Jurnal
Teknologi

Full Paper

EFFECT OF TIP CLEARANCE ON THE FLOW FIELD OF THE MIXED FLOW TURBOCHARGER TURBINE

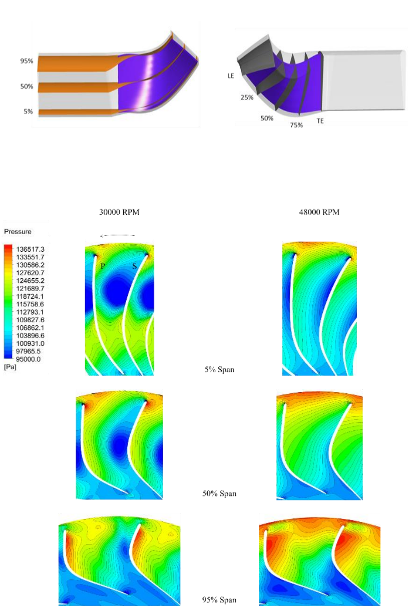
M. S. Kamarudin, M. Zulkeflee, M. H. Padzillah*

UTM-Centre for Low Carbon Transport in cooperation with Imperial College London, Universiti Teknologi Malaysia, 81310 UTM Johor Bharu, Johor, Malaysia

Article history
Received
21 January 2017
Received in revised form
31 May 2017
Accepted
17 August 2017

*Corresponding author
mhasbullah@utm.my

Graphical abstract



Abstract

Turbocharger is a device comprising mainly of a turbine and a compressor. The demand of turbocharger in the automotive industry increases as it significantly enhances the output power of an Internal Combustion Engine and reduces emission. The use of mixed flow turbine to replace the conventional radial turbine gives better impact since the turbine transient response increases and operates more efficiently at low velocity ratio. The flow behaviour inside the turbine affects the torque generation by the turbine. It is necessary for the turbine to have a specified clearance between the rotor tip and the casing of the turbomachine. This undoubtedly will contribute to clearance loss and mixing of leaked flow which produce disturbance to the exhaust flow. In order to investigate this effect, the validated Computational Fluid Dynamics method is chosen in order to replicate the flow field inside the mixed flow turbine. The simulation is carried out at the optimum operating condition which is at turbine total-to-static efficiency of 79% with inlet mass flow rate of 0.5kg/s. The flow inside the passage is plotted into pressure and velocity contours which are compared at 50% (30000rpm) and 80% (48000rpm) of the turbine design speed. The comparison between having 0% (no leakage) and 3% shroud tip clearance are then compared. Through the analysis, it is suggested that clearance leakage and flow separation cause disruption to the desirable uniform flow inside the turbine passage. The presence of Coriolis effect that resists the clearance leakage flow only at near leading edge of the rotor is observed. Furthermore, a low pressure region is perceived at rotor hub which absent in the radial flow turbine. These factors eventually reduce the performances of the actual turbine.

Keywords: Mixed-flow turbine, computational fluid dynamics, steady flow, tip clearance

1.0 INTRODUCTION

The modern diesel internal combustion engine (ICE) successfully converted average about 40% energy of the fuel burned into brake power while the remaining energy are lost in the form of aerodynamic drag, to mechanical friction and heat loss to surrounding via heat transfer or exhaust [1]. The major energy loss from the internal combustion engine is heat energy excreted through the exhaust. Thus, it is essential to recover the heat energy and turns it into useful work. In a turbocharged engine, the compressor will increase the density of air entering the combustion chamber which is directly obtained from the energy available from the turbine [2]. Appropriate design of turbine is needed to ensure the flow of exhaust inside the turbine behave accordingly and hence, maximize the extraction of energy in the turbine.

Experimental method of turbocharger enables researcher to investigate the turbine performance and numerical method seems to be a good complimentary to the experimental method. Numerical method for instances Computational Fluid Dynamics (CFD) provides the details of flow field inside the turbine [3]–[5]. The predicted flow field inside the turbine is not only used for analysis, but also used for turbine design development assistance. Eventually, the flow field inside the turbine will highly affect the torque generated by the turbine.

Based on research of tip clearance on the annular turbine rotor, it is discovered the presence of horse-shoe vortex for the zero clearance which disappears for 3% clearance that indicating that the pressure forces possess better influence than the viscous force for larger clearance [6]. In fact, the secondary flow and the tip clearance losses increase drastically at downstream of turbine stage.

Previous researches have been conducted to study the effect of clearance tip on different turbine. For example, Wells turbine with larger tip clearance had wider operating range of flow without stalling is quoted by Taha [7]. The author also states that as the tip clearance increases, the turbine peak efficiency will moves towards the high value of flow coefficient. This may happen at higher operating condition as the blade expansion compromises the larger tip clearance.

