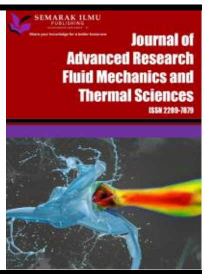
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 99, Issue 1 (2022) 104-113

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879



Investigation on the Flow Angle of a Mixed Flow Turbocharger Turbine Under Steady State Operating Conditions

Wan Muhammad Aqib Wan Rosli[1], Muhamad Hasbullah Padzillah[1,*]

[1] UTM Centre for Low Carbon Transport in Cooperation with Imperial College London, Universiti Teknologi Malaysia, 81300 Johor, Malaysia

ARTICLE INFO

Article history:
Received 4 April 2022
Received in revised form 23 July 2022
Accepted 2 August 2022
Available online 30 August 2022

Keywords:
Turbocharger; mixed flow turbine; computational fluid dynamics; steady state flow; incidence angle

ABSTRACT

Turbocharger is widely used in automotive field for the capability to extract or boost the downsizing internal combustion (IC) engine as it has been considered as a way to provide highly efficient of small engine. Turbocharger also reduces the fuel consumption as well as the carbon dioxide (CO_2) emission. Turbine is one of the main components of a turbocharger. Generally, this study is mainly focusing on a mixed flow turbine that work under steady flow operation. Numerical work using Computational Fluid Dynamics (CFD) is presented in this report. This report aims to compare the flow angle analysis of three selected cases under steady state flow operating conditions. A 3D modelling on a mixed flow turbine is generated accordingly to 'Rotor A' design by using a commercial software of Ansys CFX. Three selected cases were selected in order to compare the flow angle at the mid span circumference location at vane and rotor inlet. The simulation results show the capabilities of the 3D model developed in capturing the trend of available experiment data at turbine speed of 48kRPM. The mixed flow turbine achieves optimum total-to-static efficiency of 74.16 % at velocity ratio of 0.65 with mass flow rate of 0.5kg/s. Besides that, the results shown that the flow angle distribution at the rotor inlet is varied throughout the rotor circumference location between the three cases. Case 2 achieved the optimum incidence angle compared to the other cases thus showing the impact of incidence angle in determining the turbine performance.

1. Introduction

Over decades, the globe determined in restricting the gas emissions that could endanger the ecosystem by causing the greenhouse effect as a result of temperature and weather shifting pattern. The world is currently moving towards the efforts in achieving zero carbon dioxide emission by 2050 [1]. The world still could not meet the international climate goals as demanded by the Paris Agreement (PICC) [2]. In order to meet the demand in reducing the greenhouse gas (GHG) effect, the automotive industry has engrossed to reduce the CO_2 emission by downsizing an engine volume resulting low power output of the engine. A device known as a turbocharger have the capability in boosting the power output of a downsized engine that similar to the higher power output of natural aspirated engine. Generally, an automotive turbocharger is driven by the exhaust gases of an engine

* *Corresponding author.*
E-mail address: mhasbullah@utm.my

https://doi.org/10.37934/arfmts.99.1.104113

where a turbine powered a centrifugal compressor [3]. Turbocharging technique is widely used on both compression and spark ignition IC engines as the technique is proven to improve performance and reduce total displacement [4].

A mixed flow turbine capable in achieving better mass flow rate and efficiency at highly expansion ratio compare to the radial flow turbine that have performance limitation due to the blade configuration [5,6]. The efficiency of a mixed flow turbine peaks at lower velocity ratio of 0.707 than a radial flow turbine [7]. The capability of a mixed flow turbine in achieving non-zero inlet angle blade by maintaining allowable stress limit [8]. The optimum incidence angle range for a radial turbine fall within the range of -20° to - 40° [9].

Palfreyman and Martinez-Botas [10] carried out study on the flow field of a mixed flow turbine with a detailed 3D CFD analysis. The study was performed in a different approach compare to Lam *et al.,* [11]. The boundary conditions were applied on the whole domain accordingly experimental work achieved by Karamanis *et al.,* [12]. Padzillah [13] later conducted a study on a mixed flow turbocharger turbine under steady flow conditions in both experimentally and numerically. Padzillah [13] stated the importance role of the incidence flow angle in determining the turbine performance. The incidence flow angle is a relative discrepancy between the relative flow and blade angles [14]. Lee *et al.,* [15] stated the presence of axial flow element at the inlet rotor was discovered to obtain swallowing capacity than the radial flow element as it is mainly due to the variation of distribution of an incidence at the respective location. Lee *et al.,* [16] further stated that the efficiency at low velocity ratio and loss correlated to the flow angle could be improve with the use of a mixed flow turbine. Leonard *et al.,* [17] found a significant presence of secondary flow and loss at highly incidence angle as the flow enter the rotor. Leonard *et al.,* [17] further stated this presence led to disruption of the flow behavior within the turbine passage thus reduced the turbine efficiency. Gurunathan *et al.,* [18] performed study on the incidence angle under steady state between two different volute design as the asymmetric design have a better incidence flow angle variation thus improving the turbine efficiency. The purposes of this paper are to develop a full stage mixed flow turbocharger turbine and to investigate the flow angle of a mixed flow turbocharger turbine under steady state operating conditions.

