Jurnal
Teknologi

Full Paper

FLOW FIELD ANALYSIS OF AN AUTOMOTIVE MIXED FLOW TURBOCHARGER TURBINE

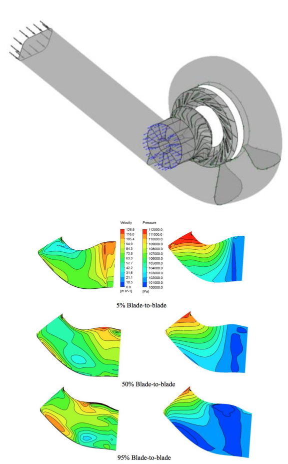
M. H. Padzillah[a*], S. Rajoo[a], R. F. Martinez-Botas[b]

[a]UTM Centre for Low Carbon Transport in Cooperation with Imperial College London, Faculty of Mechanical Engineering Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[b]Department of Mechanical Engineering, Imperial College London, London SW7 2AZ, United Kingdom

Article history
Received
20 July 2015
Received in revised form
23 September 2015
Accepted
22 October 2015

*Corresponding author
mhasbullah@utm.my

Graphical abstract



Abstract

Traditionally, the turbocharger has been an essential tool to boost the engine power especially the diesel engine. However, in recent years it is seen as an enabling technology for engine downsizing of all internal combustion engines. The use of mixed flow turbine as replacement for radial turbine in an automotive turbocharger has been proven to deliver better efficiency at high loading conditions. Furthermore, the use vanes that match the geometrical properties at the turbine leading edge could further increase its performance. However, improvement on the overall turbocharger performance is currently limited due to lack of understanding on the flow feature within the turbine stage. Therefore, the use of validated Computational Fluid Dynamics (CFD) in resolving this issue is necessary. This research attempts to provide description of flow field within the turbocharger turbine stage by plotting velocity and pressure contours at different planes. To achieve this aim, a numerical model of a full stage turbocharger turbine operating at 30000rpm under its optimum condition (pressure ratio of 1.3) is developed and validated. Results indicated strong tip-clearance flow downstream of the turbine mid-chord. Evidence of flow separations at the turbine leading edge are also seen despite turbine operating at its optimum condition.

Keywords: Computational fluid dynamics, pulsating flow, mixed flow turbine

1.0 INTRODUCTION

One of the early attempts to perform an experimentally validated 3-D computational on the turbocharger turbine was that of Lymberopoulos *et al*.[1] in 1988. In this work they used a simplified quasi-three-dimensional solution of the Euler equation. The radial and tangential components of velocity were fully solved, but the axial component was only treated to simulate the mixing of two streams. Although the model is not fully resolved in axial direction, they concluded that significant variations in flow properties exist around the exit circumference of the volute.

The use of CFD is not limited only to understand the flow field. Some of the researchers utilized this tool to assist development of new turbine concept. Barr and Spence [2] used CFD approach to develop a back swept turbine blade to improve turbine efficiency at low velocity ratio. They managed to obtain 2% efficiency improvement when compared to the original design. Barr and Spence also concluded that the back swept radial turbine performed less efficiently at high velocity ratio. Barr *et al*. [3] later extended their work to include experimental validations. Despite that, the experimental data obtained could only covered half the operating points and the lowest velocity ratio that was predicted to have the most significant improvement could not be captured.

As a turbocharger turbine often operates at off-design conditions, Walkingshaw *et al*.[4] conducted a

CFD analysis in a turbine that operated at velocity ratios as low as 0.34. They indicated that the tip clearance vortices that emerged from the vanes could also have favourable effect on the circumferential incidence angle distribution. Walkingshaw *et al*. indicated that the potential reason for flow distortion was more likely due to the wake flow originating from the vane trailing edge. However, within the blade passage, they observed strong separation on the suction side surface attributed to poor inlet flow alignment and that the flow struggled to reattach to the suction surface and therefore resulted in negative blade loading. However, the simulation works that were carried out have not been verified by experimental results.

