Journal of Applied Sciences 12 (24): 2511-2518, 2012
ISSN 1812-5654 / DOI: 10.3923/jas.2012.2511.2518

CFD Simulation of the Intermediate Passage of Gas Turbine with Energy Promoters

[1]Hussain H. Al-Kayiem and [2]Tan Fok Hon
[1]Department of Mechanical Engineering, Universiti Teknologi PETRONAS,
Bandar Seri Iskandar, 31750 Tronoh, Perak, Malaysia
[2]Pipeline Engineer at Sarawak Operation, PETRONAS, Malaysia

Abstract: The bulky size of gas turbine has always become a great factor and limit in designing oil and gas offshore platforms. This study presents simulations results on the effect of energy promoters in an aggressive intermediate gas turbine diffuser. A typical S-shaped diffuser is modeled and simulated to become in FLUENT 6.3.26 assuming 2-D and incompressible flow. The promoters were selected with triangular shape of 25° frontal edge. The effect of energy promoters was investigated with various configurations to obtain the best height and position. The simulation was carried out with five different heights of 0.5, 1.0, 1.5, 2.0 and 2.5 mm and five different promoter locations in the upstream part of the diffuser. The exit static pressure recovery, which ultimately affects the diffuser's efficiency, was adopted as an indication for the diffuser performance. The simulation results show that the energy promoters works as intended but still far from reaching the benchmark efficiency of a normal/ideal diffuser. Among 5, 10, 15, 20 and 25° of location angles in the inlet bend, the 10° location was found to be the most effective location. The promoters with 2.0 mm height display the highest exit pressure recovery. Further simulation will be required in three dimensional, as well as experimental investigations to explore furthermore on the promoters contribution in enhancing the diffusers efficiency.

Key words: Gas turbine, diffuser, energy promoter, separation, pressure recovery, aggressive diffuser

INTRODUCTION

Gas Turbine (GT) design has always become competitive for low fuel consumption, meets the environmental codes, while keeping the GT light in weight and compact in size. This delivers great advantage in aero engines, which lowers the overall weight and structure integrity of the aircraft, as well as keeping the space efficient for related industry. Also, GT engines are widely used in oil and gas industry, to generate electricity power as well as operation of rotational machines for oil gas lift. The bulky size of gas turbine-compressor/generator set has always become a great factor and limit in designing oil and gas offshore platform. It also limits the flexibility of improving the existing oil and gas platform by exchanging different model of the GTs according to needs.

DIFFUSER FLOW

The best design idea to favor the function range of gas turbine is to reduce the size of gas turbine. A diffuser is located in between the High Pressure Stages (HP) and Low Pressure Stage (LP) turbine, Fig. 1, namely intermediate turbine diffuser with the function of pressure recovery. This diffuser's size increases with higher power output, due to higher combustor exit flow.

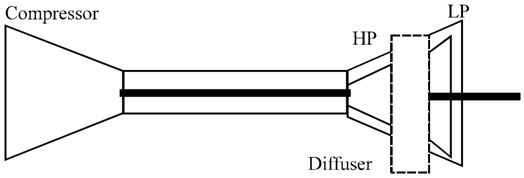

Fig. 1: Schematic representation of the double shaft gas turbine

The diffuser function is to increase the flow's static pressure. However, by shortening this diffuser while retaining its inlet and outlet size (increasing the cone angle), flow separation will occur, creating a boundary layer that will significantly reduce the diffuser's efficiency, thus the gas turbine engine's performance.

Numerous researches are carried out to study the diffuser's behavior to improve the GT performance. Mehta and Bradshaw (1979) used an aggressive diffuser as part of a wind tunnel design. They mentioned one of

---

Corresponding Author: Hussain H. Al-Kayiem, Department of Mechanical Engineering, Universiti Teknologi PETRONAS, Bandar Seri Iskandar, 31750 Tronoh, Perak, Malaysia

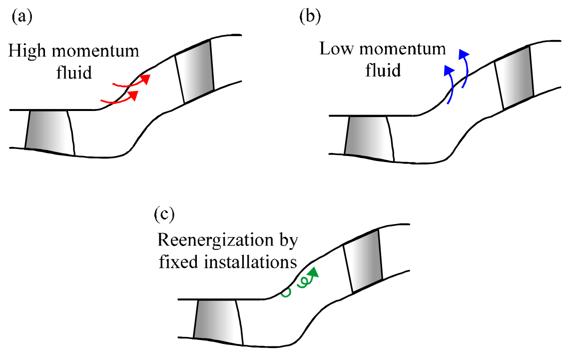

Fig. 2(a-c): Flow control mechanism (a-b) Active and (c) Passive measures for re-energization of boundary layer (Lord *et al*., 2000)

the problems that they have faced while using this type of diffuser is flow separation due to the cone angle of the diffuser is higher than 5° and he managed to solve it by introducing screens made up by woven wire gauze in the diffuser.