2. Methodology

Figure 1 illustrated the three main domains of the model that consist of volute, vane and rotor that generated in two different software packages available in the Ansys software such as ICEM CFD and TurboGrid. developed with the available tools in the ANSYS software. A single-entry volute developed by Padzillah [13] was used in the current research where modification was made by adding the of vane section and tongue region on the Holset H3B volute. The purposes of the modification are to allow variation of types of vane setup to be tested on a same volute and to replicate the flow circulation of an actual volute near the tongue region. The volute was divided into four parts to ease the meshing process with the ICEM CFD. The vane is developed according to the profile lines of the hub, shroud and blades by using the Ansys Turbogrid. The lean vane geometry of 50^{o} from the hub surface is made according to the NACA 0015 profile. Furthermore, the vanes are designed to have 15 blades with different chord length for the hub and shroud of 22.3 mm and 26.3 mm respectively.

The rotor of the turbine is developed well according to the geometry of 'Rotor A' designed by Abidat [19]. Similar with the vane development, the profile lines consist of the hub, shroud and blades of 'Rotor A' are imported and generated in the TurboGrid. However, the rotor development requires more number of profile lines compared to the vane due to the complexity of its geometry as there are about nine profile lines imported from a profile.crv to the Turbogrid that undergo the Bezier

Polynomials. The Bezier Polynomials constant parameters are extracted from Abidat [19] and Palfreyman and Martinez-Botas [10] as a guidance to developed the mixed flow turbine rotor. The number of nodes of the full stage model was summarized in Table 1.

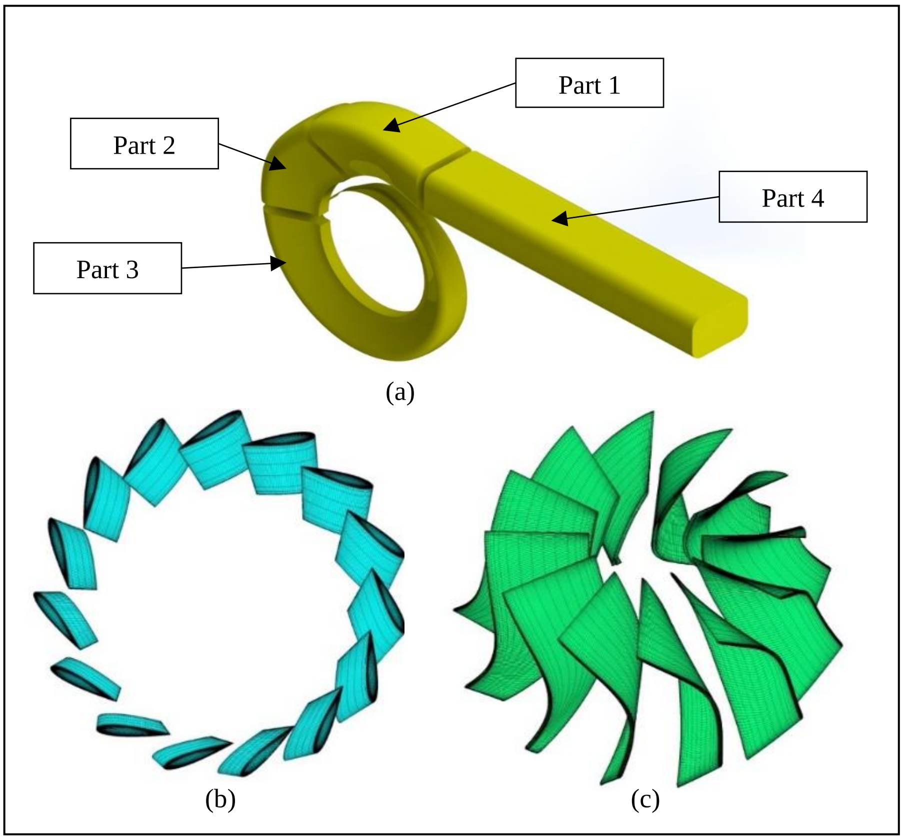


Fig. 1. The three main domain of the full staged mixed flow turbocharger turbine (a) volute, (b) vane and (c) rotor

Table 1
Summary on the meshing domains

Domain	Number of Nodes	Element Type
Inlet Duct	332992	Hexahedral Ogrid
Volute	875838	Hexahedral
Vane	809745	Hexahedral
Rotor	3242760	Hexahedral
Total	5261335	