Padzillah *et al*.[5] indicated that failure to precisely model the actual geometry could result in under prediction of static pressure distribution on the blade surfaces, thus the overall turbine performance. This results in erroneous prediction of flow pattern within the numerical domain hence the complex nature of the flow will not be accurately predicted. Although the application of CFD seems to have been able to sufficiently capture the flow details, one has to be careful on its viability. Denton [6] has outlined some limitations of CFD as a tool used for routine turbomachinery design. These include modelling errors due to numerical errors of finite difference approximations, unknown boundary conditions, and assumption of steady flow. However, Denton also indicated that CFD will remain as valuable tool for turbomachinery design and that the importance of validating numerical results with experimental data is paramount.

This work aims to provide description of flow field within the turbocharger turbine stage by plotting velocity and pressure contours at different planes by means of validated CFD.

2.0 METHODOLOGY

2.1 Experimental Setup

Figure 1 shows the schematics of the turbocharger test facility used in this research. The facility is located at Imperial College London for cold-flow testing and could be used for steady and pulsating flow testing. For the current research, only steady flow cases were tested. The compressed air for the test rig is supplied by three screw-type compressors with capacity up to 1 kg/s at maximum absolute pressure of 5 bars. The air is heated by a heater-stack to the temperature between 330K to 345K to prevent condensation during gas expansion in the turbine. The flow is then channelled into two 81.40mm limbs, namely outer and inner limb due to its relative position. This enables testing not only for single entry turbine but also for double or twin entry turbine. The mass flow rate in both limbs is measured using orifice plates. Downstream the orifice plates is a pulse generator originally designed by Dale and Watson[7] in 1986. The pulse generator enables actual pressure pulse in the exhaust manifold to be replicated in the facility with the frequency up to 80Hz. For the current research, the pulse generator is defaulted to 'fully open' position to allow maximum steady-state flow area. Downstream the pulse generator is the 'measurement plane', where all the parameters for the turbine inlet were acquired. This includes total and static pressure sensors and thermocouples.

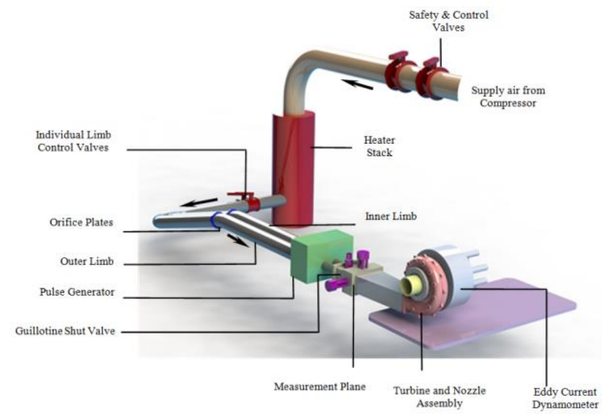


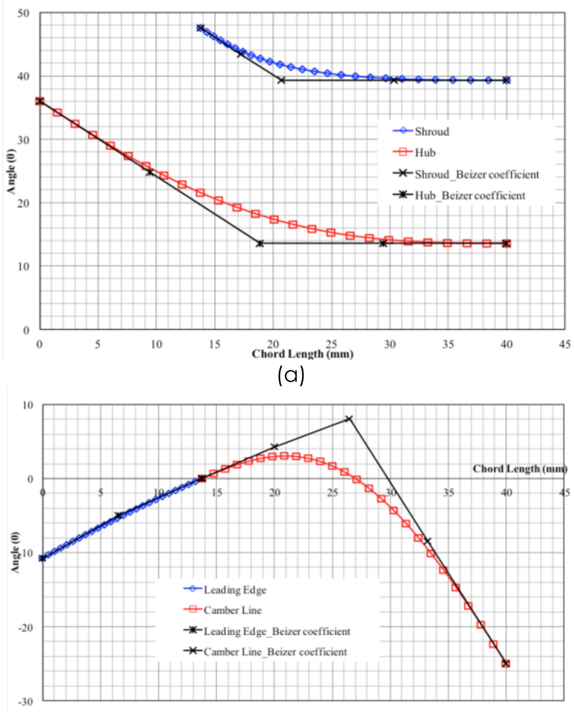
Figure 1 Imperial College 'cold flow' turbocharger test facility