On the other hand, numerical analysis has predicted the performance in CFD at low turbine speed and also stated that the mean efficiency of the 4% impulse turbine reduced significantly compared of the 2% tip clearance [8]. This result shows that the turbine is sensitive to the tip clearance.

There are also interactions between backface clearance and tip clearance of the radial turbine when the leakage flow is strong at the upper radial region. He *et al*. [9] states that the pressure reduced drastically at the downstream side which are caused by combination of reduction of radial velocity and Coriolis force.

Based on the flow field analysis by Padzillah *et al*. [10], [11], there are existing of flow separation such a leakage flow inside the turbine passage of automotive turbocharger turbine. Furthermore, it is proven of the separation which is unique trait of mixed flow turbine as seen at the near rotor hub.

Thus, the purpose of this research is to analyze the flow behavior in the turbine passage with the shroud tip clearance with different rotor speed. The pressure and velocity distribution will be analyzed based on a validated numerical model.

2.0 METHODOLOGY

2.1 Geometry

The CFD method is adopted to produce the required flow field to complete the analysis. The software used for this research is Ansys CFX 16.0. The geometry used for numerical model comprise of two components which are rotor and vane to simulate the flow inside single passage. The rotor used in this experiment, 'Rotor A' is designed in house by Abidat [12] with chord length of 40.00mm. The vane used are composed of 15 blades with lean NACA 0015 profile developed by Rajoo [13]-[14].

The vane geometry had chord length of 22.3mm for hub and 26.3mm for shroud with leaning angle of 50° from the hub surface. Generation of vane requires three profile lines to plot in the Ansys TurboGrid.

The rotor geometry is generated with same method and due to intricate geometry of rotor, 9 profile lines are required for the finalized turbine design. The profile lines are generated by employing Bezier Polynomial as the guide to define the rotor camberline, hub and shroud [4], [15]. The coordinate obtained from the polynomial are converted to the Cartesian coordinate before imported to Ansys TurboGrid for the turbine generation.

The rotor and vane are assembled in Ansys CFX-Pre. The Table 1 shows the total nodes and element of the meshed geometry. The similar steps are repeated to produce identical geometry with different shroud tip clearance.

Table 1 The total nodes and elements of the turbine

Domain	Nodes	Elements
Rotor	552880	524874
Vane	114554	105128
All Domains	667434	630002

2.2 Boundary Conditions

The present study adopts a steady and an incompressible flow simulation. The boundary condition is defined before progressing to the simulation. The inlet flow is defined so the velocity component acting normal to the inlet plane. Constant static atmospheric pressure is applied at outlet plane, so will obtain the static pressure value at outlet. No-slip boundary condition is applied to the rotor and vane surfaces and the turbulence model used for the simulation is k-epsilon model.

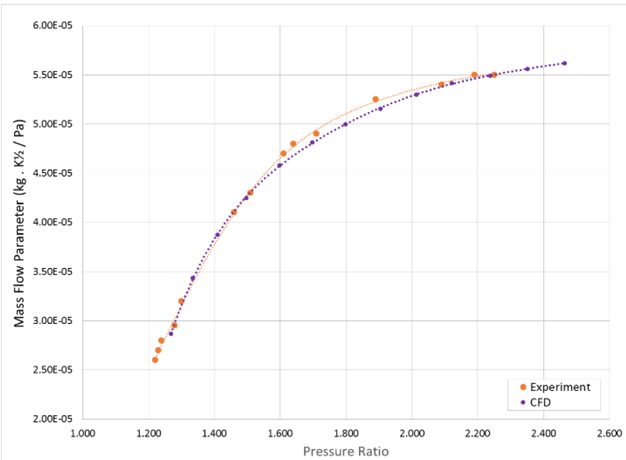
The simulation is carried out with varying mass flow rate instead of varying pressure ratio at inlet domain. The simulation is ranging from 0.2kg/s to 0.9kg/s with interval of 0.05kg/s and run at constant speed of 48000rpm for the validation process. After the validation process is done, the run is conducted at optimum condition which is 0.5kg/s at 30000rpm and 48000rpm for the flow field analysis process. The selection of these operating conditions are made due to abundance of experimental data for the purpose of model validations [16]–[20]. The analysis comprises of different shroud tip clearance of 3% and 0% which represent the experimental condition and ideal condition respectively.