Lord *et al*. (2000) investigated on active or passive flow controls to design more aggressive transition duct geometries with larger radial offsets. The first type can either be energization of boundary layer by injecting high energy fluid or removal of low energy fluid from critical wall region, as in Fig. 2.

Lin (2002) performed a thorough review on low-profile vortex generator and their ability to prevent flow separation. The working principle is to transport high momentum fluid from the core flow into the boundary layer by means of stream wise vortices.

Fukudome *et al*. (2005) did an analysis on separation control of high angle of attack air foil for vertical axis wind turbines. One of the methods they used is by using turbulence promoters. They had concluded that the present of turbulence promoter is useful to modify the aerodynamic performance of the vertical axis wind turbine by altering the separation.

Sieker and Seume (2007) discussed that the power and efficiency of turbines are strongly depend on the performance of the exhaust turbine diffuser. They did an experimental analysis to relate the influence of rotating wakes on separation in the turbine exhaust diffusers.

Zulkefli and Ahmad (2010) had reported numerical simulation results of the effect of streamwise vortices on turbulent flow structure. Their objective in their simulation is to obtain the optimum parameter of sub-boundary layer vortex generator. They had used commercial code fluent 6.3^{TM} to simulate their model. They have stated two different types of flow control devices, which are passive and active control device.

Intermediate passageway and energy promoters geometry: Reducing the size of gas turbine can have many advantages. This includes increasing the stability of the shaft, where shorter shaft has better stability and balance and reducing its total weight. Other than that, this can contribute to more flexible position allocation for GT at offshore platform structure, as the space available is very limited.

Two patents held by general electric, namely Graziosi and Kirtley (2006) and Widenhoefer *et al*. (2009) introduces inter turbine diffusers with different type of method in eliminating flow separation. In both inventions, secondary air is injected to energize the boundary layer to prevent the separation. The air will be taken from the compressor section of the gas turbine, due to suitable static pressure ratio between suction port and the injection slot. Santer *et al*. (2010) explained that passive flow control is less complex than active flow control, because there is no need for handling additional fluid streams at unsteady flow rates. Merely the installation of fixed components at the right position would be very beneficial to re-energize boundary layer. He also installed a work package, namely EU project AIDA to evaluate the application of passive flow control devices in both compressor and turbine transition ducts. Low vortex generators have been designated for one of their super aggressive intermediate turbine diffuser setup, AIDA and TTTF at Graz University of Technology. These ducts shows fully separated flow on casing wall and therefore suitable for the study of passive flow control devices in order to show improvements after installation.

Therefore, the objective of the present work is to investigate the possibility of shortening the intermediate diffuser in a GT by CFD simulation. Passive and aggressive diffusers were modelled and simulate assuming 2-D incompressible gas flow using FLUENT commercial software. The installation of energy promoters with five different heights (0.5, 1.0, 1.5, 2.0 and 2.5 mm), at five different promoters location in the upstream part of the diffuser were investigated.

MATERIALS AND METHODS

For this study, CFD simulation was carried out using both GAMBIT version 2.2.30 for the diffuser design and FLUENT version 6.3.26 for simulation. The simulations is to prove the existence of flow separation as the diffuser's cone angle increases and reduce the diverse effect of the separation by introducing energy promoters. Accordingly, two main configuration were to be considered in the simulation, namely, the diffuser and secondly, the promoter.

