World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:10, No:2, 2016

Lightweight High-Pressure Ratio Centrifugal Compressor for Vehicles-Investigation of Pipe Diffuser Designs by Means of CFD

Eleni Ioannou, Pascal Nucara, Keith Pullen

Abstract—The subject of this paper is the investigation of the best efficiency design of a compressor diffuser applied in new lightweight, ultra efficient micro-gas turbine engines for vehicles. The Computational Fluid Dynamics (CFD) results are obtained utilizing steady state simulations for a wedge and an "oval" type pipe diffuser in an effort to identify the beneficial effects of the pipe diffuser design. The basic flow features are presented with particular focus on the optimization of the pipe diffuser leading to higher efficiencies for the compressor stage. The optimised pipe diffuser is designed to exploit the 3D freedom enabled by Selective Laser Melting, hence purposely involves an investigation of geometric characteristics that do not follow the traditional diffuser concept.

Keywords—CFD, centrifugal compressor, micro-gas turbine, pipe diffuser, SLM, wedge diffuser.

NOMENCLATURE

r_2	Impeller tip radius, [mm]
r_3	Diffuser vane leading edge radius, [mm]
W	Width, [mm]
AR	Aspect Ratio
CFD	Computational Fluid Dynamics
LWR	Length to Width Ratio
PR	Pressure Ratio
$2-D$	Two-Dimensional
$3-D$	Three-Dimensional
Subscript	
ex	Outlet of the channel
th	Throat of the channel
Greek Letters	
ζ	Stagger angle, [degrees]
θ	Divergence angle, [degrees]

I. INTRODUCTION

A centrifugal compressor comprises an impeller, a diffuser and a volute, where the diffuser can be vaneless or vaned. Due to the geometry constraints in automotive applications, centrifugal compressors require very compact design and increased diffusion and pressure recovery in the diffusion section. Centrifugal compressor stages with vaned diffusers are characterised by their high efficiency [1] and smaller radial length, as compared to vaneless diffusers, but more challenging designs are due to the complex flow phenomena that the vanes experience at the exit of the impeller. Another aspect in automotive applications is the developing trend of high-pressure ratio technologies ($PR_{stage} > 3$), and the consequently high Mach numbers that a diffuser experiences which makes it harder to achieve a good diffuser design.

E. Ioannou, Pascal Nucara and K. Pullen are with the School of Mathematics, Computer Science and Engineering, City University London, London, EC1V 0HB, UK (e-mail: Eleni.Ioannou.2@city.ac.uk, Pascal.Nucara.1@city.ac.uk, K.Pullen@city.ac.uk).

The vaned diffuser in a centrifugal compressor stage can vary in shape consisting of airfoil (cascade), channel (wedge) or conical (pipes) passages. The cascade design is usually based on a standard airfoil shape. The wedge diffuser is 2-D design where passages are formed by discrete wedge-shaped vanes while the common pipe diffuser design employs 3-D conical passages drilled around the circumference. Among the established different diffuser designs, the pipe diffuser has been found to result some significant improvements in stage efficiency when the diffuser operates in high Mach numbers, being less sensitive in high inlet blockage when transonic flow approaches the vane leading edge [2], [3].

To design a highly effective vaned diffuser, many geometric parameters are need to be investigated, such as the number of the passages, the throat area, the Length-to-Width Ratio (LWR), the divergence angle. Any vaned diffuser design comprises a vaneless and semi-vaneless space and the diffusion part. The additional characteristic found in pipe diffuser is the pseudovaneless space that contains ridges formed by intersection of two pipes [4] that leads to the formation of the pipe leading edge geometry. This formation is claimed to handle the flow leaving the impeller, that is characterised by very high speed and unsteadiness, without large losses [5]. Therefore, the extend of the vaneless space, that defines the radial gap between the impeller trailing edge and the diffuser leading edge, plays an important role in the performance of the compressor stage, while another constraint is the effect on the noise and mechanical loading of the compressor [6].