The three main domains were assembled in the Ansys CFX in order to prepare the model for simulation run under steady state flow operating condition. The preparation was made by defining the boundary conditions on the full stage mixed flow turbocharger turbine model for numerical computation calculation as illustrated the Figure 2. Firstly, the turbulence model for the simulation was made accordingly with the aim to predict precisely on the flow behavior for the turbocharger turbine performance where a standard $k-\varepsilon$ is chosen. The mass flow rate and stagnation temperature are set to enter normal to the inlet plane while the outlet boundary is defined with temperature and relative pressure. As for the wall parts such as volute wall, shroud, hub and blade are set to have a free slip wall boundary. The interface between the vane and rotor is set to be Frozen Rotor. The mass flow rate entered the inlet plane from 0.2 kg/s up to 0.8 kg/s at both rotor speed of

30kRPM and 48kRPM as the simulation run for a thousand iteration with 1×10^{-5} of convergence criteria.

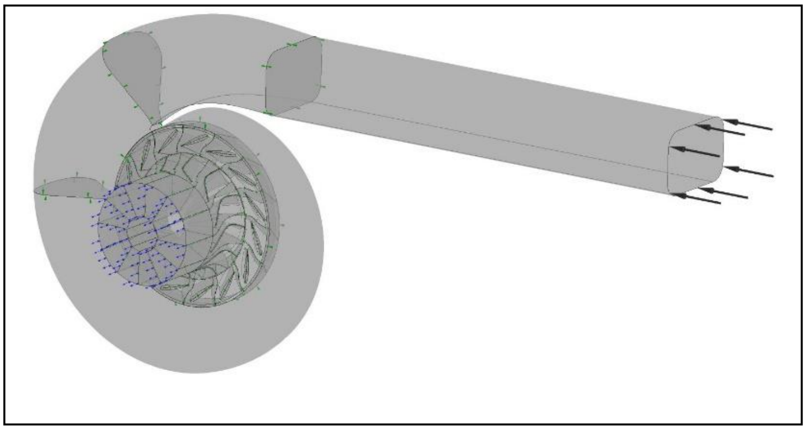


Fig. 2. The boundary definition on the three domains of the full assembly on the full stage turbocharger turbine

The full stage model will be validated with the available experimental data in order to determine the accuracy of the model generated computationally for turbine speed of 48kRPM under steady flow operating conditions. The validation will be performed by calculating the turbine performance parameters such as mass flow parameter, pressure ratio, velocity ratio and isentropic efficiency on the computed model. The equations for the turbine performance parameters are defined as

$$\mathrm{MFP} = \frac{\dot{m}\sqrt{T_{01}}}{P_{01}} \tag{1}$$

$$\mathrm{PR} = \frac{P_{01}}{P_5} \tag{2}$$

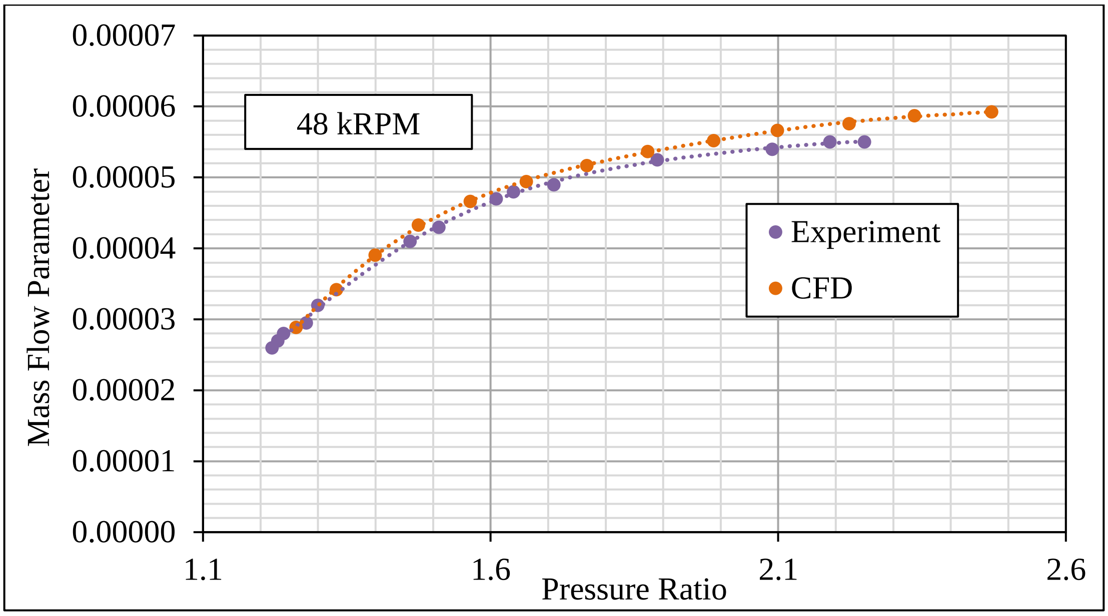
$$\mathrm{VR} = \frac{U}{C_{is}} \tag{3}$$