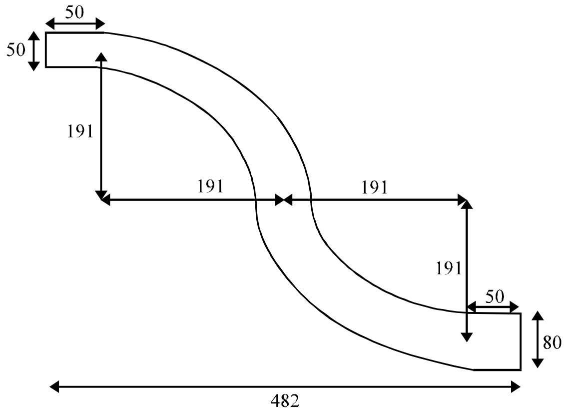

Fig. 3: The benchmark S-shape diffuser

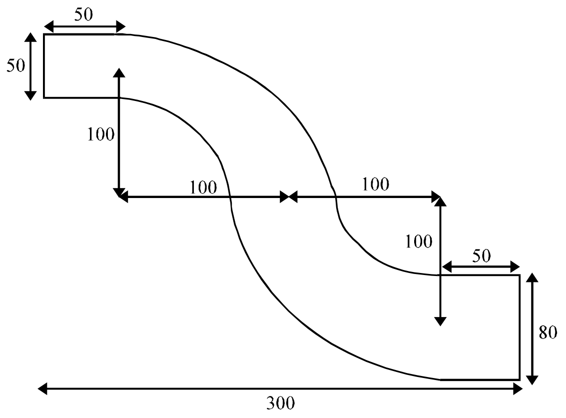

Fig. 4: The aggressive diffuser

The diffusers: Firstly, S-shape diffuser with normal cone angle was designed to create a benchmark for comparison, as shown in Fig. 3. The total length of the diffuser is 482 mm with inlet height of 50 mm and exit height of 80 mm. The radii of curvature of the bends are 191 mm. The total length of the stream wise from inlet to outlet is 600 mm. Then, the diffuser was modified by shortening its length to create another model of an aggressive diffuser, as shown in Fig. 4.

A very aggressive and separating duct design with an area ratio AR of 1.62 and L/h_{in} of 2.56 was chosen based on the recommendations of Gopaliya *et al.* (2011). The length was reduced from 482-300 mm, while the inlet and outlet were maintained with 50 and 80 mm, respectively.

The aggressive diffuser has 100 mm radii of curvature and the total stream wise total length of 314 mm.

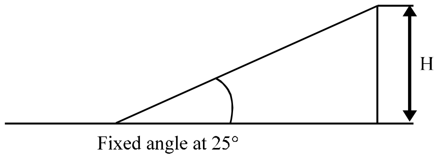

Fig. 5: Geometries of the energy promoters, H: Various heights of 0.5, 1.0, 1.5, 2.0 and 2.5

The promoters: The energy promoter were introduced and tested at different position and different energy promoter height. A vortex generator model has been adopted for the investigation of various configurations within a design of experiments for a flow controlled intermediate turbine diffuser by Wallin and Eriksson (2006, 2008). Four parameters with influence on the vortex generator performance were allowed to vary within the bounds:

- A non-dimensional location of the trailing edges relative to the baseline separation line $\Delta S_{VG}/h_{VG} = 0\text{-}11$
- The minimum height corresponds with the boundary layer thickness height
- $h_{VG} = 1.3\text{-}2.9$ mm
- Non-dimensional length
- $L_{VG}/h_{VG} = 25\text{-}5.5\times h_{VG}$
- Angle of attack (10-26°)

According to that, the final selected shape of the promoters in the present investigation is shown in Fig. 5. The promoters were selected with triangular shape of 25° frontal edge. Five different heights of 0.5, 1.0, 1.5, 2.0 and 2.5 mm were adopted in this investigated. And, five different promoters location in the upstream part of the diffuser were simulated.

SIMULATION AND EVALUATION PROCEDURE

Numerical implementation: The modeling of the diffusers and promoters geometries with various was carried out by GAMBIT version 2.2.30. The CFD simulation was carried out by FLUENT version 6.3.26 software. The field was meshed using the volume meshing tool. Only one element type was successfully applied that is the triangular quadrilateral type. The mesh was unstructured, as shown in Fig. 6. Errors were received when attempting to use any other element type. The mesh was refined gradually to prove the grid independency.

The boundary conditions applied for the simulation purpose are 200 m sec^{-1} inlet gas velocity, with 10% turbulence intensity.

At the outlet, the pressure was set at 0.0 Pa gauge scale. The solid boundaries have roughness of 0.01 and no slip shear conditions. When the governing equations of mass, momentum and energy conservation solved, the turbulence model selected is the k-ε model.

