Jurnal Mekanikal
December 2012, No.35 , 63-81

HEAT DISTRIBUTION STUDY ON TURBOCHARGER TURBINE'S VOLUTE

Ibthisham A. Mohd*, S. Rajoo and A. Nordin Darus

Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia,
81200 Skudai, Johor Bahru

ABSTRACT

The aimed of this project is to evaluate turbine's performance based on its actual condition. Holset H3B nozzles turbine geometry was used as simulation model. Turbine's actual working condition was simulated by using finite volume discretiztion method. Initial analysis was done by one-dimensional and two-dimensional analysis. Further investigation was done in three-dimensional with heat loss via turbine volute by the mode of convection. Parameters studied are corrected mass flow, turbine's efficiency at different heat cases, temperature distribution along turbine's volute and difference in temperature between inner and outer wall temperature. Temperature difference within turbine's volute is the major factor that deteriorates turbine's efficiency. Since turbine wall is thin, small temperature difference will result to high heat loss.

Keywords : *Holset turbochargers, heat transfer coefficients, turbine, efficiency*

1.0 INTRODUCTION

By the aid of turbocharger, engine can produce more power at the same speed of Naturally Aspirated (NA) engine. Technically speaking, turbocharger forcing more air into combustion chamber thus, this will increase and improve volumetric efficiency [1].

The needs of turbocharger can be seen significantly on heavy vehicle that travel up hills. Travelling through hills area (high altitude) requires engine than can overcome vehicle rolling resistance and more importantly, gradient resistance. Gradient resistance tend to create additional load based on hill gradient. This kind of load will burden vehicle's engine. Furthermore pressure, air density and temperature are decreased inversely proportional with hill's altitude. With lack of air density present during cruising along hill, it's quite difficult for an engine to breathe. However, at this point, one could see the importance of turbocharger when it could give additional power by meant of forcing air into combustion cylinder thus, increasing air density. With the aid of turbocharger, eventhough a vehicle is used under lack of air density condition, the engine still can be running effectively as compared to naturally aspirated engine.

Heat is an energy that can be used which can be transformed in many ways. In turbocharger, heat loss can deteriorate its performance. Energy in turbine is calculated based on the temperature of inlet and outlet of the turbine but within that, there is heat loss occurring at turbine volute that creates massive reduction of turbine efficiency.

*Corresponding author : ibthisham@fkm.utm.my

Heat loss in turbocharger can be separated into two categories, namely internal and external heat transfer [3]. Internal heat transfer can be describe as heat loss through internal part of components such through bearing, shaft and through compressor side while external heat transfer is heat loss through turbine volute. Heat loss through turbine volute is more significant due to large area that is exposed to ambient. Heat loss by convection and radiation are dominant around turbine volute and heat loss by conduction is minimal. The heat loss via three medium of heat transfer (convection, radiation and conduction) is illustrated in Figure 1.

Turbocharger is designed in such a way that it can cope with heat transfer but, the capability is only approximately 30% (internal heat transfer) of the total heat loss [3]. Approximately one third of the total heat loss is taken care by lubrication oil, which is located between turbine and compressor. Eventhough internal heat transfer is absorbed by lubrication oil, but there is still heat loss that reach the compressor. Heat from turbine that flows by the mean of conduction will deteriorate compressor efficiency [4] and [5]. The remaining 70% of heat loss is lost through surroundings/ambient [6]. Heat loss through surroundings by heat convection through turbine volute is dominant and there is no device or system that compensates this heat loss.