The turbine is attached to a 60kW eddy current dynamometer designed by Szymko[8] where it is placed on a gimbal bearing. The reaction force on the dynamometer assembly is measured by a 20kg load cell where the torque is then calculated. The dynamometer also places a high flow rate water cooling system to disperse excessive heat absorbed by the magnetic plate. In addition, an optical sensor for instantaneous speed measurement is also installed within the dynamometer assembly.

2.2 Numerical Setup

The simulation works conducted in this research were executed using commercial software Ansys CFX 14.1. The 3-Dimensional turbocharger turbine geometry consists of 5 main components which are the inlet duct, a modified Holset H3B turbine volute [9], 15 NACA 0015 profiled vanes and a mixed flow turbine with 40mm chord length. The inlet duct and the volute were constructed using Solidworks and meshed using Ansys ICEM CFD. For the nozzle stage, 15 lean vanes were constructed by importing 3 profile lines into TurboGrid software where structured hexahedral meshed is automatically generated. Similar method is used to mesh the mixed flow turbine except 8 profile lines are needed due to its more complex geometry. The profile lines of the turbine blade were created using Bezier polynomial where its control points are shown in Figure 2. The turbine used is an in-house turbine designated Rotor A created by Abidat[10] in 1991 for application in high loading operations.

Subsequently, all meshed components were assembled in Ansys CFX-Pre as shown in Figure 3. The interfaces between each component are specified during this stage. The interfaces between inlet duct and volute, and also between volute to vane are specified as general connection. Frozen rotor interface is specified between vane and rotor domains. For steady state simulations, the use of frozen rotor interface is well justified as it will not impair accuracy of calculated model [11].



(a)

(b)