$$\eta_{t-s} = \frac{W_{act}}{W_{isen}} \tag{4}$$

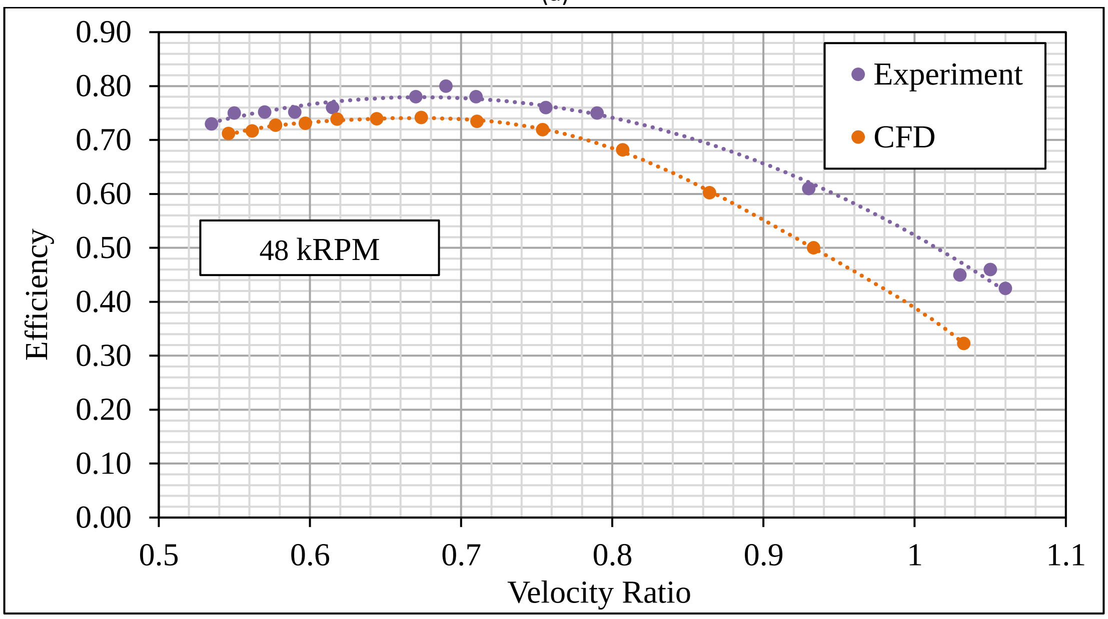
3. Results and Discussion

3.1 Validation Work

Figure 3 plotted the turbine performance parameter of Figure 3(a) Mass flow parameter and Figure 3(b) Total-to-static efficiency that were calculated from the simulation results and directly compared them with available experimental data at rotor speed of 48kRPM. Figure 3(a) plots the mass flow parameter against pressure ratio for both simulation and experimental results. The pattern is quite similar between simulation and experimental results plotted in Figure 3(a) as the mass flow parameter calculated from the simulation results nearly the same with the experimental results at any pressure ratio. The model is computed to have 2.3 value of RMSE. The model is recorded to achieve peak efficiency at velocity ratio of 0.66 with 74.16% of total-to-static efficiency at lower mass flow rate of 0.5 kg/s as plotted in Figure 3(b). The model of the full stage mixed flow turbocharger turbine that performed under steady flow operating conditions proved to be able capture the trend of existing experimental results at rotor speed of 48 kRPM. Furthermore, the model achieved peak efficiency under velocity ratio of 0.7 for both rotor speed but at different mass flow rate as discussed in the introduction section.



(a)



(b)

Fig. 3. Validation exercise on (a) Mass flow parameter and (b) Total-to-static efficiency between experimental data and simulation results at rotor speed of 48kRPM

3.2 Flow Angle Analysis

The flow angle distribution is measured at the mid span of the vane and rotor inlet circumference location as illustrated in the Figure 4. The cases were selected at mass flow rate of 0.2 kg/s, 0.5 kg/s and 0.8 kg/s under steady state operating conditions. These cases were selected based on the turbine performance plotted in the Figure 2(b) as the turbine performance is summarized in the Table 2.

