal compressor, despite of the fact that at low mass flow rate it shows some deviation from experimental results. This deviation from experimental data can be allocated to the limitations of turbulence model and stage mixing frame approach applied at the interfaces.

Effect of Diffuser exit width on Performance

The performance of centrifugal compressor has been studied and evaluated by extracting data for total pressure ratio and total to total isentropic efficiency. The study of flow field has been carried out using ANSYS-CFX for numerical simulations. Performance and flow field analysis is given below:

B. Overall Performance:

The performance curves shown in Fig. 8 as centrifugal compressor has been analyzed for design conditions; mass flow rate of 2.55 kg/s, rotational speed of 50,000 1/min and design exit width of 7.06 mm. The numerical simulations predicted the total pressure ratio and isentropic efficiency very close to design experimental test statistics. Now, off design conditions are applied. By increasing diffuser exit width, isentropic efficiency reduces, which results in flow separation and flow obstruction (due to change in incidence angle to impeller inlet). The performance of centrifugal compressor has been analyzed as the diffuser exit width is raised to 8mm, 9mm and 10mm.
At each diffuser exit width performance map has been drawn and compared it with design performance map for both total pressure ratio and isentropic efficiency as shown in Fig. 8 and 9. It has been observed that total pressure ratio for all the cases is lower than design pressure ratio and efficiency is also reduced for each increment in diffuser exit width. The reduction in the performance parameters is because of secondary flow losses at impeller exit. Operating range is reduced in each case of incremented diffuser exit width as depicted in Fig. 8.

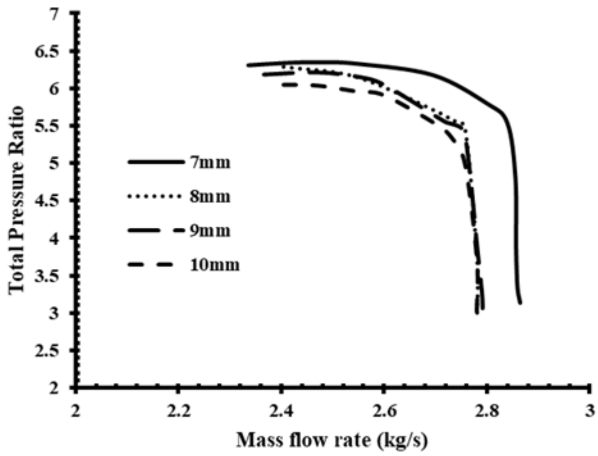

Figure 8:Performance Map for Total Pressure Ratio at Different Diffuser Exit Width

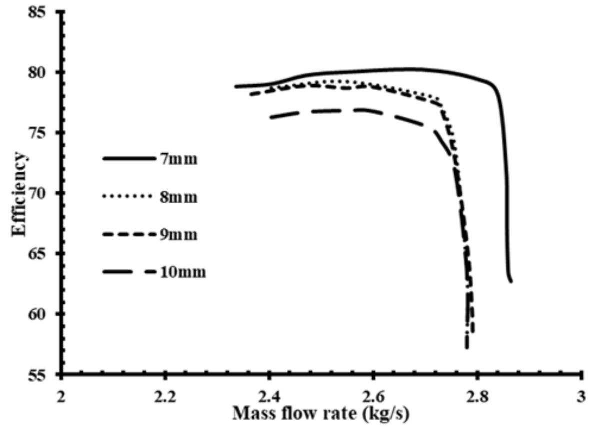

Figure 9: Performance Map for Efficiency at Different Diffuser Exit Width

A. Flow Field analysis:

To find out energy losses in flow circulation path, flow field analysis has been performed. The main reason behind flow losses are flow mixing, flow separation, flow-leakage and flow obstruction. Due to centrifugal and Coriolis forces, secondary flow vortex is formed by flow separation boundary layer and tip-leakage flow as mostly flow in centrifugal impeller is three dimensional. This secondary flow mixes with primary flow at impeller exit and diffuser inlet, which cause diffuser to stall phenomena and key contributor towards stall phenomena is flow separation near hub of diffuser. This flow separation results in entropy generation at impeller exit. secondary flow pass through the high entropy regions, which results in compressor losses as depicted in Fig. 10. The flow field analysis for the test case compressor has been performed near choke and stall conditions and both points showed energy losses due to high-negative or high-positive incidences respectively. Drop in static pressure occurs at impeller inlet at choke mass flow rate, as the relative Mach number is too high due to high-negative incidence at impeller inlet shown in Fig. 11. The performance map obtained from ANSYS-CFX were analyzed for stall, choke and design point mass flow rate for various diffuser exit widths as shown in Fig. 9.