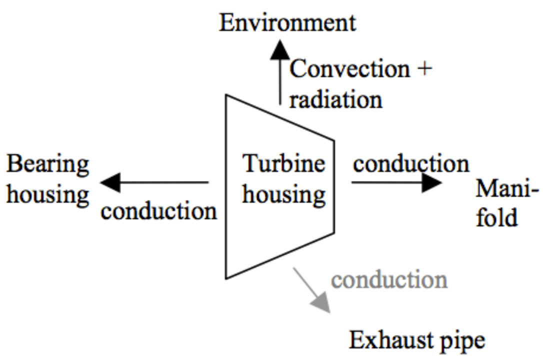


Figure 1 : Heat losses around turbine volute [3]

Reduction of pressure and temperature along turbine's volute are converted as an output of turbine's power. However, analysis made by turbocharger manufacturers only take into account pressure and temperature drop along turbine volute without take into consideration external heat loss via turbine volute for turbine's energy properties (power and efficiency). It is quite difficult to justify actual heat loss since there are many parameters effecting turbochargers performance on actual condition. Contradiction of turbine power possibly exists between data from turbine maps given by turbocharger manufacturer and data from actual turbine operation on engine.

2.0 TURBOCHARGER TURBNINE MAP

Predicted output of turbine is given based on turbine map. Turbine maps contains output of turbine based on throttle opening, mass flow rate and the value of parameters are depends on the turbocharger manufacturer geometry. The performance characteristics of turbine or turbine map typically are displayed in term of efficiency and mass flow rate with varying pressure ratio on a steady flow rig [8]. The graph that illustrated in the turbine map is made based on adiabatic conditions as portrayed in Figure 2. Thus the results will slightly deviate from real time situation because there is no heat transfer effect is taken into account (adiabatic condition).

Turbine map gives the information about the pressure drop in turbine and turbine's efficiency. Pressure drop along turbine volute is namely as pressure ratio (PR) in the turbine map and this parameter is strongly related to mass flow of air (exhaust gas) that flow into turbine. At higher PR, more air flows into turbine and this shows turbine possesses more power at higher PR due to high influx of exhaust gas. Mass flow inside turbine volute depends on cross section area of volute. The smaller cross section, the higher the mass flow will be thus, resulting to a higher boots pressure or higher PR.

Parameter such as corrected mass flow is used in typical turbine characteristics to demonstrate the strong relationship with PR furthermore it is a dimensionless parameter to calculate swallowing capacity of turbine [9]. Characteristics or parameters should be independent from any variable, thus corrected mass flow is introduced in turbine characteristics in order to avoid dependency of turbine map on temperature and pressure upstream of turbine [9]. The equation of corrected mass flow is depicted in Equation 2 below:

$$PR = \frac{P_{in}}{P_{out}} \tag{1}$$

$$\dot{m}_{corr} = \frac{\dot{m}\sqrt{\frac{T_{in}}{T_{ref}}}}{\frac{P_{in}}{P_{ref}}} \tag{2}$$

Equation 2 shows corrected mass flow for typical turbocharger characteristics which refers to actual turbine's mass flow rate, while T_{ref} and P_{ref} are ambient temperature and pressure which have value of 298K and 101.3kPa respectively. Usually, turbine maps come along efficiency maps that plotted turbine's efficiency based on certain engine revolution per-minute (RPM) and certain PR. Typical turbine's efficiency is calculated based on ratio of isentropic works that assuming reversible process without friction and heat interaction within the control volume with actual works calculated based on the difference of temperature between turbine inlet and outlet temperature. The equations involve during calculation of turbine's efficiency are given as follows:

$$W_{act} = \dot{m} c_p (T_{in} - T_{out}) \tag{3}$$

$$\frac{T_{out}}{T_{in}} = \left(\frac{P_{out}}{P_{in}}\right)^{\frac{k-1}{k}}$$

$$T_{out} = T_{in}\left(\frac{P_{out}}{P_{in}}\right)^{\frac{k-1}{k}} \tag{4}$$

$$W_{act} = \dot{m} c_p (T_{in} - T_{out}) \tag{5}$$

Substituting Equation (3) into Equation (4) gives:

$$W_{isen} = \dot{m} c_p T_{in} (1 - \left[\frac{P_{out}}{P_{in}} \right]^{\frac{k-1}{k}}) \tag{6}$$