The vaneless space can be defined based on differential equations [7], [8]; however, it is a complicated matter depending on a number of parameters and thus still remains a complicated design choice. This aspect has been investigated by many researchers, among them Clements and Artt [9] and Aungier [10] who almost agree that the recommended vaneless radius ratio r_3/r_2 is between 1.06 and 1.10 and between 1.06 and 1.10, respectively, in order to achieve optimum stage performance. Other published studies have reported higher optimum radius ratio values showing the dependency of the value on the compressor stage design. Jiang and Yang [11] have reported a favouring range between 1.15 and 1.20 mm radius ratio. Benini et al. [12] have concluded through an optimisation algorithm that for cases where the absolute Mach number of the flow leaving the impeller is quite high, a longer vaneless gap is needed to reduce the losses before the flow hits the vane leading edge. For higher radius ratios Benini et al. also compared the numerical results regarding the pressure ratio at the nominal speed against the experimental data finding that the steady approach is sufficiently accurate to predict the characteristics of the diffuser.

A number of other design parameters has been found to affect the design of efficient diffusers while only a few published studies contain relevant information [13], [14]. The aim of the present work is to compare different diffuser types in order to provide an optimum centrifugal compressor design for a micro-gas turbine engine focusing on fuel efficient vehicles. For that purpose, a steady state CFD approach has been applied and comparable results are presented between the wedge and pipe type diffuser. In the second part of the paper, the optimisation of the pipe diffuser by means of CFD is presented focusing on specific design parameters such as the radial length of the vaneless space in an effort to provide a better understanding on the way the optimum value is dependent on the characteristics of the compressor stage. An improved pipe design model is finally presented that exploits the 3-D freedom enabled by Selective Laser Melting methods. It is thought that the current study will enhance the understanding of the influence of specific geometric parameters on the development of efficient pipe diffusers that will be manufactured with innovative and optimal procedures.

II. Centrifugal Compressor Stage

For the purpose of comparison between different diffuser designs, a simplified cascade design was used as reference point. The initial three-dimensional shape of the centrifugal compressor impeller and cascade diffuser was designed by industrial partners with the use of CFD. Two diffuser types were investigated further and are presented in this paper, a channel diffuser (wedge), and a pipe diffuser with a characteristic "oval" shape of the throat area. However, emphasis is given on the pipe type diffuser and the wedge diffuser is briefly presented here.

The impeller contains 13 full blades without splitter blades, while the diffuser number of passages varies depending on the design type. The specifications of the stage are shown in Table I while Figs. 1, 2 show the main design parameters for the wedge and pipe diffuser, respectively.

TABLE I
Geometrical Data of Centrifugal Compressor

	Wedge	Oval Pipe
Number of passages	25	30
W_{th}	3.2	3.05
ζ	74	74
LWR	7.2	7.2
AR	0.715	0.75

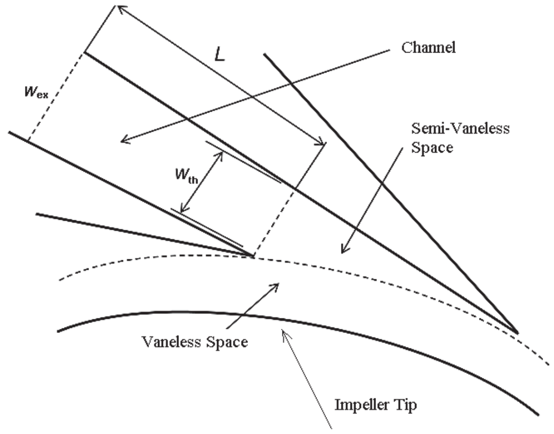

Fig. 1 Wedge diffuser sketch

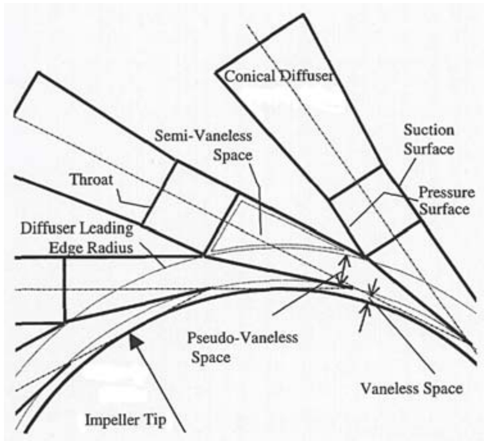

Fig. 2 Pipe diffuser sketch [16]