The performance evaluation: The exit static pressure is then recorded to identify the efficiency of the diffuser, which would be the ultimate goal for comparison, in determining the diffuser performance at various configurations and promoters' geometries.

The pressure rise coefficient given by:

$$Cp = \frac{p_2 - p_1}{q_1}$$

Where:

$$q_1 = \frac{1}{2}\rho_g v_1^2$$

as the dynamic head at inlet.

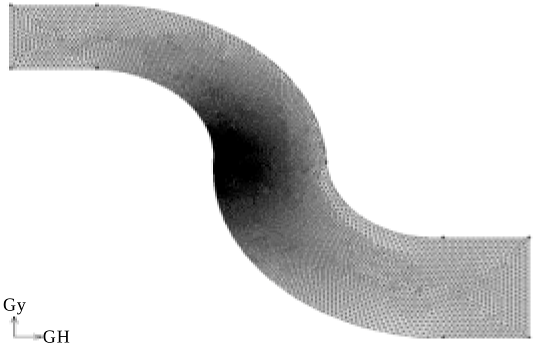

Fig. 6: The unstructured meshing with tetrahedral element type

Since T = 200°C, then:

$$\rho_g(\text{kg m}^{-3}) = 0.748$$

the diffuser's efficiency, is:

$$\eta_D = \left(\frac{C_p}{C_{pi}}\right)$$

where, C_{pi} is the ideal pressure raise coefficient, as:

$$C_{pi} = 1 - \left[\frac{v_1}{v_2}\right]^2 = 1 - \left[\frac{1}{A_R^2}\right]$$

RESULTS AND DISCUSSION

Results of different configurations are plotted accordingly. The results are presented in form of velocity vectors, pressure contours and static pressure profiles at inlet and outlet, at different promoter's positions and sizes.

Flow analysis: The velocity vector plots are used to indicate the flow behaviour. The flows in normal S-diffuser and aggressive diffuser are shown in Fig. 7 and 8, respectively.

Qualitative analysis of the flow could be dropped from this simulation results. The comparison will focus on

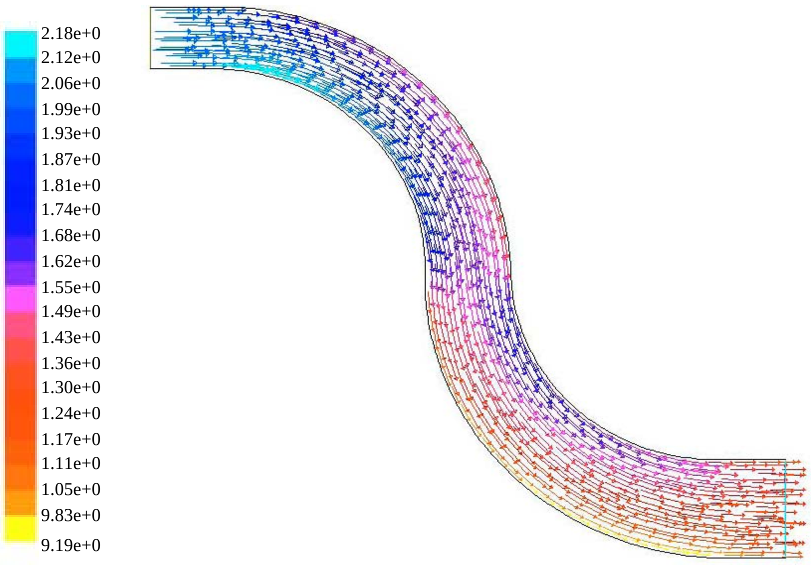

Fig. 7: Velocity vectors by magnitude of the flow in normal diffuser, V_{in} = 200 m sec^{-1}