By definition of efficiency:

$$\begin{aligned} \eta_{turbine} &= \frac{W_{act}}{W_{isen}} \\ \eta_{turbine} &= \frac{(T_{in} - T_{out})}{T_{in} (1 - \left[\frac{P_{out}}{P_{in}} \right]^{\frac{k-1}{k}})} \\ \eta_{turbine} &= \frac{(T_{in} - T_{out})}{T_{in} (1 - \left[\frac{1}{PR} \right])^{\frac{k-1}{k}}} \end{aligned} \tag{7}$$

Thus, the final equation for turbine efficiency is given by Equation 7. From the derivation above, there is no heat interaction taken into account. The calculation is based on adiabatic manner thus, actual turbine's performance deviates from actual condition. Turbine's efficiency is plotted over certain range of PR, as illustrated in Figure 3(typical turbine efficiency map).

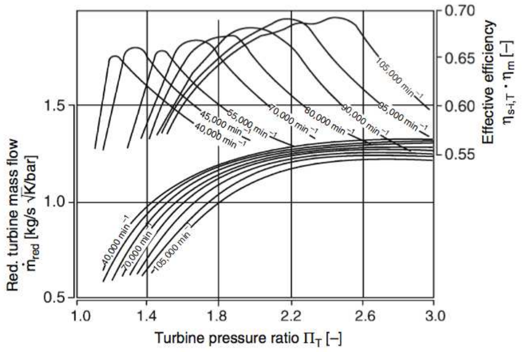


Figure 3 : Turbine map with mass flow parameter and trubine efficeincy [10]

3.0 TURBINE NON-ADIABATIC ANALYSIS

As discussed earlier, predicted power or work given by turbine map is based on turbocharger manufacturer and adiabatic analysis, which deviates from real turbocharger operation. Heat transfer effect that excludes in the analysis had made the predicted result far beyond actual data set.

Experimental and simulation work carried by F. Westin (2004) to determine heat loss within turbocharger turbine shows that simulated turbine exit temperature deviates far beyond experimental analysis as illustrated in Figure 4. Based on the figure, the deviation is around 50K. This analysis shows that there is a need to investigate the heat loss phenomenon and incorporate its effect to the current turbine map to obtain the true value of power output or efficiency of turbocharger turbine.

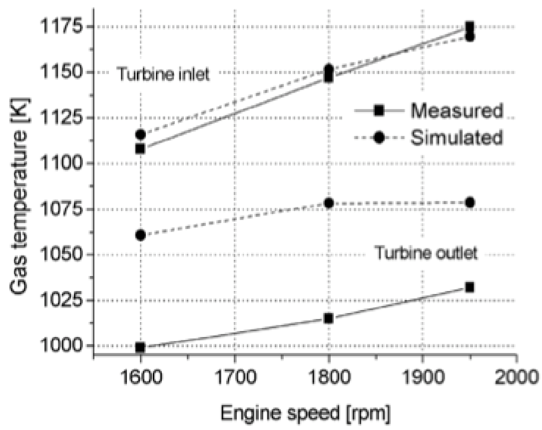


Figure 4 : Measured and simulated turbine inlet/oulet temperature [2]

4.0 SIMULATION ON TURBINE SIDE

Simulation was carried out to evaluate and mimic the real condition with the aid of several assumptions to ease the simulation. Assumptions made are based on capability of computing and degree of accuracy of the answer.Computational Fluids Dynamics (CFD) solve partial differential equation (PDE) and ordinary differential equation (ODE) that represent the nature of flow physics. Physics of flow are illustrated in the form of equation, which are in terms of conservation of mass which is Equation 8, conservation of momentum which is Equation 9 and conservation of energy which is Equation10.