A. Wedge Diffuser

The throat area of the wedge diffuser was sized based on a combination of previous design (cascade) and literature survey. The number of wedge passages was set at 25 and the total throat area of the wedge diffuser is 183.2 mm^2. The throat area was designed to match the height of the impeller exit including the tip clearance. Another geometric constraint was the outlet diameter of the diffuser which is constant.

B. Pipe Diffuser

The initial pipe diffuser design was based on the design characteristics of the wedge diffuser. An oval type throat design was chosen based on the work published by Reeves [15] and Bennett et al. [16] in order to match the width and height of the throat of the wedge passages. To keep the throat area consistent with the wedge design, the number of passages is needed to increase due to the fixed size of the throat height. In effect, other parameters will also be affected by the increase of the number of passages. In order to keep the crucial design parameters in accordance with the wedge diffuser design, the throat area, the radius of vaneless space, the stagger angle and the outlet diffuser diameter remain the same as with the wedge diffuser. The LWR and divergence angle (2θ) are changed by increasing the length of the diffusion part of the pipe. The parameters of the baseline pipe diffuser against the wedge are shown in Table I.

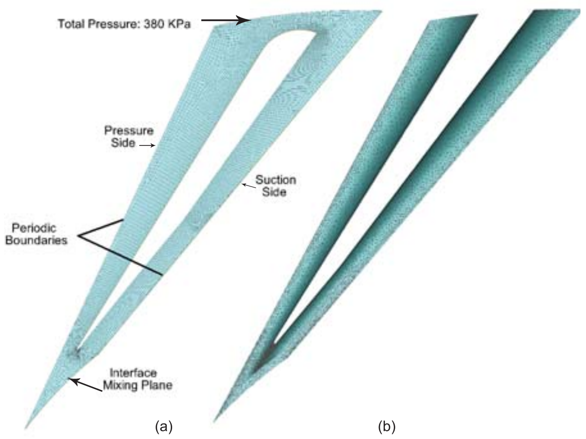

Fig. 3 Meshed wedge (a) pipe (b) diffuser sector

III. Numerical Model

For the purpose of the current work, a standard commercial CFD tool, ANSYS CFX, was used for the numerical simulation of the compressor stage. The mixing planes approach was applied at the interfaces between the impeller and diffuser domain with repeated periodic boundaries for each calculation. The steady state simulation is fundamental in the optimization of the diffuser where several time-consuming CFD calculations are needed to achieve the final solution. The influence of unsteady effects due to the impeller and diffuser interaction has not always been found to affect the mixing plane solution [17], [18] especially in cases such as the one presented here, where the coupling between the impeller and the diffuser is not very strong, with the vaneless radius ratio r_3/r_2 being between 1.085-1.1.

For the spatial discretisation, structured single-block H-type grids were used to mesh the rotating blade passage, while semi-structured grids were used to mesh the diffuser passages. Providing a grid independent solution, the overall grid for the compressor stage consisted of about 1250000 nodes including the tip clearance in the impeller model. Fig. 3 shows the computational grid for the wedge and pipe diffuser sectors with the corresponding boundaries. A standard $k-\epsilon$ model [19] was used to account for turbulence in the mean flow and a wall function approach was chosen to solve the boundary layer. Total pressure and total temperature values were applied at the inlet boundary. By varying static pressures at the exit boundary, computational flow points were shifted from choke towards stall.

IV. Results and Discussion

A. Comparison between Wedge and Pipe Diffuser

As discussed above, it is claimed that higher compressor efficiency can be achieved by adopting pipe diffusers, within the same geometrical constraints of a wedge diffuser, particularly under high pressure ratios [16]. In order to carry out the optimisation of the pipe diffuser, a comparative study was performed first, between the baseline 25-wedge diffuser design and the corresponding 30-pipe model. Fig. 4 shows the performance maps of the compressor stage for the two diffuser designs (wedge and pipe diffuser). From the plots, it is observed an increase in efficiency that is about 2.5% for the pipe compared to the corresponding wedge diffuser design at the design point.