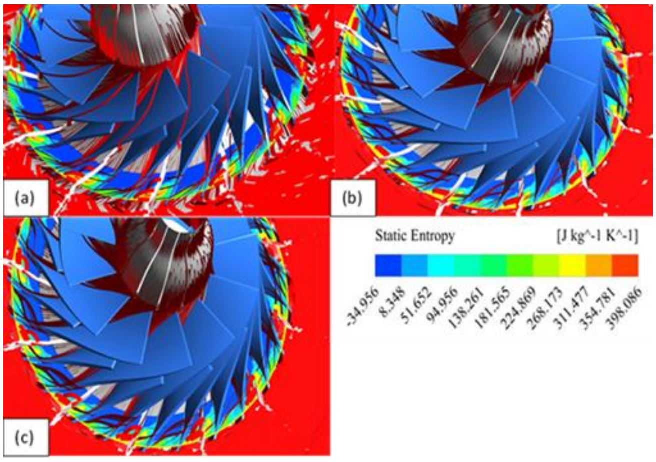

Figure 10: Secondary Flow Field at diffuser exit width (a)7mm (b)9mm (c)10mm

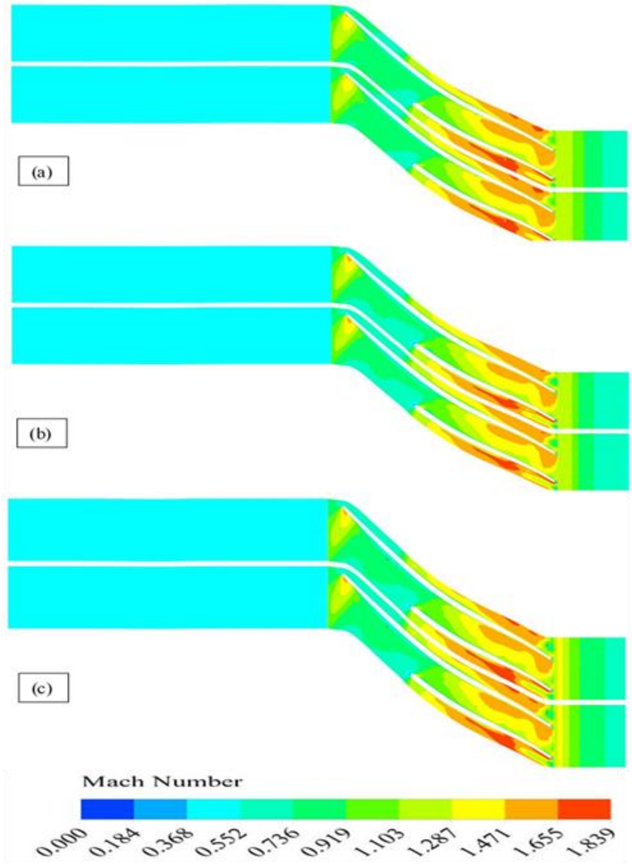

Figure 11: Mach Number for Different Diffuser Exit Widths (a)7.06mm (b)9mm (c)10mm

A. Relative Mach number:

As relative Mach number is too high near choke mass flow rate, so blade to blade analysis has been carried out near choke point for all values of diffuser exit width (7.06mm, 9mm and 10mm). As shown in Fig. 11 as the diffuser exit width increases, it increases Mach number at the suction-side of splitter and main blade. It can be observed from the Fig. 11 that wake area increases as the diffuser exit width increases and hence, variation in wake structure at impeller exit is detected.