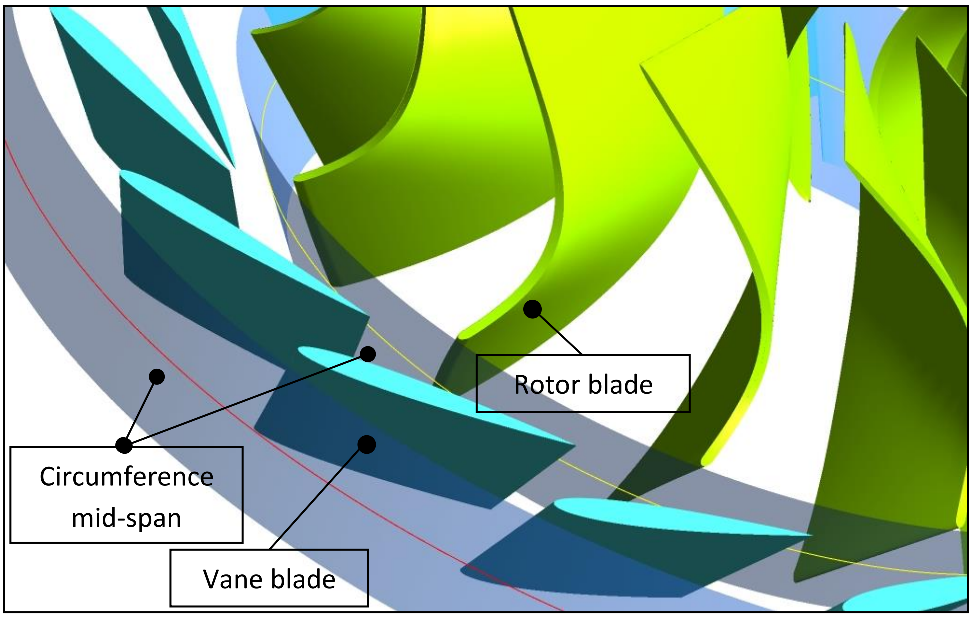


Fig. 4. The mid-span circumference location at vane and rotor inlets

Table 2
The selected cases under steady state flow operating conditions

Case	Mass Flow Rate (kg/s)	Efficiency, $_{t-s}$ (%)
1	0.2	32.25
2	0.5	74.16
3	0.8	71.20

Figure 5 plots the velocity absolute flow angle at inlet vane circumference for the three selected cases. Ideally, the volute is designed to flow out the volute at 69 but each of the cases shows a fluctuation of absolute flow angle fluctuates at the mid-span circumference. Based on observation, the distribution of flow angle varies along the vane inlet circumference location and it fluctuate at almost similar trend of the three cases. However, the flow angle is seen to be minimum near the tongue region (labelled as X) due to the flow circulation occurred near the tongue region before the three cases fluctuates closer to each other. Besides that, the flow angle excites towards the end of the volute as the flow velocity increases due to the spiral shape of the volute. Aside from that, the closeness of vane arrangement towards the volute exit could be the cause of flow angle fluctuations at the vane inlet circumference. The average flow angle throughout the inlet vane circumference is recorded with a value of 74° approximately for all condition which indicates the flow is almost tangential at the inlet vane circumference location.

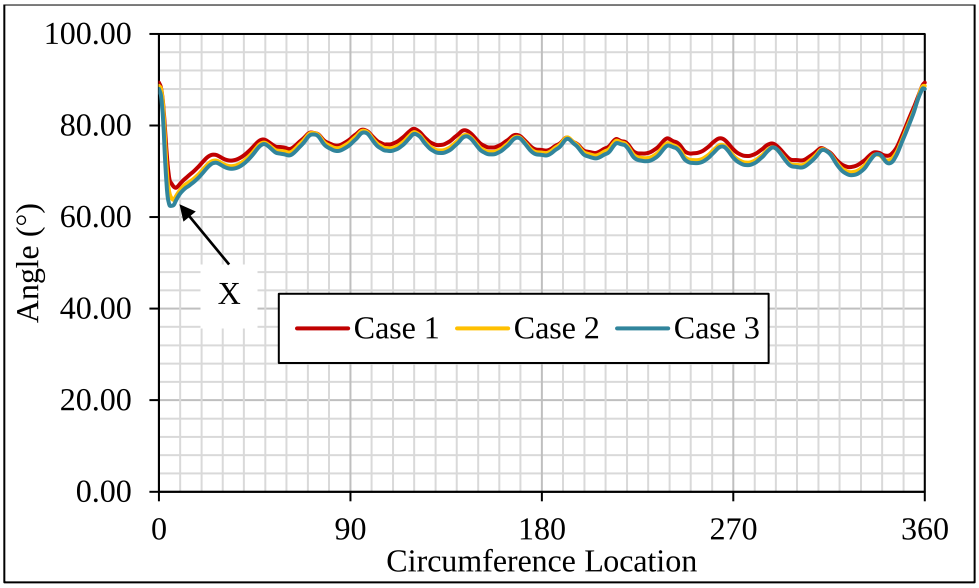


Fig. 5. Flow angle of three different conditions at the vane inlet

Fundamentally, the purpose of vane placement is to guide the flow to enter uniformly throughout the rotor inlet circumference at designated value of flow angle. Nonetheless the flow angle as plotted in the Figure 6 is shown to be fluctuating rather than uniformly distributed along the rotor inlet circumference for the three cases. At case 1, the flow angle fluctuates vigorously along the inlet rotor circumference location. This is might due to the flow separation happened in the vane domain as it impedes the flow as it approaching the rotor inlet. The case 1 could not show a relatively constant fluctuation from the beginning towards the end of the inlet rotor circumference with average value of 63° indicating the flow is bad as it deviated away than the designated angle. In contrast to the case 2 and 3, the flow angle distribution shows a relatively constant fluctuation as the flow angle fluctuates almost close to each other. The average value of the flow angle for case 2 and 3 is recorded at 70° thus indicating that the vanes are capable in delivering the flow at designated angle as the flow approaching the inlet rotor.