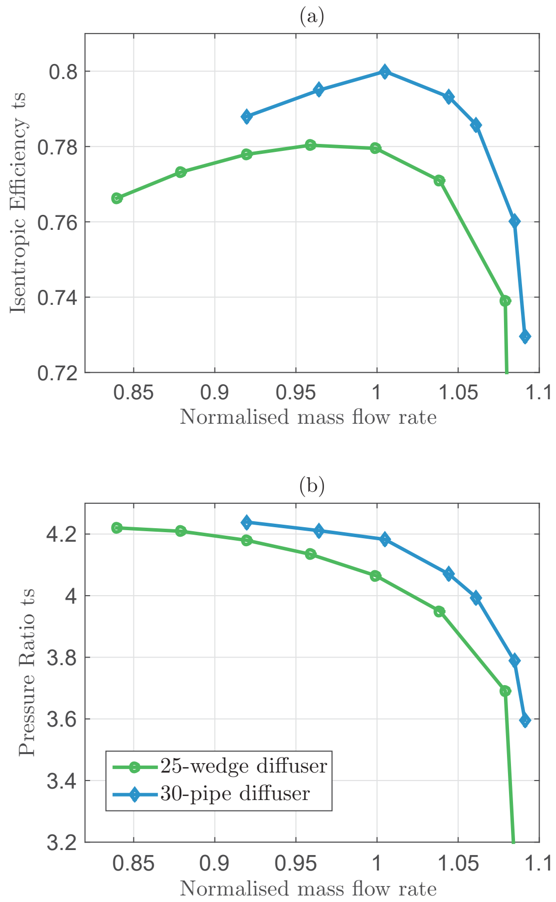

Fig. 4 Compressor performance map for different diffuser designs: (a) isentropic efficiency (b) total to static pressure ratio

The Mach number contours within the two diffusers near design point are shown in Fig. 5 for one diffuser section at midspan. The low velocity area downstream the wedge diffuser (Fig. 5 (a)) indicates some flow circulation which is not present at the pipe diffuser model (Fig. (b)). The distribution of the flow along the diffuser is controlled by many design parameters. Among those, the most crucial ones are the throat area of the diffuser and the diffuser leading edge. In this study, the total throat area was kept constant for the comparison between the diffuser models. In case of pipe diffuser though, the leading edges are created by the intersection of the pipes which result into the generation of counter-rotating vortices withing the diffuser pipe as shown in Fig. 6.

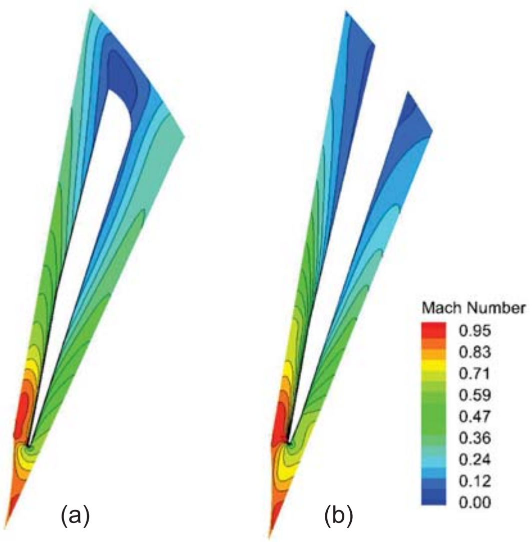

Fig. 5 Mach number contours on midspan plane: (a) wedge diffuser (b) pipe diffuser

The effect of the vortices at the leading edge of the pipe is not well understood as only a few studies deal with that aspect. The formation of the vortices leads to a better mixing of the distorted flow coming from the impeller exit making the flow more uniform. In effect, the flow propagates without exhibiting strong separation downstream allowing higher pressure recovery when compared to the wedge diffuser. The characteristic of the pipe leading edge is even more significant for cases such as the one presented in this paper, where the diffuser leading edge may experience high velocities exiting the impeller, hence the position and shape of the leading edge plays important role on the flow distribution and the consequent aerodynamic losses throughout the diffuser.