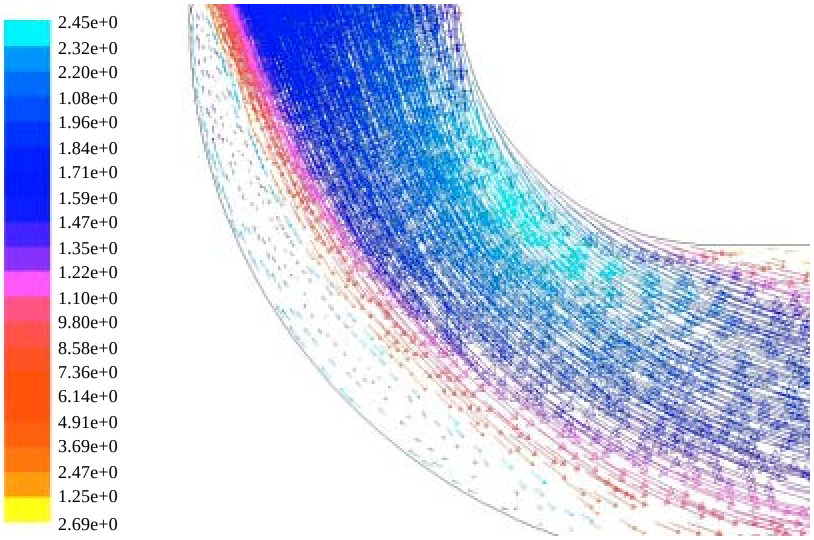

Fig. 8: Velocity vectors by magnitude in aggressive diffuser without promoters, V_{in} = 200 m sec^{-1}

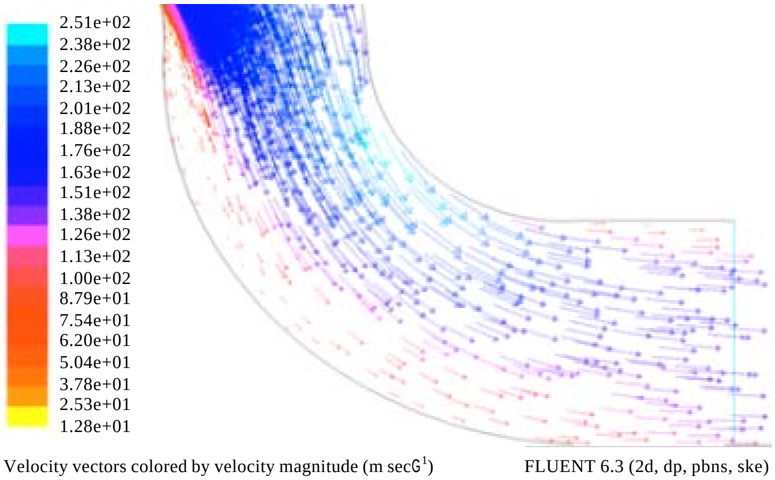

Fig. 9: Velocity vectors by magnitude of the flow in aggressive diffuser with promoters, V_{in} = 200 m sec^{-1}

the critical portion of the diffuser at the bend of the lower surface, where the flow decelerates dramatically due to the adverse pressure. The same flow passes through the normal diffuser smoothly, as in can be noticed in Fig. 5. While, by reducing the diffuser length from 482 mm to 300, a reversal flow and separation are taking place in the case of the aggressive diffuser, by passing the same gas flow. The resulted circulation bubble in the second case will effects the performance and reduces the pressure recovery at the exit of the diffuser. To maintain the advantage of reducing the length and increasing the pressure losses due to separation, an energy promoter (H = 1.5 mm) is introduced in the flow of the aggressive diffuser just upstream of the allocated separation starting point. The resulted simulation of the flow structure is shown in Fig. 9. It is quite clear that the promoter added momentum to the flow particles in the high bend portion and successfully eliminated the occurrence of the reversal flow.

Analysis of promoter's position: Five different locations of the promoters in the upstream of the bend have been simulated with promoter of 1.5 mm height. Those positions are at 5, 10, 15, 20 and 25° counterclockwise of the horizontal of the central point of the bend, as below:

The five positions have been simulated in the aggressive diffuser. The results of the static pressure are shown in Fig. 10. The predicted values of the pressure are in the centerline of the duct. The inlet static pressure is the same in all promoter position cases. At outlet, best performance to reduce the back pressure is gained at promoter position at 10° upstream the second bend, as shown in Fig. 11.

The exit static pressure is highest value at position of energy promoter at 10°. Positioning the promoter at angles of 25, 20 and 15, the exit static pressure shows slightly lower pressure recovery. However, in all cases of position angles, the promoters increase the momentum of the fluid to overcome the adverse pressure and result in smaller separation region as compared with cases of with no promoter installation. On the other hand, the exit static pressure bounces back when the position of the energy promoter exceeds 10 to 5°. The simulation analysis shows that the best contribution of the energy promoter is at 10° position.