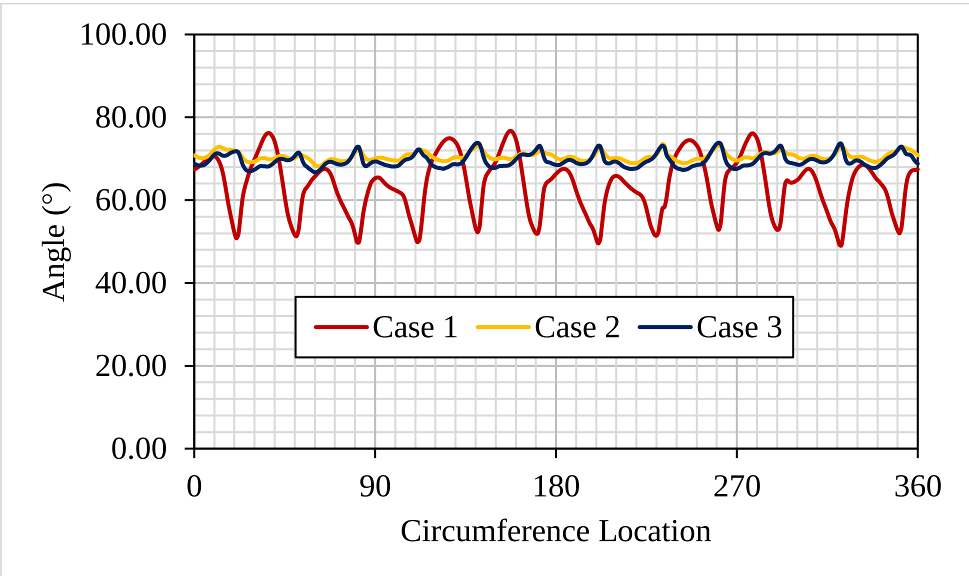


Fig. 6. Absolute flow angle of three different operating conditions at the mid-span of rotor inlet circumference location

Figure 7 shows the relative flow angle distribution along the mid span of inlet rotor circumference. A clear observation could be made as the relative flow angle between the three selected cases were fluctuates further than another. The plotted relative angle could give a glimpse on the incidence flow angle distribution as the incidence angle is achieve by the difference of relative angle and the blade angle which the blade designed in this current study is backswept at 20°. Figure 8 plotted the incidence flow angle at the rotor inlet circumferential location for the three cases. Clearly, there is a significant different of incidence angle distribution along the rotor inlet circumference location. The incidence flow angle distribution at case 2 was observed to be fell within the optimum incidence range (yellow shaded region) compared to the other two cases. Since the incidence flow angle at case 2 distributed within the optimum range, there would be less flow separation and minimized the secondary flow along the rotor passages. The highly negative incidence angle at case 1 would resulting a flow separation at the blade pressure surface while the positive incidence at case 3 would cause flow separation at the blade suction surface.

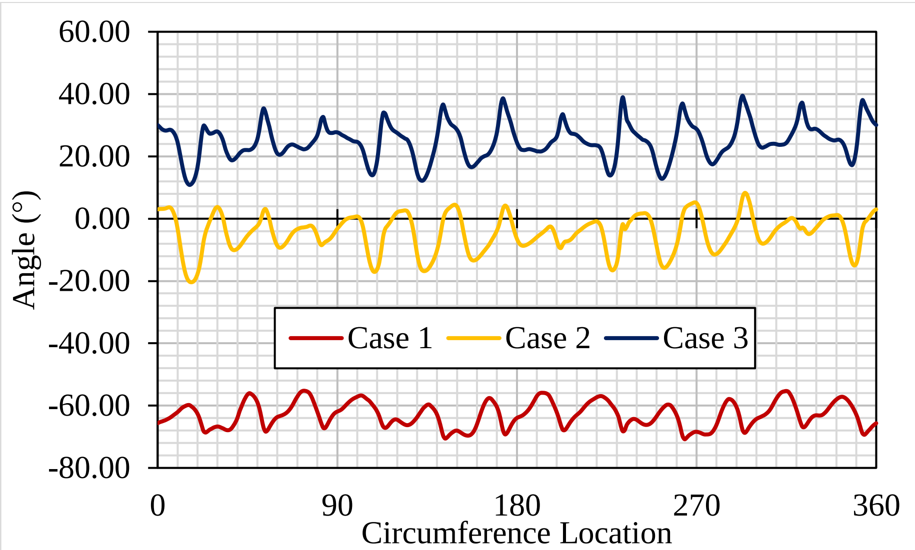


Fig. 7. Relative flow angle of three different operating conditions at the rotor inlet circumference location