B. Entropy Generation:

The main reason behind entropy generation is the secondary flow vortices and flow separation at the shroud of diffuser. The streamwise location of high entropy is impeller exit and diffuser inlet. Different contours have been drawn to analyze the effect of entropy for all values of diffuser exit width and results showed that entropy generation increases as the diffuser exit width increases as shown in Fig. 12 and 13.

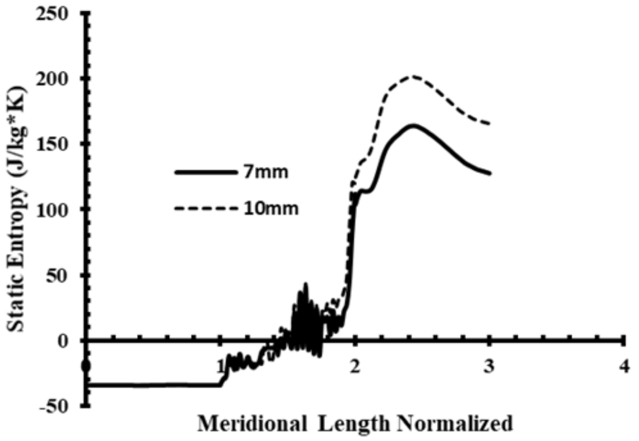

Figure 12: Static Entropy along Meridional Length for different diffuser Exit width

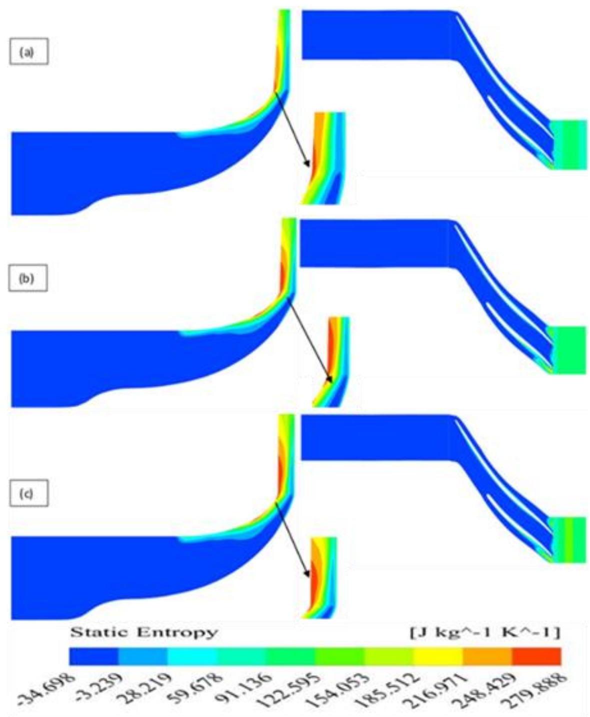

Figure 13: Static Entropy contours at meridional length of impeller at diffuser exit width of (a) 7mm (b) 9mm (c) 10mm

A. Static Pressure Variation along meridional length:

Static pressure rise has been investigated along meridional length normalized from inlet to outlet for test case compressor from stall to choke and then static pressure rise has been examined at design mass flow rate for 7 mm and 10 mm diffuser exit width. As it can be evaluated from the Fig. 14 that static pressure appears to decrease at choke point as the flow usually accelerate at that point due to negative flow incidence. Static pressure rise along meridional length from inlet to outlet for all diffuser exit widths investigated is shown in Fig. 15. It has been observed that at different diffuser exit widths, static pressure rise at choke point does not change along meridional length.