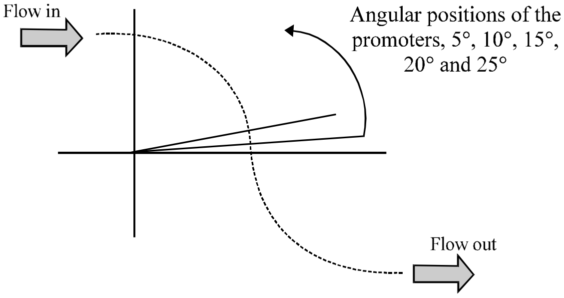

Fig. 10: The promoter positioning angle

Analysis of the promoter's size: Having the best position of the promoters found, which 10° is, the height effect of the energy promoter is tested at that angle. For all cases, the promoter frontal angle of attack is fixed at 25°, as recommended by Wallin and Erriksson (2006, 2008), while the height, H_{pr} is varied. and the length of the promoter is changing, accordingly.

Looking at Fig. 12, the energy promoter at height of 2.0 mm delivers the highest static pressure recovery. The other heights, 0.5, 1.0, 1.5 and 2.5 mm are resulted in lowers static pressure recovery, compared to the case of 2.0 mm height.

Efficiency analysis: Comparison of static pressure recovery between the normal S-shaped diffuser, aggressive diffuser and aggressive diffuser with energy promoters shows that energy promoters are able to increase the aggressive diffuser's efficiency. Simulation results summarized in Table 1 show a comparison of the predicted static pressure at the three cases of diffusers. The normal S-shaped diffuser is simulated, showing good diffuser's efficiency, at 71.2 %. The S-shaped diffuser's length is reduced by 38.4% and flow separation occurs, with efficiency at 16.4 % only. After the energy promoter is introduced, efficiency of the aggressive diffuser is increased to 38%.

But, it is nowhere near a benchmark normal diffuser's static pressure recovery. Some of the reason contributes to this pattern, may be due to over-aggressive diffuser design (very short centre-line length), which creates a large separated region that is not fully eliminate-able of separation. Other than that, due to insufficient resources on current existing aggressive diffuser design geometry,

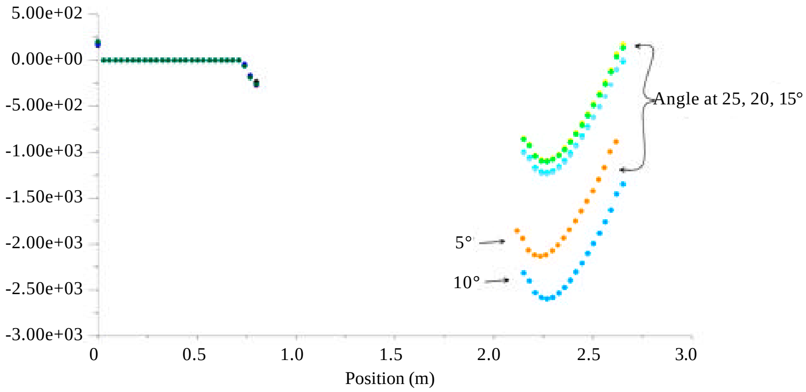

Fig. 11: Predicted static pressure results in the aggressive diffuser at various promoters positions, H_{pr} = 1.5 mm, V_{in} = 200 m sec^{-1}

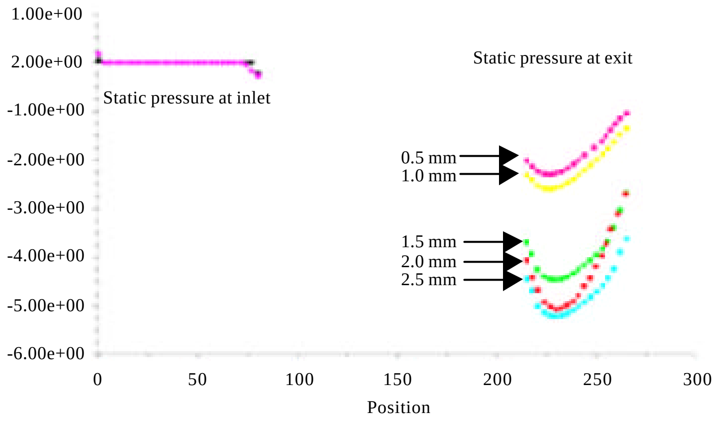

Fig. 12: Predicted static pressure results in the aggressive diffuser at various promoters heights, position angle = 10° and V_{in} = 200 m sec^{-1}