3.0 RESULTS AND DISCUSSION

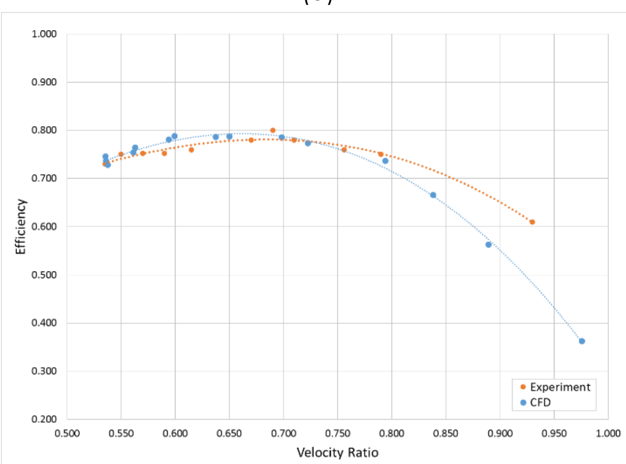
3.1 Model Validation

The validation is crucial to ensure that the numerical analysis is accurate enough to depict the real experimental set up. The simulation data is compared to the available experimental data by

Padzillah [15]. The validation is done based on the turbine performance parameters, which are mass flow parameter (MFP), pressure ratio (PR), efficiency and velocity ratio (VR).



(a)



(b)

Figure 1 Comparison of Experimental to CFD data (a) Mass Flow Parameter vs Pressure Ratio (b) Efficiency vs Velocity Ratio

Figure 1(a) shows the plot of MFP against PR at rotor speed of 48000rpm. The CFD data is plotted with the experimental data to view the trend of the data. It can be seen the data converged at most point before diverging at Pressure Ratio of 1.5 and reattached back at Pressure Ratio of 2.25. The overall Root Mean Square Deviation is estimated to be about 1.7%. Figure 1(b) shows the plot of efficiency against VR. The validation of the efficiency seems to be unstable and far to reach the experimental data due to the multiple parameters that effects turbine efficiency such as torque, temperature and pressure. The employment of fixed specific heat would lead to over and under estimation of turbine efficiency. Most of the CFD data diverge from experimental data but only converge at VR of 0.72. The validation is acceptable since the CFD data performed with sufficient accuracy and the numerical model can be used for the flow field analysis

3.2 Flow Field Analysis

The analysis of the pressure and velocity contour can be less complicated by projecting the contour into several planes to give the global view of the flow. The plane used for the flow analysis is the spanwise and streamwise plane. The orientations of the plane used for the analysis are depicted in Figure 2.

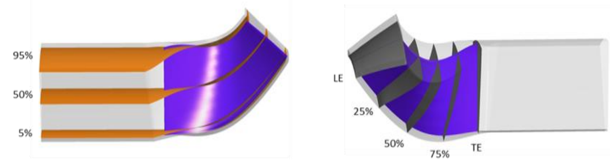


Figure 2 The selected plane for flow analysis (a) spanwise location (b) streamwise location

Figure 3 and Figure 4 show the pressure contour at each spanwise location of 3% and 0% shroud tip clearance respectively. The pressure contour at rotor speed of 30000rpm and 48000rpm are plotted side by side to enable direct comparison of pressure contour with respect to different rotational speed.

Based on observation on both 3% and 0% tip clearances, there is low pressure region exists at suction surface which is started at 5% span location and the magnitude reduces when the flow moves to 50% span location. The effect of low pressure region diminishes when the rotor speed increases at 48000rpm and absences at 50% span location.

In Figure 3, uneven pressure flow inside the passage presences but the flow becomes uniform when the rotor speed increases. The effect of tip clearance is clearly seen at 95% spanwise location when there is sudden pressure drop from 20% to 60% streamwise location before the flow reach the pressure surface. While in Figure 4, the flow is uniform towards the downstream of turbine passage due to absence of tip leakage effect. Hence, with uniform pressure drops along pressure surface and suction surface, towards the trailing edge the rotor will induce better torque generated at near shroud of turbine blade.