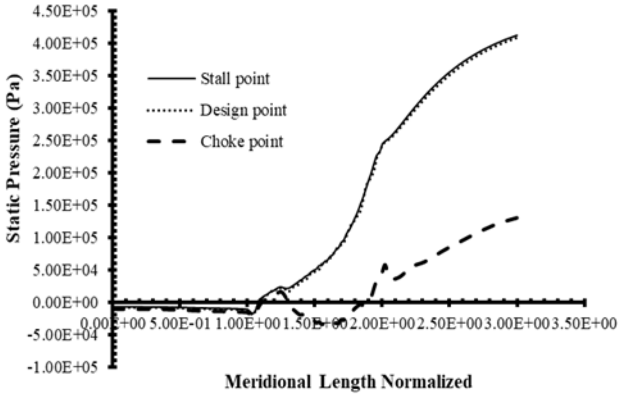

Figure 14: Static Pressure Rise for Stall, Design and Choke point

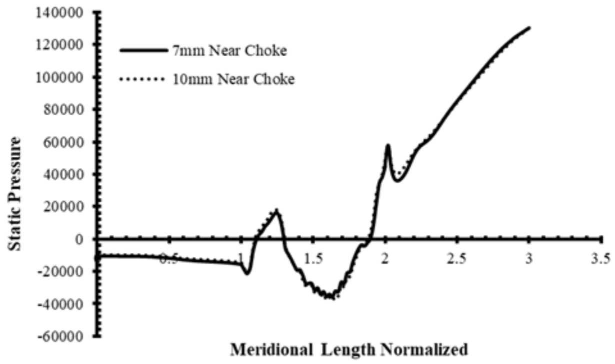

Figure 15: Static Pressure Rise for diffuser exit width of 7mm and 10mm at Choke point

CONCLUSIONS

Numerical simulations have been performed on the test case compressor (SRV2-O) and validated the experimental test results and then the effect of variation of diffuser exit width on the performance and flow pattern has been simulated. The following evaluations have been drawn from results:

- The test case compressor has been validated using ANSYS-CFX and evaluation showed quite similar results to design experimental data as it overpredicts the experimental data by only 9%
- As diffuser exit width has been increased, it decreases the performance parameters (Isentropic efficiency and pressure ratio)
- Entropy generation in a compressor showed that entropy increases as diffuser exit width increases, which results in drop in efficiency and pressure ratio
- Peak pressure ratio and peak efficiency at off design condition is lower than the experimental test case
- Static pressure at choke condition for all diffuser exit widths has no significant change

ACKNOWLEDGMENT

The authors wish to express their gratitude to U.S-Pakistan Center for Advance Studies in Energy (USPCAS-E), National University of Sciences and Technology (NUST) and the United States Agency for International Development (USAID).

REFERENCE

[1] S. N. Danish, Chaochen Ma, and Y. Ce, “The influence of tip clearance on centrifugal compressor stage of a turbocharger,” Proc. 4th WSEAS Int. Conf. Fluid Mech. Aerodyn. Elounda, Greece, vol. 2006, pp. 6–11, 2006.

[2] G. Eisenlohr et al., “Gt-2002-30394 Investigations of the Flow Through a High,” pp. 1–9, 2002.

[3] G. Eisenlohr, P. Dalbert, H. Krain, H. Pröll, F.-A. Richter, and K.-H. Rohne, “Analysis of the Transonic Flow at the Inlet of a High Pressure Ratio Centrifugal Impeller,” p. V001T01A007, 2014.

[4] A. Jaatinen-Värri, P. Röyttä, T. Turunen-Saaresti, and A. Grönman, “Experimental study of centrifugal compressor vaneless diffuser width,” J. Mech. Sci. Technol., vol. 27, no. 4, pp. 1011–1020, 2013.

[5] A. S. Rathore and K. B. Ravichandrakumar, “International Journal of Aerospace and Mechanical Engineering PARAMETRIC STUDY ON IMPELLER EXIT BLADE WIDTH VARIATION ON CENTRIFUGAL COMPRESSOR International Journal of Aerospace and Mechanical Engineering,” vol. 4, no. 3, pp. 16–22, 2017.

[6] Q. H. Nagpurwala, “Numerical Investigation of the Effect of Exit Width Trim on the Performance of a Centrifugal Compressor Impeller Numerical Investigation of the Effect of Exit Width Trim on the Performance of a Centrifugal Compressor Impeller,” no. January 2010, 2018.

[7] T. C. Siva Reddy, G. V. Ramana Murty, M. V. S. S. S. M. Prasad, and D. N. Reddy, “Experimental studies on the effect of impeller width on centrifugal compressor stage performance with low solidity vaned diffusers,” Proc. Inst. Mech. Eng. Part A J. Power Energy, vol. 221, no. 4, pp. 519–533, 2007.