Figure 2 Development of turbine geometry using Bezier Polynomial

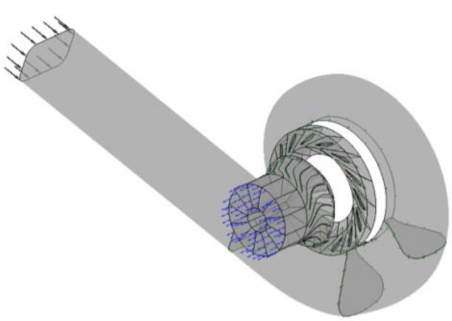


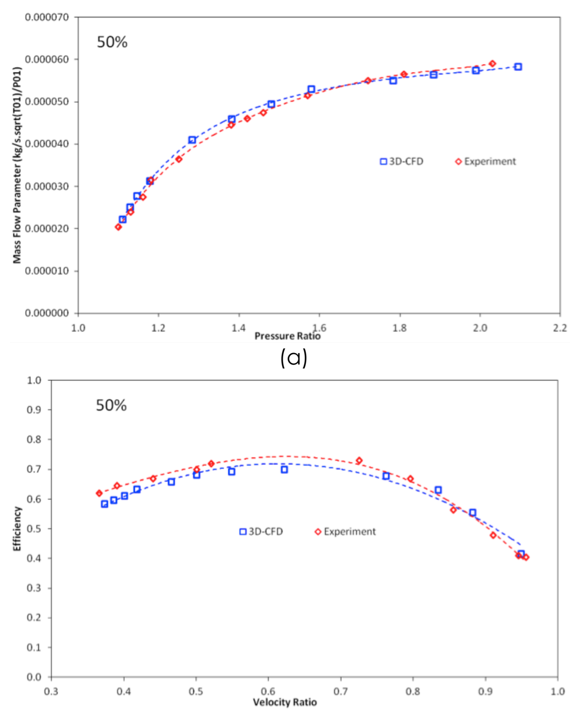
Figure 3 Assembly of domain in CFX-Pre

At the domain inlet (inlet duct), various total pressure and total temperature are specified according to the turbine operating condition. The direction of inlet flow is defined so that the only velocity component that exists is normal to the inlet plane. The values for inlet boundary are taken directly from experimentally acquired values at the measurement plane. The outlet boundary condition requires the static pressure value. For this purpose, a constant atmospheric pressure is applied at the domain outlet. No-slip boundary condition is specified at all walls including vanes and rotor blades. The rotor speed is set to 30000rpm which is equivalent to 50% design speed. This particular speed is selected due to availability of extensive experimental data which is valuable for model validation.

3.0 RESULTS AND DISCUSSION

3.1 Validation Exercises

Before the analysis, the computed models are verified with experimental results. This procedure is done by computing the turbine performance parameters which are the turbine mass flow parameter (also known as swallowing capacity), pressure ratio, efficiency and velocity ratio from CFD results and comparing them with available experimental data.



(a)

(b)

Figure 4 Comparison between CFD and Experimental data of (a) Mass Flow Parameter vs Pressure Ratio and (b) Efficiency vs Velocity Ratio

Figure 4(a) shows the plot of mass flow parameter against pressure ratio for the turbine rotating at 30000rpm. Both CFD and experimental data are plotted on similar axis to enable direct comparison. From the figure, it can be seen that the model used in this research are able to capture the trend and magnitude of the turbine swallowing capacity well for the entire operating range. The RMS of the deviation from experimental data is recorded to be 2%.

The validation procedure involving turbine efficiency has been proven to be more difficult since efficiency is a derived parameter. Therefore, it depends on accuracy of several calculated parameter such as mass flow rate, torque, temperature and pressure. In addition, use of fixed values such as the turbine speed and specific heat ratio could also lead to over or under prediction of turbine efficiency. Figure 4(b) shows the comparison between experimental data and CFD prediction of the turbine efficiency against velocity ratio. It can be seen that CFD tend to under predict the efficiency value for velocity ratio less than 0.85. The maximum deviation is recorded to be 5 efficiency points at velocity ratio of 0.36. The RMS deviation recorded for this plot is 2 efficiency points. Minimal RMS for both plots indicated that the developed model has achieved sufficient accuracy and as such could be used for further analysis.

3.2 Flow Field Analysis

While analysing the flow field within the rotor stage, it is also useful to scrutinize the flow development by plotting the velocity and pressure contour on a few planes of specific orientations, namely streamwise, spanwise and blade-to-blade planes. Each orientation of the plane is represented in Figure 5. The employment of these planes would enable a more global view of the flow field behaviour bounded by the shroud wall, the hub, the suction surface as well as the pressure surface.

The relative velocity and pressure contour of the streamwise planes is plotted side by side In Figure 6. It is also worth noting that the contour in Figure 6 is plotted such a way that that an observer is facing towards the upstream incoming flow. With regards to the velocity contour at the rotor leading edge, there exist a low velocity flow close to the pressure surface between 50% span and the shroud wall. As the flow moves further downstream to 50% streamwise location, the low velocity flow migrates closer to the hub (about 30% span) and also moved towards the mid-pitch region. Subsequently, at 75% streamwise location, the low velocity region has completely attached to the suction surface where it later moves towards the shroud wall as the flow reaches the rotor trailing edge.

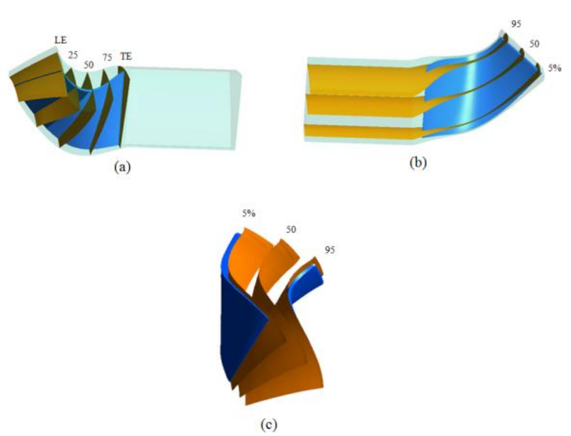


Figure 5 Locations of pressure and velocity contour in the rotor stage for (a) streamwise plane, (b) spanwise plane and (c) blade-to-blade plane