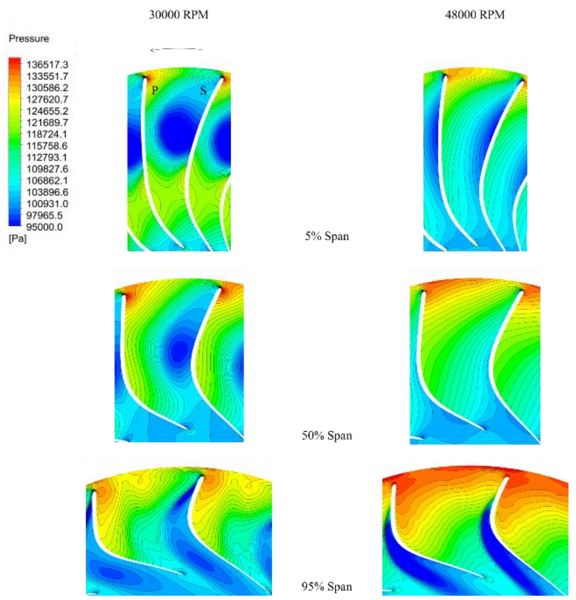


Figure 3 Pressure contour of each spanwise location of 3% shroud tip clearance

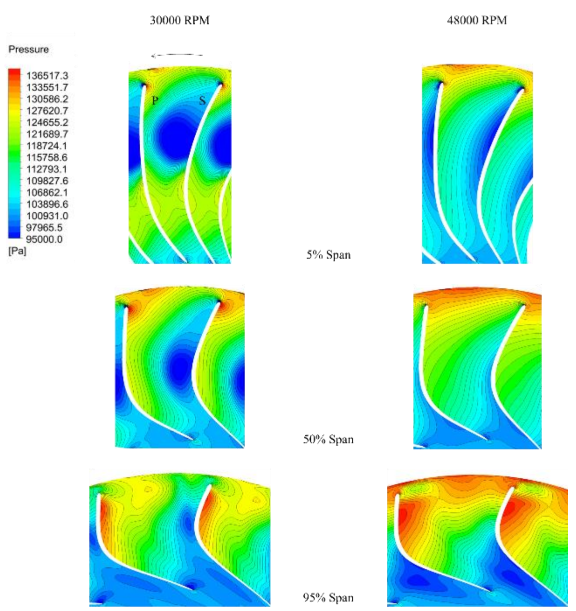


Figure 4 Pressure contour of each spanwise location of 0% shroud tip clearance

Figure 5 and Figure 6 show the pressure distribution at each streamwise location of 3% and 0% shroud tip clearance respectively. It is noted the plotted contours are according the flow facing towards the downstream of passage.

It is observed from Figure 5 and 6 that pressure gradually increases from suction surface to the pressure surface as it is common behavior of flow inside passage of any turbine. In Figure 5 and Figure 6, there is low pressure region originated from the turbine hub at the suction surface. The formation of low pressure region starts at 25% streamwise location and migrates towards the shroud as the flow goes downstream and finally moves to mid pitch of passage at trailing edge. There is also the formation of low pressure region caused by tip leakage which observed in Figure 5 at 95% span location but is not observed at leading edge and trailing edge of the rotor. As observed in Figure 6, the flow inside 0% shroud tip clearance resembles to the 3% shroud tip clearance but more uniform pressure drops from shroud to the hub.

Figure 8 shows the comparison of magnitude of pressure distribution between 3% and 0% shroud tip clearance are required at 95% blade span (its location is shown in Figure 7). Based on both Figure 8(a) and Figure 8(b), it is clearly seen that turbine with 0% shroud tip clearance model will produce better pressure loading on rotor blade as expected. It can be seen the pattern of pressure distribution gradually decreasing at pressure surface due to tip flow leakage in 3% shroud tip clearance model at both 50% and 80% design speeds. Besides, the flow separation effect can be seen clearly reduces the pressure loading at suction surface and the effect increase as the rotor speed increases. The increase flow separation is compensated with the high inlet pressure of turbine at high rotor speed.

It can be said that the flow separation existed at the rotor hub near the suction surface in all cases. The phenomenon is normally occur inside mixed flow turbine as mentioned by Palfreyman and Martinez-Botas [21].