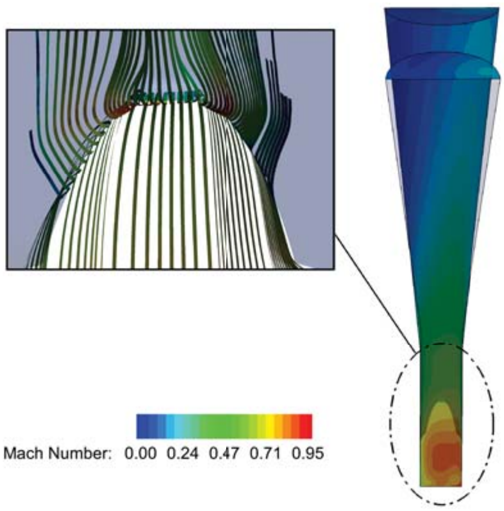

Fig. 6 Mach number contours with zoom at the leading edge vortices

B. Pipe Diffuser: Influence of the Diffuser Inlet-to-Impeller Exit Radius Ratio

As discussed above, the position of the leading edge can affect widely the diffusion process through the pipe passages. A parametric study was performed in order to investigate the optimum radius ratio of the diffuser leading edge radius over the impeller tip radius. The range variation was chosen based on the values that have been published in open literature (1.06-1.11) and was centred around the corresponding value of the wedge design, that was equal to 1.085. The scope of this study is to add some value to the published values regarding the choice of the radius ratio that results to the maximum pressure rise and minimum pressure losses into the pipe.

The compressor map, Fig. 7, shows the comparison between different values with the optimum one being between 1.085-1.11. Any further reduction of the radial length of the vaneless space will result in an efficiency penalty of about 3% for the lowest radius ratio value (1.06). The chosen radius ratio for the current study corresponds to 1.085 in order to be consistent with the initial choice made for the wedge design.

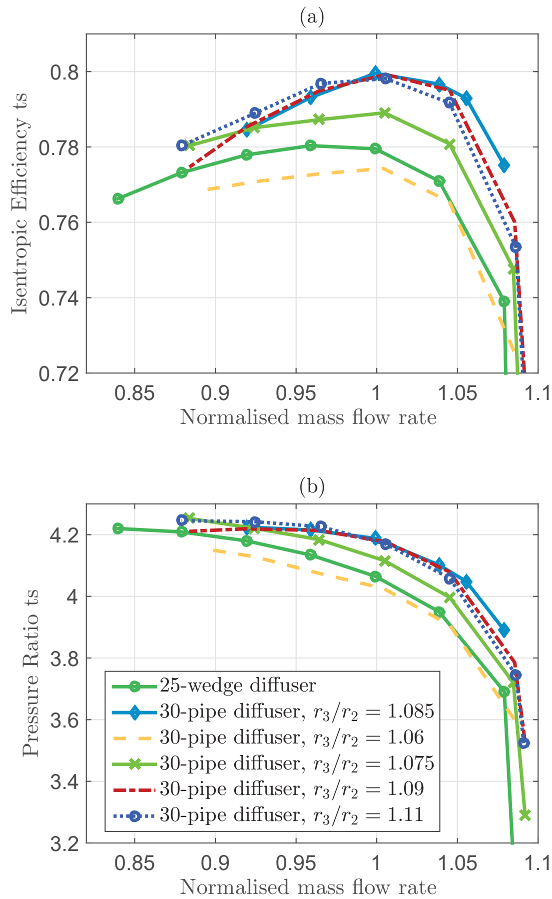

Fig. 7 Compressor performance map for various radius ratio: (a) isentropic efficiency (b) total to static pressure ratio