Another interesting observation that can be seen in Figure 6 is the development of the tip leakage flow. At the leading edge as well as at 25% streamwise location, one can hardly notice any development of the tip leakage flow. However, from 50% onwards, it can be clearly seen that the tip leakage flow has started to show its effect on the overall distribution of velocity in the particular plane as it emerged at the suction side near the shroud wall, and continues to exist until the flow reaches trailing edge. The suppression of tip clearance effect on the velocity contour from the leading edge up to 25% streamwise location is attributed to the Coriolis effect [12].

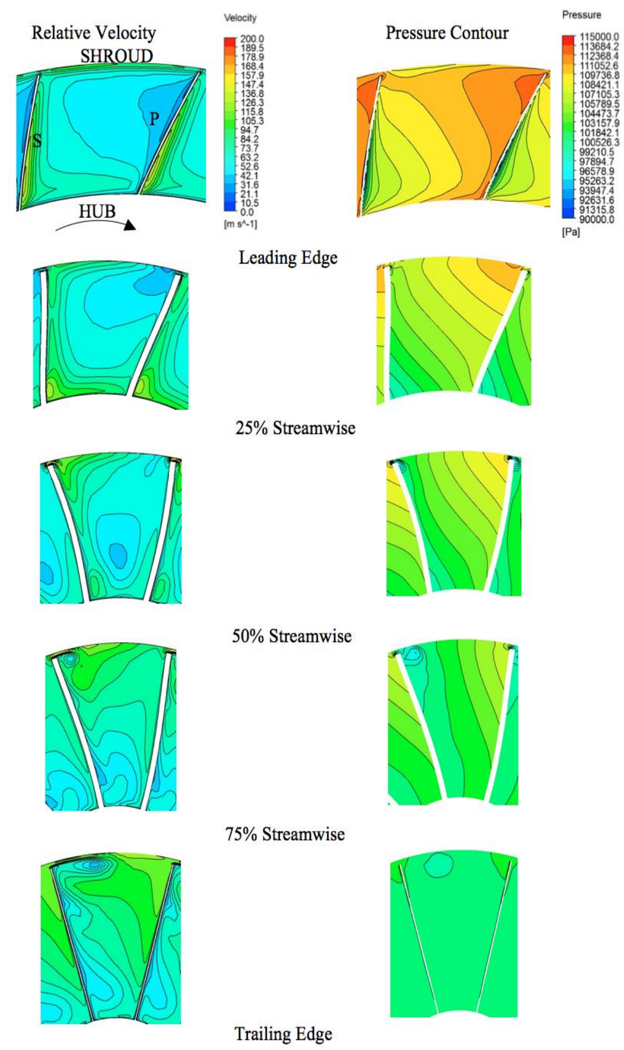


Figure 6 Pressure and velocity contour for each streamwise plane

The corresponding pressure contour for each streamwise plane is plotted on the right hand side of Figure 6. In general, for all streamwise plane except at trailing edge, it can be seen that the pressure gradually increase from suction to pressure surface.

This behaviour is generally expected in any type of turbine. It can also be seen that there exist a very low pressure region at the suction surface of the rotor leading edge which is clear indication of flow separation despite optimum incidence flow angle. With regards to the tip leakage flow that has been discussed before, it can be seen that the influence begins to increase and inevitably has its effect on the pressure distribution starting at 50% streamwise location. The tip leakage flow that enters the turbine passage created a low pressure region as the flow velocity is high and potentially induces flow separation close to the suction surface. Therefore the low pressure region which is originally recorded close to the suction surface hub (25% streamwise) has migrated towards the shroud region as the flow proceeds downstream. Finally, as the flow exits the turbine, the pressure becomes stable and close to the ambient pressure where hardly any spatial variation in the rotor passage is noticed.