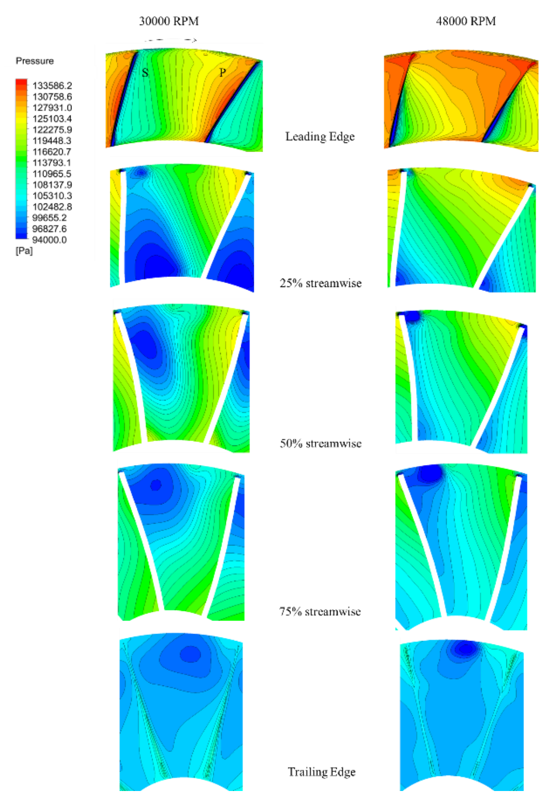


Figure 5 Pressure contour of each streamwise location of 3% shroud tip clearance

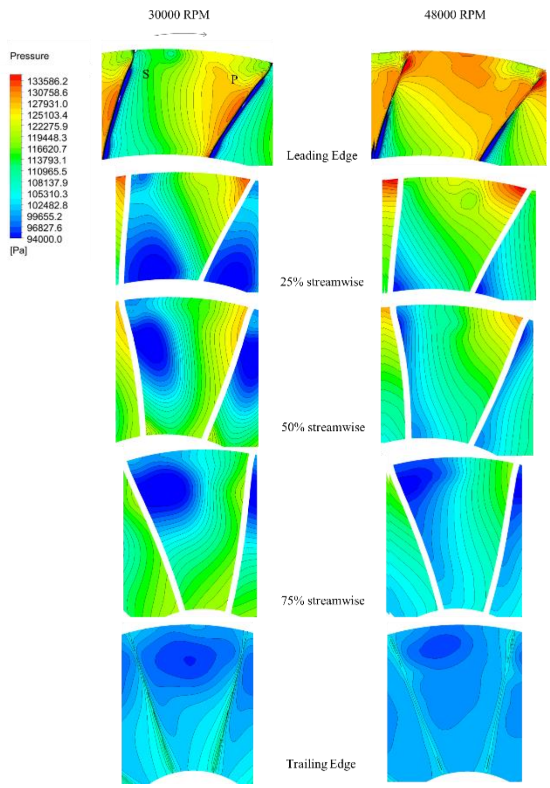


Figure 6 Pressure contour of each streamwise location of 0% shroud tip clearance

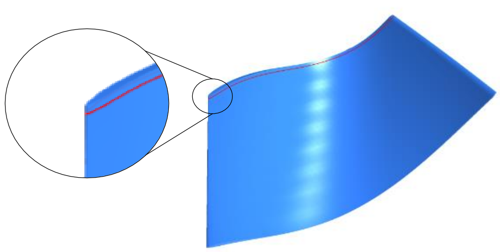
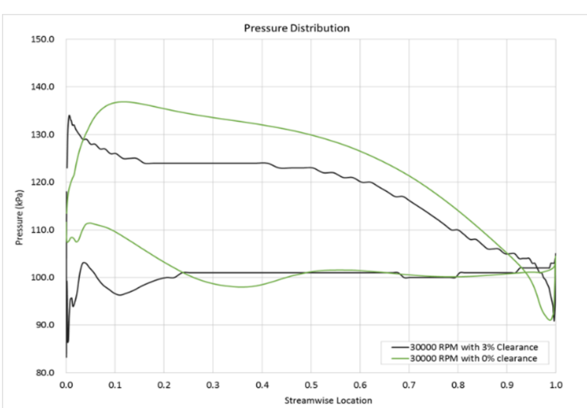
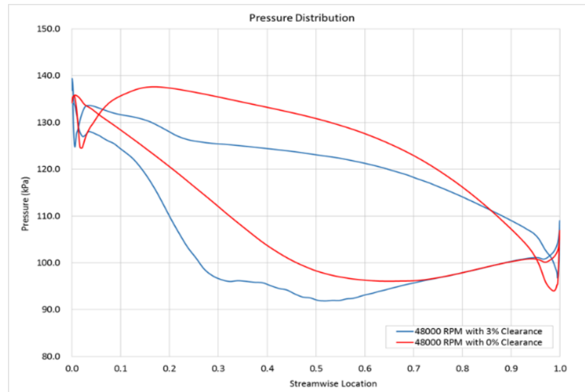


Figure 7 The location of plotted pressure distribution on the rotor blade



(a)



(b)

Figure 8 Pressure loading at (a) 30000rpm (b) 48000rpm

The understanding of pressure distribution in flow field is clearly stated previously. Another analysis is carried out to give better understanding of flow field behavior inside passage of mixed flow turbine. Hence, velocity contours are plotted with respect of difference in rotor speed and tip clearance.

There is formation of very low velocity region in Figure 9 that attached to the suction surface which clearly seen at 50% span location but lesser formation at 5% and 95% span location and this causing the flow splits starting from 20% streamwise location up to 50% streamwise location. Furthermore, it is observed that tip clearance effect at 95% span location which give to low velocity region and this effect increases with the rotor speed.