Table 1: Efficiency comparison between different cases of promoters, all cases with C_{pi} = 0.61

Configuration	V_1 (m sec^{-1})	V_2 (m sec^{-1})	P_{exit} (Pa)	Cp	η_D (%)
Normal diffuser	200	124	6500	0.43	71.2
Aggressive diffuser	200	159	1500	0.10	16.4
Aggressive diffuser with energy promoter	200	138	3500	0.23	38.4

a typical S-shaped diffuser was chosen instead. This raises the problem that, perhaps this diffuser design, was not in the effective range of energy promoter's utilization.

CONCLUSION

The gas flow through normal S-diffuser, aggressive S-diffuser with and without energy promoters have been simulated by FLUENT commercial software to investigate turbine size reduction by shortening the intermediate turbine diffusers downstream of HP turbine stages. The reduction of the normal S-diffuser length from 482 mm to be aggressive with 300 mm, have reduced the diffusion efficiency from 71 to 16%. Insertion of energy promoters has enhanced the aggressive diffuser efficiency to be 38%. This simulation findings are demonstrated the possibility of reducing the S-diffuser length but energy promoters are required to be inserted upstream of the separation bubble to reduce the reversal flow and enhance the static pressure recovery. The optimum promoters (with 25° frontal angle) height is 2.0 mm and location at upstream the second bend at 10° position angle.

Since the investigation is performed in 2-D, a similar simulation is recommended to be conducted in 3-D CFD to further investigate the flow behavior and its effect and relate them to this 2-D simulation.

ACKNOWLEDGMENT

The authors acknowledge Universiti Teknologi PETRONAS for the support to conduct the work and the presentation of the results in the present study.

REFERENCES

Fukudome, K., M. Watanabe, A. Iida and A. Mizuno, 2005. Separation control of high angle of attack airfoil for vertical axis wind turbines. Proceedings of the 6th KSME-JSME Thermal and Fluids Engineering Conference, March 20-23, 2005, Jeju, Korea.

Gopaliya, M.K., P. Goel, S. Parshar and A. Dutt, 2011. CFD analysis of performance characteristics of S-shaped diffusers with combined horizontal and vertical offsets. Comput. Fluids, 40: 280-290.

Graziosi, P. and K.R. Kirtley, 2006. High-area-ratio inter-tuurbine duct with inlet blowing. United States Patent, 7137245 B2.

Lin, J.C., 2002. Review of research on low-profile vortex generators to control boundary-layer separation. Prog. Aerosp. Sci., 38: 389-420.

Lord, W.K., D.G. MacMartin and T.G. Tillman, 2000. Flow control opportunities in gas turbine engines. AIAA paper AIAA-2000-2234.

Mehta, R.D. and P. Bradshaw, 1979. Design rules for small low speed wind tunnels. Aeronautical J., 83: 443-449.

Santer, C., E. Gottlich, A. Marn, J. Hubinka and B. Paradiso, 2010. The application of low-profile vortex generators in an intermediate turbine diffuser. ASME Paper GT2010-22892.

Sieker, O. and J.R. Seume, 2007. Influence of rotating wakes on separation in turbine exhaust diffusers. Proceedings of the 8th International Sumposium on Experimental and Computational Aerothermodynamics of Internal Flows, July, 2007, Lyon, France.

Wallin, F. and L.E. Eriksson, 2008. Design of an aggressive flow controlled turbine duct. ASME Paper GT2008-51202.

Wallin, F. and L.E. Erriksson, 2006. A tuning-free body-force vortex generator model. AIAA paper AIAA-2006-0873.

Widenhoefer, J.F., P. Graziosi and K.R. Kirtley, 2009. Multi-slot inter-turbine duct assembly for use in a turbine engine. United States Patent 7549282 B2.

Zulkefli, N.F. and K.A. Ahmad, 2010. Numerical simulation of the effect of streamwise vortices on Turbulent boundary layer. Proceedings of the Conference on Aerospace and Mechanical Engineering, World Engineering Congress, August 2-5, 2010, Kuching, Malaysia.