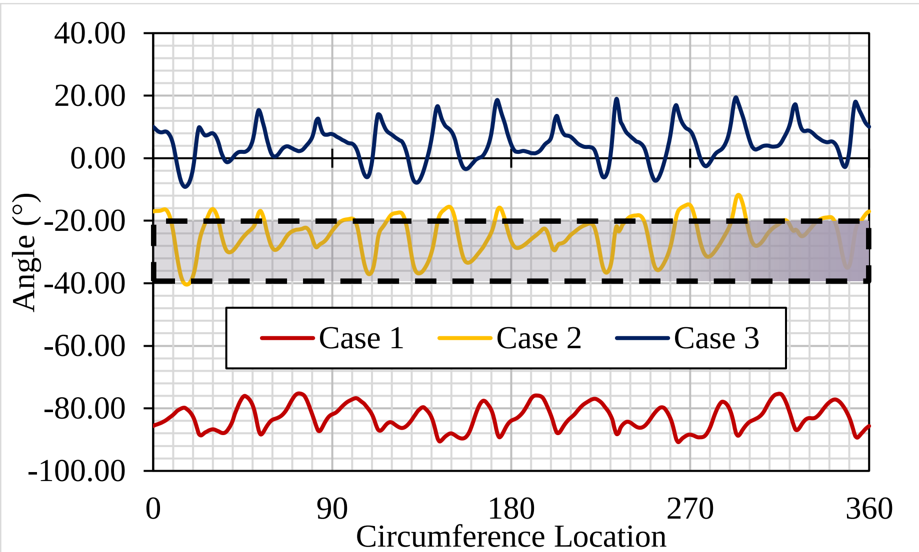


Fig. 8. Incidence flow angle of three different operating conditions at the rotor inlet circumference location

Last but not least, the analysis on the flow angle along the mid span of the inlet vane and rotor circumference locations shows a clear different of the flow angle distribution. The flow angle at the inlet vane is almost relatively the same pattern as the three cases fluctuates closer to each other compare to at the rotor inlet. Furthermore, the case 2 is recorded to have an average value of -24.58° of incidence flow angle while the case 1 and case 3 have an average incidence flow angle of -82.95° and 5.00° respectively. This average incidence angle value provides a glimpse on the flow behavior that would occur within the turbine passages. The presence of flow separation and secondary flow could hinder the flow propagation within the turbine resulting a bad turbine performance. The case 1 have a bad flow propagation as it approaching the rotor inlet compared to the other cases thus resulting low total-to-static efficiency of 32.25 % compared to case 2 and case 3 with 74.16 % and 71.20 % respectively.

4. Conclusion

In conclusion, a numerical study on a 3D full stage of a mixed flow turbocharger turbine under steady state flow operating conditions was conducted. The numerical model was successfully validated at specified mass flow rate with existing experimental data at constant turbine speed of 48 kRPM. The entire simulation run achieved total to static efficiency of 74.16% as the mass flow rate of 0.5 kg/s enter at the measurement plane. Besides, the numerical results proved that the efficiency peaks at lower velocity ratio than 0.707 as discussed in the introduction section.

The analysis performed shows a significant difference on the flow angle distribution at the mid span of the inlet rotor circumference location compared to the flow angle at the inlet vane circumference location. The analysis results show only the case 2 was capable in achieving the optimum incidence range as the incidence flow angle fell within the yellow region compared to the other two cases. The capability of the flow in achieving the optimum incidence angle at case 2 help in reducing and minimizing the presence of flow separation and secondary flow within the turbine passage thus led to a better total to static efficiency of 74.16 %. As for the case 1, the total to static efficiency is the lowest among the three selected cases with 32.25 % due to the highly negative incidence at the rotor inlet. Thus, the numerical results show the important role of incidence flow angle in determining the turbine performance.

Acknowledgement
This research is funded by Ministry of Higher Education, Malaysia under Fundamental Research Grant Scheme no FRGS/1/2019/TK07/UTM/02/17.