Based on observation in Figure 10, there is high velocity region starting from leading edge to trailing edge at 50% span location. This pattern occurs at 3% shroud tip clearance model but larger in magnitude for 0% shroud tip clearance model. Furthermore, the tip leakage absences at shroud tip. Besides that, it can be seen that there is several low velocity region at leading edge to 20% streamwise location. This may be due to the high pressure flow rush insides the turbine passage.

Based on Figure 11, there is low velocity region attached to the suction surfaces only at the 25% streamwise location. There is tip leakage effect in Figure 11 observed at 25% streamwise and existed until the trailing edge. Tip leakage effect worsens as the rotor speed increases. This leakage suppression occur up to 25% streamwise location which caused by Coriolis Effect. The Coriolis Effect is caused by the movement of flow from the hub to the shroud which occur at near leading edge due to geometry of rotor changes the flow from radial to axial as the flow moves downstream [15]. Thus, the leakage only occurs after 25% streamwise location due to absence of Coriolis Effect.

The Coriolis Effect also occur in the 0% shroud tip clearance model (shown in Figure 12). Unlike the 3% shroud clearance model, there is high velocity region originates at suction surfaces, gradually attached to shroud and mid pitch of passage. The region then migrates to the pressure surface at the trailing edge.

Thus, the main trait of the mixed flow turbine is the existence of Coriolis Effect that suppresses the shroud tip leakage effect from leading edge until the 25% streamwise location. Then, the tip leakage takes effect in disrupting the flow field inside a turbine passage. Inevitably, the tip leakage effect worsens as the rotor speed increases. Also the leaked flow from adjacent passage will created a flow separation which will cause the flow inside passage become uneven and thus reduces the torque generated.