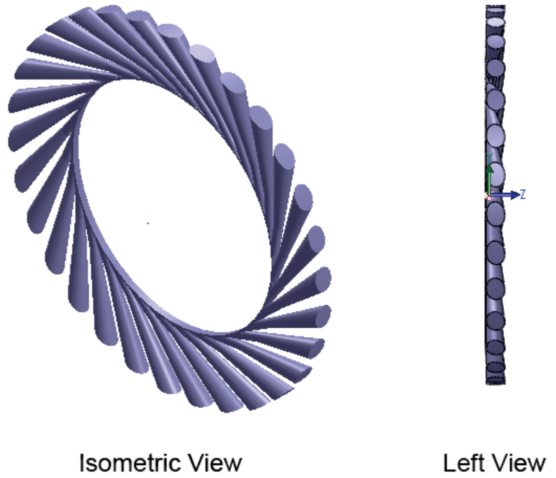

Fig. 8 Modified "tilted " 30-pipe geometry - isometric and left view

C. Pipe Diffuser: "Tilted" Pipe Design

Observation of the Mach number contours shown in Fig. 6 leads to the flow distribution along the pipe with a region of high circulation near the tip. It is apparent that an optimum design should result to uniform flow distribution to avoid any losses in the efficiency of the compressor stage. A pipe model in which the diffusion part is tilted towards the hub as shown in Fig. 8 was designed in order to adapt to the flow distribution.

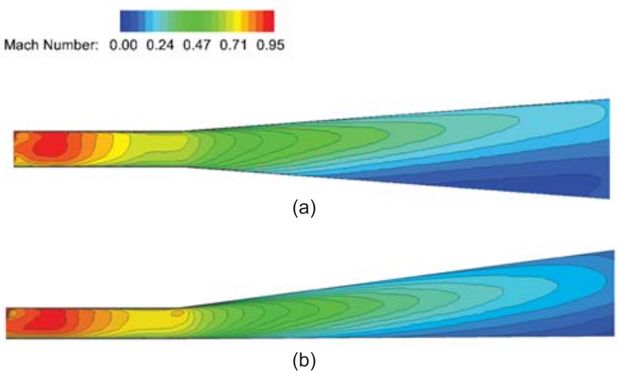

Fig. 9 Mach number contours on suction side plane: (a) baseline pipe diffuser (b) "tilted" pipe diffuser

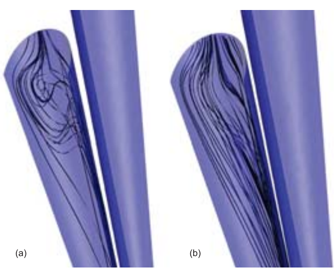

Fig. 10 Velocity streamlines near the exit of the pipe: (a) baseline pipe diffuser (b) "tilted" pipe diffuser

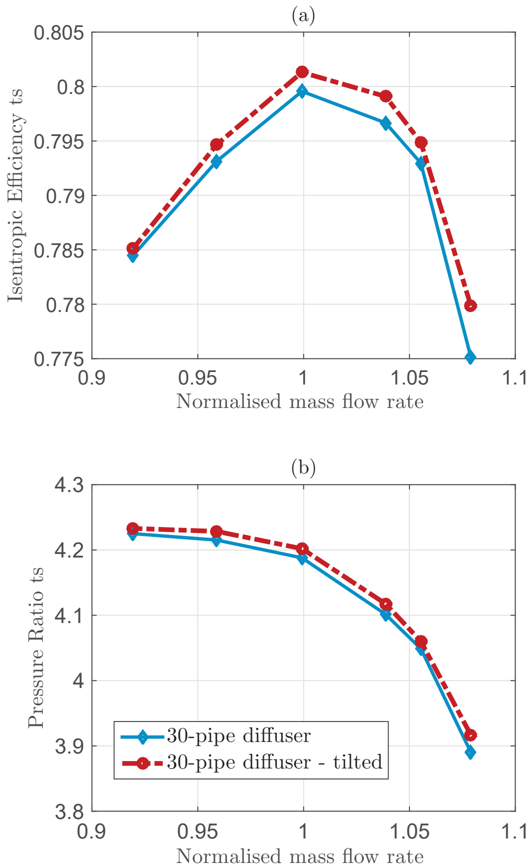

Fig. 11 Compressor performance map for the "tilted" pipe model against the baseline: (a) isentropic efficiency (b) total to static pressure ratio