The relative velocity and pressure contour on three spanwise planes are plotted in Figure 7. The contour is plotted such that one is looking down towards the hub of the rotor. At 5% spanwise location which is very close to the hub (about 1 mm from the hub), it can be seen that there is a high velocity region attached close to the suction surface near inlet of the rotor. Generally, in a rotor passage, one would expect the flow velocity to increase as it reaches the rotor exit. While the statement holds true as the whole system is taken into consideration, it is however not the case for the flow field close the hub. There is even a low velocity island exists at about 75% streamwise location close to the pressure surface. The irregular observation as opposed to commonly accepted theory is mainly attributed to a combined effect of the flow turning very sharply close to the hub as well as the hub end of the wall shear stress.

At mid-span, a small separation region is developed at the pressure surface near the rotor inlet (labelled as X in Figure 7 at 50% span). The most severe and highly irregular flow field is recorded at 95% spanwise location. Two major separation regions are detected. The first one occurs close to the turbine leading edge and up to 25% streamwise location near the pressure surface (labelled as Y in Figure 7).

Meanwhile, the second and more severe separation occur at about 45% streamwise location close to the suction surface (labelled as Z in Figure 7). The second separation developed further downstream where it occupies 55% of the pitch and therefore disturbing the primary flow. This behaviour is related to the proximity of the 95% spanwise plane to the shroud wall and as such, the tip clearance. It has been discussed before in Figure 6 that the tip leakage flow starts to develop at about 50% streamwise location where its effect to the rest of the passage can be clearly seen in Figure 7. The pressure contour at 95% spanwise location also shows the effect of having tip leakage flow to the rotor blade capability to generate torque. It can be seen that at any similar streamwise location, the pressure would gradually increase as the pitch distance from suction surface increases. However, close to the pressure surface, the pressure suddenly drops (circled in Figure 7), therefore minimizing the blade capability to generate torque. This phenomenon occurs due to the flow ability at particular spanwise location to accelerate and slip through the tip clearance region, therefore reducing the pressure close to pressure surface.