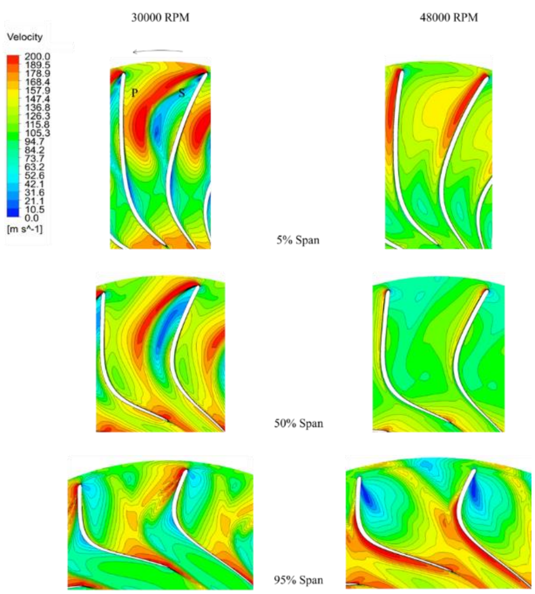


Figure 9 Velocity contour of each spanwise location of 3% shroud tip clearance

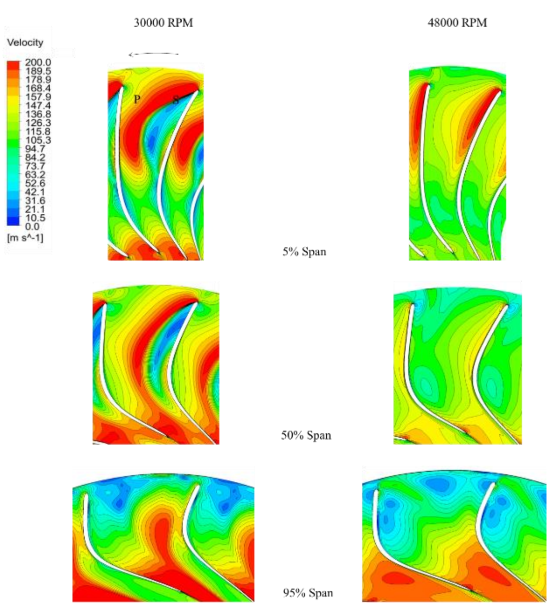


Figure 10 Velocity contour of each spanwise location of 0% shroud tip clearance

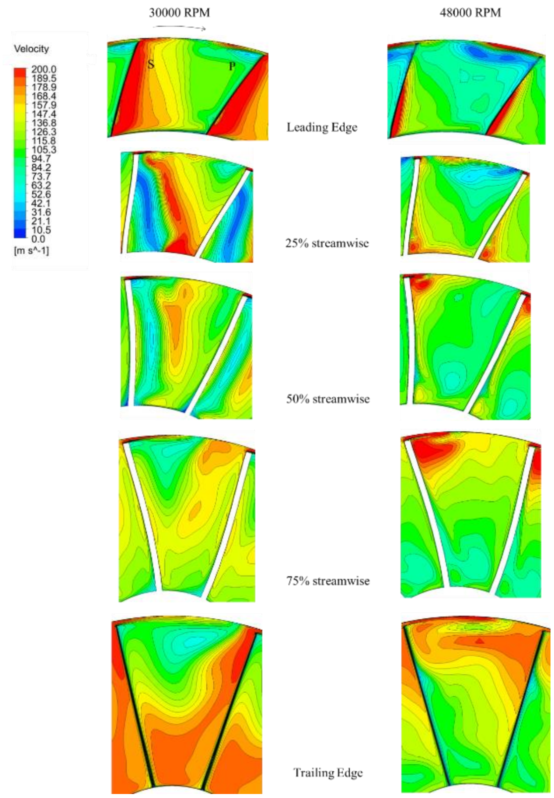


Figure 11 Velocity contour of each streamwise location of 3% shroud tip clearance

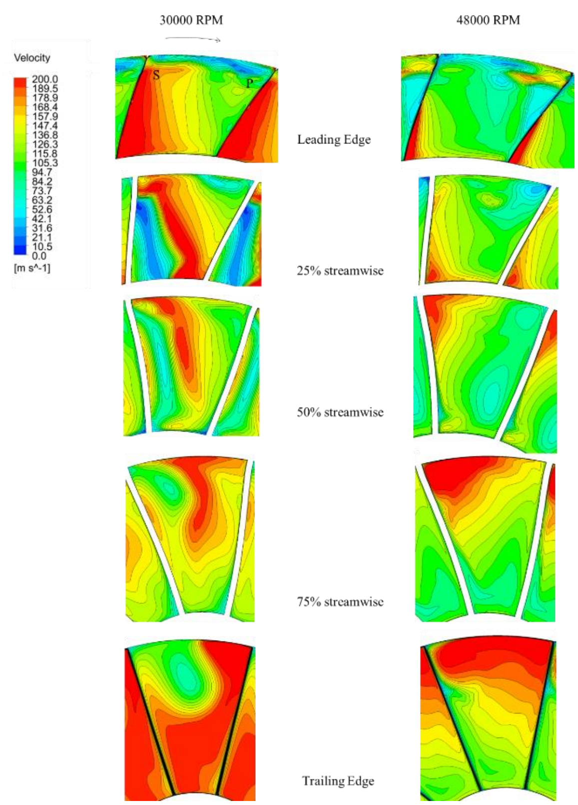


Figure 12 Velocity contour of each streamwise location of 0% shroud tip clearance

4.0 CONCLUSION

The validation exercise had been done prior to the flow field analysis. The computational data favours the experimental data with overall Root Mean Square Deviation of 1.7%.

The flow field analysis inside a mixed flow turbine passage is done at optimum operating condition which is the peak efficiency of the turbine (79%). The flow separation existed at the rotor hub near the suction surface in all cases are analysed which proven the specific characteristic of mixed flow turbine. The phenomenon of Coriolis Effect is proven and suppressing the tip leakage effect near the leading edge of rotor. The flow separation is detected due to tip leakage effect and high turning curvature inside turbine passage and the effect worsen as the rotor speed increases. These factors affect the turbine efficiency and also differs when the operating speed changes.

The tip leakage occurs at 3% shroud tip clearance model which producing region of low pressure. However, the tip leakage disappears in the corresponding 0% model as predicted. The existence of shroud clearance is a must in actual model since it allows the expansion of rotor blade which prevents the blade tip colliding with the shroud of the turbine. The shroud tip clearance loss is accounted during designing process of the actual turbine at expense of performance reduction when compared to an ideal clearance. Hence, through the CFD analysis of steady flow, it shows the actual model almost match the ideal model to certain degree and differs only in the flow field near the tip of turbine rotor.

Acknowledgement

The authors express gratitude to the Malaysian Ministry of Education (MOE) and Universiti Teknologi Malaysia FRGS research grant (R.J130000.7824.4F833) and facility.

References

[1] J. B. Heywood. 1988. *Internal Combustion Engine Fundamentals*. McGraw-Hill, Inc.

[2] W. W. Pulkrabek. 2007. *Engineering Fundamental of the Internal Combustion Engine*. Second Edi. Pearson.