In Fig. 9, the Mach contours show the distribution of the flow along the suction side of the pipe for the baseline (Fig. 9 (a)) and the "tilted" pipe model (Fig. 9 (b)), respectively. It is apparent that the "tilted" model (b) exhibits more uniform flow distribution along the pipe minimising the separation downstream compared to the baseline case (a). The effect on the losses is more clear in Fig. 10 where the velocity streamlines are shown between the hub and tip and towards the exit of the pipe. The circulation shown in the baseline model (Fig. 10 (a)) is eliminated when the pipe is tilted (Fig. 10 (b)) resulting in a more uniform velocity distribution and hence lower losses for the compressor stage.

The effects on the performance are described in Fig. 11 where the compressor map is presented for the isentropic efficiency and pressure ratio of the baseline pipe diffuser model compared against the "tilted" pipe model. The increase in efficiency that is achieved through the "tilted" pipe model corresponds to 0.25% at the design point resulting in a total benefit of 2.75% for the "tilted" pipe compared to the corresponding wedge diffuser design that was presented earlier in this paper.

V. Conclusion

In this paper, a pipe diffuser model for high-pressure ratio centrifugal compressor has been analysed by means of CFD. The benefits of the pipe diffuser model were investigated against the initial wedge diffuser design. The study was followed by an optimisation process of specific design parameters. The main conclusions are as follows:

- A 25-wedge diffuser model was used a benchmark for the comparison against a 30-pipe diffuser model within the same geometric constraints. The pipe diffuser design resulted a 2.5% increase of the isentropic efficiency compared to the original wedge diffuser design.
- A parametric study was performed to investigate the optimum value of the radius ratio. The range of variation was chosen according to the published values and was centred around the corresponding value of the original wedge design. Among the values that were tested, the range between 1.085-1.11 was found to be the optimum one for the current design achieving the maximum pressure rise. Any further reduction of the radius ratio showed a decrease in efficiency with the maximum decrease being about 3% that corresponds to the lowest radius ratio tested (1.06).
- The flow distribution into the diffuser part of the pipe revealed a separation are towards the tip of the pipe. A new pipe model was then investigated where the diffusion part is tilted towards the hub of the pipe. The results exhibited a decrease in the separation area with a consequent increase in efficiency equal to 0.25%. The model does not follow the traditional pipe diffuser model where pipes are usually drilled circumferentially as it is designed to exploit the freedom enabled by Selective Laser Melting methods.

Acknowledgment

The work has been part of the The SLaME (Selective Laser Manufacture for Engines) project funded by Innovate UK. The authors would like to thank Delta Motorsport for their support of this research.