References
[1] Yaacob, Nur Fatma Fadilah, Muhamad Razuhanafi Mat Yazid, Khairul Nizam Abdul Maulud, and Noor Ezlin Ahmad Basri. "A review of the measurement method, analysis and implementation policy of carbon dioxide emission from transportation." *Sustainability* 12, no. 14 (2020): 5873. https://doi.org/10.3390/su12145873

[2] Loutun, Mark Jason Thomas, Djamal Hissein Didane, Mohd Faizal Mohideen Batcha, Kamil Abdullah, Mas Fawzi Mohd Ali, Akmal Nizam Mohammed, and Lukmon Owolabi Afolabi. "2D CFD Simulation Study on the Performance of Various NACA Airfoils." *CFD Letters* 13, no. 4 (2021): 38-50. https://doi.org/10.37934/cfdl.13.4.3850

[3] Chanshetty, Vijaykumar B., and Harshavardhan. "Study of Turbo Charging in Six Stroke Diesel Engine." *International Journal on Emerging Technologies* 7, no. 2 (2016): 366-370.

[4] De Bellis, Vincenzo, Silvia Marelli, Fabio Bozza, and Massimo Capobianco. "1D simulation and experimental analysis of a turbocharger turbine for automotive engines under steady and unsteady flow conditions." *Energy Procedia* 45 (2014): 909-918. https://doi.org/10.1016/j.egypro.2014.01.096

[5] Baines, N. C. "Radial and mixed flow turbine options for high boost turbochargers." In *7th International Conference on Turbochargers and Turbocharging*. 2002.

[6] Pesiridis, Apostolos, and Ricardo F. Martinez-Botas. "Experimental evaluation of active flow control mixed-flow turbine for automotive turbocharger application." *Journal of Turbomachinery* 126, no. 1 (2007): 44-52. https://doi.org/10.1115/1.2372778

[7] Rajoo, Srithar, and Ricardo Martinez-Botas. "Mixed flow turbine research: A review." *Journal of Turbomachinery* 130, no. 4 (2008): 044001. https://doi.org/10.1115/1.2812326

[8] Lüddecke, Bernhardt, Dietmar Filsinger, and Jan Ehrhard. "On mixed flow turbines for automotive turbocharger applications." *International Journal of Rotating Machinery* 2012 (2012). https://doi.org/10.1155/2012/589720

[9] Japikse, David, and Nicholas C. Baines. *Introduction to turbomachinery*. Concepts ETI, Inc./Oxford University Press, 1994.

[10] Palfreyman, D., and R. F. Martinez-Botas. "The pulsating flow field in a mixed flow turbocharger turbine: an experimental and computational study." *Journal of Turbomachinery* 127, no. 1 (2005): 144-155. https://doi.org/10.1115/1.1812322

[11] Lam, J. K. W., Q. D. H. Roberts, and G. T. McDonnell. "Flow modelling of a turbocharger turbine under pulsating flow." In *ImechE. Conference Transactions from 7th International Conference on Turbochargers and Turbocharging*, pp. 14-15. 2002.

[12] Karamanis, N., R. F. Martinez-Botas, and C. C. Su. "Mixed flow turbines: Inlet and exit flow under steady and pulsating conditions." *Journal of Turbomachinery* 123, no. 2 (2001): 359-371. https://doi.org/10.1115/1.1354141

[13] Padzillah, Muhamad Hasbullah. "Experimental and numerical investigation of an automotive mixed flow turbocharger turbine under pulsating flow conditions." *PhD diss., Imperial College London*, 2014.

[14] Morrison, Richard, Stephen Spence, Sung Kim, Dietmar Filsinger, and Thomas Leonard. "Investigation of the effects of flow conditions at rotor inlet on mixed flow turbine performance for automotive applications." In *International Turbocharging Seminar*, p. 1-12. 2016.

[15] Lee, Samuel P., Simon M. Barrans, Martyn L. Jupp, and Ambrose K. Nickson. "Investigation into the impact of span-wise flow distribution on the performance of a mixed flow turbine." In *Turbo Expo: Power for Land, Sea, and Air*, vol. 51005, p. V02BT44A027. American Society of Mechanical Engineers, 2018. https://doi.org/10.1115/GT2018-76992

[16] Lee, Samuel P., Martyn L. Jupp, Simon M. Barrans, and Ambrose K. Nickson. "Analysis of leading edge flow characteristics in a mixed flow turbine under pulsating flows." *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* 233, no. 1 (2019): 78-95. https://doi.org/10.1177/0957650918778661

[17] Leonard, Thomas, Stephen Spence, Andre Starke, and Dietmar Filsinger. "Numerical and experimental investigation of the impact of mixed flow turbine inlet cone angle and inlet blade angle." *Journal of Turbomachinery* 141, no. 8 (2019): 081001. https://doi.org/10.1115/1.4042652

[18] Gurunathan, Balamurugan A., Uswah Khairuddin, Nazrun Nabill Azlan Shah, and Ricardo Martinez-Botas. "Influence of Double Entry Volute on Incidence Angle Variation Under Steady Flow: Numerical Investigation." *CFD Letters* 12, no. 10 (2020): 75-89. https://doi.org/10.37934/cfdl.12.10.7589

[19] Abidat, Miloud. "Design and testing of a highly loaded mixed flow turbine." *PhD diss., University of London*, 1991.