[3] J. .-W. Lam, Q. D. H. Roberts, and G. T. McDonnel. 2002. Flow Modelling of a Turbocharger Turbine Under Pulsating Flow. *Preceedings IMechE Int. Conf. Turbochargers Turbocharging (Mechanical Eng. Publ. London)*. 181-197.

[4] D. Palfreyman. 2004. Aerodynamics of a Mixed Flow Turbocharger Turbine under Steady and Pulse Flow Conditions: A Numerical Study, Imperial College of Science, Technology and Medicine, University of London.

[5] P. Newton, R. Martinez-Botas, and M. Seiler. 2014. A 3-Dimensional Computational Study of Pulsating Flow Inside a Double Entry Turbine. *Turbomachinery*. 2D: V02DT42A026.

[6] C. Venkateswara, N. Sitaram, and M. Govardhan. 2002. Experimental Investigation of Tip Clearance Effects on Flow Field in an Annular Turbine rotor Cascade. *Indian Journal of Engineering and Materials Sciences*. 9: 424-431.

[7] Z. Taha, T. M. Y. S. T. Ya, and T. Sawada. 2011. Numerical Investigation on the Performance of Wells Turbine with Non-Uniform Tip Clearance for Wave Energy Conversion. *Appl. Ocean Res.* 33(4): 321-331.

[8] A. Thakker and T. S. Dhanasekaran. 2004. Computed Effects of Tip Clearance on Performance of Impulse Turbine for Wave Energy Conversion. *Renew. Energy*. 29(4): 529-547.

[9] P. He, Z. Sun, B. Guo, H. Chen, and C. Tan, Oct. 2012. Aerothermal Investigation of Backface Clearance Flow in Deeply Scalloped Radial Turbines. *J. Turbomach.* 135(2): 21002.

[10] M. H. Padzillah, S. Rajoo, and R. F. Martinez-Botas. 2016, Comparison of Flow Field Between Steady and Unsteady Flow of an Automotive Mixed Flow Turbocharger Turbine. *J. Teknol.* 78(8-4).

[11] M. H. Padzillah, S. Rajoo, M. Yang, and R. F. Martinez-Botas. 2015. Influence of Pulsating Flow Frequencies Towards the Flow Angle Distributions of an Automotive Turbocharger Mixed-Flow Turbine. *Energy Convers. Manag.* 98.

[12] M. Abidat. 1991. Design and Testing of a Highly Loaded Mixed Flow Turbine. Imperial College of Science, Technology and Medicine. University of London.

[13] S. Rajoo. 2007. Steady and Pulsating Performance of a Variable Geometry Mixed Flow Turbocharger Turbine. Imperial College of Science. Technology and Medicine.

[14] S. Rajoo and R. Martinez-Botas. 2006. Experimental Study on the Performance of a Variable Geometry Mixed Flow Turbine for Automotive Turbocharger. *8th International Conference on Turbochargers and Turbocharging*. 183-192.

[15] M. H. Padzillah. 2014. Experimental and Numerical Investigation of an Automotive Mixed Flow Turbocharger Turbine under Pulsating Flow Conditions. Imperial College London.

[16] A. Romagnoli, R. F. Martinez-Botas, and S. Rajoo. 2010. Turbine Performance Studies for Automotive Turbochargers. Part 1: Steady Analysis. *9th International Conference on Turbochargers and Turbocharging - Institution of Mechanical Engineers, Combustion Engines and Fuels Group*. 431-457.

[17] M. Y. Yang, M. H. Padzillah, W. L. Zhuge, R. F. Martinez Botas, and S. Rajoo. 2014. Comparison of the Influence Of Unsteadiness Between Nozzled and Nozzleless Mixed Flow Turbocharger Turbine. *11th International Conference on Turbochargers and Turbocharging, London, United Kingdom*. 333-345.

[18] S. Rajoo, A. Romagnoli, and R. F. Martinez-Botas, Feb. 2012. Unsteady Performance Analysis of a Twin-Entry Variable Geometry Turbocharger Turbine. *Energy*. 38(1): 176-189.

[19] N. Karamanis. 2000. Inlet and Exit Flow Characteristics of Mixed flow Turbines in Advanced Automotive Turbocharging. Imperial College of Science, Technology and Medicine. University of London.

[20] N. Karamanis, R. F. Martinez-Botas, and C. C. Su. 2001. Mixed Flow Turbines: Inlet and Exit Flow Under Steady and Pulsating Conditions. *J. Turbomach.* 123(2): 359.

[21] D. Palfreyman and R. Martinez-Botas, 2002, Numerical Study of the Internal Flow Field Characteristic in Mixed Flow Turbine. *Proc ASME Turbo Expo No. GT2002-30372*.