References

[1] D. Japikse, *Centrifugal Compressor Design and Performance*, Concepts ETI Inc. 452 p., Vermont, USA, 1996.
[2] D. Kenny, *Supersonic Radial Diffusers*, AGARD Lecture Series, Vol. 39 VKI, Brussels, Belgium, 1970.
[3] P. M. Came, *The Current State of Research and Design in High Pressure Ratio Centrifugal Compressors*, Procurement Executive, Ministry of Defence, Aeronautical Research Council, C. P. No. 1363, 1977.
[4] D. R. Grates, P. Jeschke, R. Niehuis, *Numerical Investigation of the Unsteady Flow Inside a Centrifugal Compressor Stage with Pipe Diffuser*, Journal of Turbomachinery, Vol. 136, pp. 031012-1-13, 2014.
[5] D. S. Krige, *Performance Evaluation of a Micro Gas Turbine Centrifugal Compressor Diffuser*, Master Thesis, Stellenbosch University, March 2013.
[6] G. Pavesi, *Impeller Volute and Diffuser Interaction*, Design and Analysis of High Speed Pumps, Science and Technology Organisation, RTO-EN-AVT-143, pp. 6-1- 6-28, 2006.
[7] J. Schiff, *A Preliminary Design Tool for Radial Compressors*, Master Thesis, Lund University, Sweden, June 2013.
[8] M. Casey, D. Rusch, *The Matching of a Vaned Diffuser with a Radial Compressor Impeller and Its Effect on the Stage Performance*, Proceedings of ASME Turbo Expo, GT2014-25003, Dusseldorf, Germany, June 16-20, 2014.
[9] W. W. Clements, D. W. Artt, *The Influence of Diffuser Vane Leading Edge Geometry on the Performance of a Centrifugal Compressor*, ASME Gas Turbine and Aeroengine Congress and Exposition, 89-GT-163, Toronto, Ontario, Canada, June 4-8, 1989.
[10] R. H. Aungier, *Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis*, ASME Press 328 p., ISBN: 9780791800935, 2000.
[11] T. Jiang, T. T. Yang, *Improved Vane-Island Diffusers at High Swirl*, ASME, 82-GT-68, 1982.
[12] E. Benini, A. Toffolo, A. Lazzaretto, *Experimental and Numerical Analyses to Enhance the Performance of a Microturbine Diffuser,* Experimental Thermal and Fluid Science, Vol.30 (5), pp. 427-440, 2006.
[13] M. Olivero, *Evolution of a Centrifugal Compressor, from Turbocharger to Micro Gas Turbine Applications*, Doctoral Thesis, Technical University of Delft, Netherlands, 2012.
[14] G. Han, X. Lu, Sh. Zhao, Ch. Yang, J. Zhu *Parametric Studies of Pipe Diffuser on Performance of a Highly Loaded Centrifugal Compressor,* Journal of Engineering for Gas Turbines and Power, Vol.136 (12), 122604-1-10, 2014.
[15] G. B. Reeves, *Design and Performance of Selected Pipe-Type Diffusers,* ASME, 77-GT-104, 1977.
[16] I. Bennett, A. ourlidakis, R. L. Elder *The Design and Analysis of Pipe Diffusers for Centrifugal Compressors,* Proceedings of the Institution of Mechanical Engineers, Vol.214 (1), pp. 87-96, 2000.
[17] V. G. Filipenco, S. Deniz, J. M. Johnston, E. M. Greitzer, N. A. Cumpsty *Effects of inlet flow field conditions on the performance of centrifugal compressor diffusers: Part 1Discrete-passage diffuser,* Journal of Turbomachinery, Vol.122 (1), pp. 1-10, 2000.
[18] M. Inoue, N. A. Cumpsty *Experimental Study of Centrifugal Impeller Discharge Flow in Vaneless and Vaned Diffusers,* Journal of Engineering for Gas Turbines and Power, Vol.106 (2), pp. 455-467, 2009.
[19] N. N. Mansour, J. Kim, P. Moin *Near-Wall $k-\epsilon$ Turbulence Modeling,* AIAA Journal, Vol.27 (8), pp. 1068-1073, 1989.

Eleni Ioannou

Education:

- PhD (2015) School of Mathematics, Computer Science and Engineering, City University London, UK
- MEng (2009) Department of Mechanical Engineering, Aristotle University of Thessaloniki, Greece

Professional Experience:

- Research Associate (September 2014- Present) School of Mathematics, Computer Science and Engineering, City University London, UK

Conferences/Publications:

- E. Ioannou, A. I. Sayma, "Full Annulus Numerical Study of Hot Streaks Propagation in a Hydrogen-Rich Syngas-Fired Heavy Duty Axial Turbine", *submitted to 'Journal of Turbomachinery', paper No: TURBO-15-1207*
- E. Ioannou, A. I. Sayma, L.S. Wong, "Full Annulus Numerical Study of Hot Streak Propagation in Axial Turbine Stage", *ISABE, September 9-13 2013, Busan, Korea*
- Ch. Chourpouliadis, E. Ioannou, A. Koras, A. I. Kalfas, "Comparative Study of the Power Production and Noise Emissions Impact from Two Wind Farms", *Energy Conversion and Management, Volume 60, August 2012, Pages 233-242, ISSN 0196-8904*