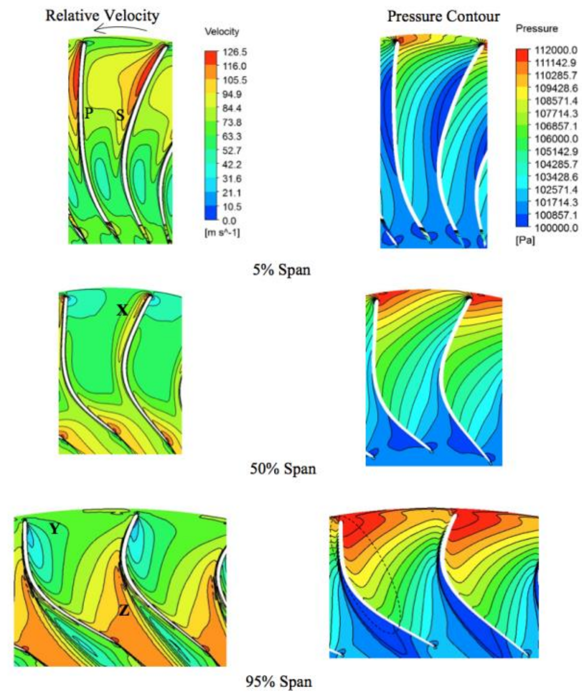


Figure 7 Velocity and pressure contour for each spanwise plane

In order to assess the power generation capability of the blade at different spanwise position, the pressure loading on a blade located at 180^0 circumferential location is plotted. This plot can be seen in Figure 8 where three lines of different spanwise locations which are at 5%, 50% and 95%. The static pressure in this plot is normalized with the turbine inlet total pressure to simplify the analysis. The plot in Figure 8 represents the pressure loading exerted by the particular blade at any streamwise location. It is desirable to have uniform pressure loading at all streamwise and spanwise locations of the blade. Figure 8 indicated that the loading exerted at mid-span has the most steady distribution from the turbine blade leading edge (streamwise location = 0.0) to its trailing edge (streamwise location = 1.0). At 95% span, there exist a large pressure difference between pressure and suction surface at 0.53 streamwise location and then the difference rapidly becomes smaller toward the blade trailing edge. This behaviour is a direct result of the flow separation induced by strong turning curvature as well as the tip leakage flow as previously explained. The lowest blade loading amongst all spanwise location is seen at 5% location where the pressure loading downstream of 0.6 streamwise location reduced quickly.

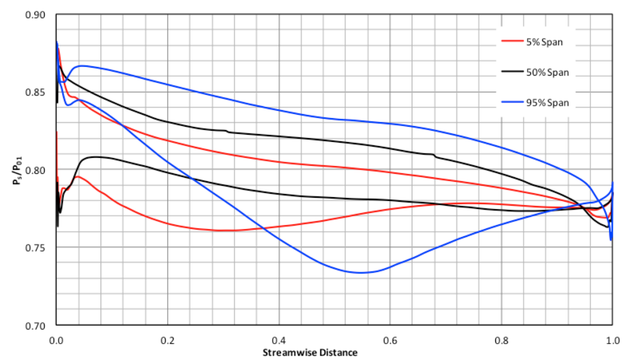


Figure 8 Pressure loading on the blade at optimum operating condition of 30000rpm

To complete the full 3-dimensional description of the flow field within the rotor passage, the velocity and pressure contour at blade-to-blade locations is also plotted. Figure 9 shows 3 locations of blade-to-blade planes on which the velocity and pressure contour is plotted, namely 5% blade-to-blade (close to pressure surface of a passage), 50% blade-to-blade (mid-passage) and 95% blade-to-blade (close to suction surface of a passage). The plane locations are shown in Figure 5. In general, the pressure contour indicated that at similar streamwise location, the pressure is always higher at shroud as compared to hub. This is due to reduced passage width as the flow gets closer to the hub, thus forcing the flow to accelerate and in turns, reduce the pressure. Furthermore, by comparing the flow structure using the velocity contour in Figure 9, it can be seen that as the blade-to-blade plane moves away from the pressure surface, there is a low velocity region that mitigates from about 20% streamwise close to the shroud to 50% streamwise and about 30 spanwise. Subsequently, this low velocity

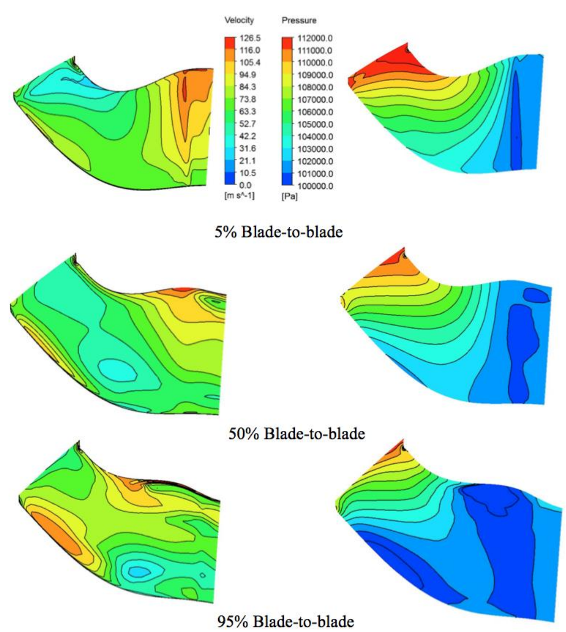


Figure 9 Velocity and pressure contour for each blade-to-blade plane

region in the passage moves to 65% streamwise close to the hub as the blade-to-blade plane approach the suction surface. Furthermore, the effect of tip leakage is clearly visible in the velocity and pressure contours at 95% blade-to-blade. It is inevitable that the effect of tip leakage is even greater closer to the shroud wall. Another obvious observation in the pressure contour of Figure 9 is that there is a flow separation exists at the rotor hub. This observation is clear at 50% and 95% blade-to-blade locations and but does not appear on 5% plane. The particular feature of the flow separation near the rotor hub of mixed flow turbine is also seen by Palfreyman and Martinez-Botas[13] Meanwhile, for the radial turbine, such feature ceases to exist[14].

4.0 CONCLUSION

The numerical model for a full stage turbocharger turbine has been successfully modelled and validated. Maximum RMS recorded for comparison of CFD and experimental data for both mass flow parameter and efficiency is 2%. Analysis of flow field has indicated the existence of tip leakage flow in the turbine passage downstream of mid-chord location. There is little evidence of tip leakage flow upstream this location due to coriolis effect. This result in low pressure region that potentially induce flow separation at the suction surface. The flow separation has adverse effect in such a way that it restricts the movement of the primary flow. It also results in lesser pressure difference between pressure and suction surface of the turbine blade thus impairing the blade ability to generate torque. Furthermore, analysis on spanwise plane has revealed that the flow close to the shroud is heavily distorted. The separation that exists at this location occupies about 55% of the pitch thus inducing blockage of the primary flow. Moreover, there is also evidence of flow separation close to the rotor hub, a feature which is unique to the mixed flow turbine.

Acknowledgement

The corresponding author would like to acknowledge Universiti Teknologi Malaysia (VOT number: Q.J130000.2724.01K70) for the financial support on this research.

References

[1] Lymberopoulos, N., Baines, N. C. and Watson, N. 1988. Flow in Single and Twin Entry Radial Turbine Volutes. *ASME Gas Turbine Aeroengine Congr*. 88-GT-59.

[2] Barr, L. and Spence, S. W. T. 2008. Improved Performance of a Radial Turbine Through the Implementation of Back Swept Blading. *Proc ASME Turbo Expo No. GT2008-50064*.

[3] Barr, L., Spence, S., Thornhill, D. and Eynon P. 2009. A Numerical and Experimental Performance Comparison of an 86 MM Radial and Back Swept Turbine. *Proc ASME Turbo Expo No. GT2009-59366*.

[4] Walkingshaw, J., Spence, S., Ehrhard, J. and Thornhill, D. 2010. A Numerical Study of the Flow Fields in a Highly Off-Design

Variable Geometry Turbine. *Proc ASME Turbo Expo No. GT2010-22669*.

[5] Padzillah, M. H., Rajoo, S. and Martinez-Botas, R. F. 2014. Influence of Speed and Frequency towards the Automotive Turbocharger Turbine Performance under Pulsating Flow Conditions. *Energy Convers. Manag.* 80: 416–428.

[6] Denton, J. D. 2010. Some Limitations of Turbomachinery CFD. In *ASME Turbo Expo 2010: Power for Land, Sea and Air*. 1–11.

[7] Dale, A. and Watson, N. 1986. Vaneless Radial Turbocharger Turbine Performance. Proc. IMechE Int. Conf. Turbocharging Turbochargers (Mechanical Eng. Publ. London). 65–76.

[8] Szymko, S. 2006. *The Development of an Eddy Current Dynamometer for Evaluation of Steady and Pulsating Turbocharger Turbine Performance*. PhD Thesisi, Imperial College of Science, Technology and Medicine, University of London.

[9] Rajoo S. 2007. *Steady and Pulsating Performance of a Variable Geometry Mixed Flow Turbochager Turbine*. PhD Thesis, Imperial College of Science, Technology and Medicine, University of London.

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

[11] Padzillah, M. H., Rajoo, S. and Martinez-Botas, R. F. 2012. Numerical Assessment of Unsteady Flow Effects on a Nozzled Turbocharger Turbine. *Proc ASME Turbo Expo No. GT2012-69062*.

[12] Japikse, D., Baines and N. C. 1994. *Introduction to Turbomachinery*. Oxford University Press.

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

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