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho v_x)+\frac{\partial}{\partial y}(\rho v_y)+\frac{\partial}{\partial z}(\rho v_z)=0 \tag{8}$$

$$\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}=-\frac{1}{\rho}\frac{\partial P}{\partial x}+\upsilon\left(\frac{\partial^2 v_x}{\partial x^2}+\frac{\partial^2 v_x}{\partial^2 y}+\frac{\partial^2 v_x}{\partial^2 z}\right) \tag{9}$$

$$\rho C_p\left(\frac{\partial T}{\partial t}+v_x\frac{\partial T}{\partial x}+v_y\frac{\partial T}{\partial y}+v_z\frac{\partial T}{\partial z}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+\phi \tag{10}$$

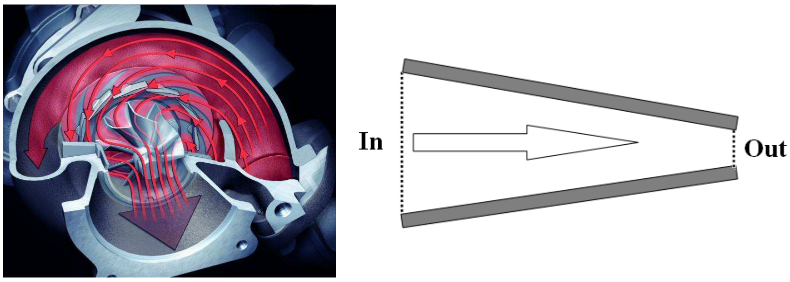
All the three equations stated above are the basic equation that involved in CFD analysis. Additional terms are included such as turbulent dissipation, if turbulent mode is taken into account in the simulation. Turbulence takes place in flow with high Reynolds number which above 4000 [11]. Solving turbulence problems is more than difficult. However, with appropriate assumptions made in turbulent model, turbulence phenomenon can be simulated. Turbulence appears to be dominant over all flow phenomena and with successful modeling technique of turbulence, numerical quality of the simulation will significantly seen [12].

Simulation of turbocharger turbine is done at turbulent mode, due to the high flow of exhaust gases into turbine's volute [13] and [14]. Appropriate turbulent model selection is deemed necessary since the working geometry is complicated and k-ε model is chosen for simulation due to its traditional choice of the automotive industry [13].

5.0 MODEL SIMPLIFICATION

To simplify the problems, several assumptions are made regarding volute modelling. Area inside volute is assumed as converging nozzle. The nozzle area will mimic the area of actual turbine from inlet through its outlet. Outlet area of turbine is the area around inner circumference of turbine volute or the region at maximum radius of turbine's rotor as illustrated in Figure 5 (a) while Figure 5 (b) shows simplified model of turbine volute.

In actual turbine volute, flow inside is discharged along inner circumference from initial start of turbine inlet. However, in turbine volute simplification model, the flow discharged at the end exit of converging nozzle. To make the simplification model to be more realistic, the exit area of nozzle is made to be equivalence to outlet area of turbine.



(a) (b)

Figure 5: (a) Actual turbine area, (b) Simplified model of turbine volute (Converging nozzle)

6.0 ONE-DIMENSIONAL MODEL

From the simplified model of turbine volute, by considering wall of converging nozzle, temperature of inner wall and outer wall can be calculated. Furthermore the difference between inner and outer wall also can be justified and compared with experimental works by [3].

The boundary conditions for the model are, convection for both inner and outer wall, with convection of exhaust gas flow at inner wall and convection of ambient air at outer wall as illustrated in Figure 6. The governing equations are given as follow:

$$Q_{ext} = h_{ext}A(T_{ext} - T_{inlet}) = \frac{kA}{\Delta x}(T_{inlet} - T_{outer}) = h_{amb}A(T_{outer} - T_{\infty}) \tag{11}$$

$$h_{ext}A(T_{ext} - T_{inlet}) = \frac{kA}{\Delta x}(T_{inlet} - T_{outer}) \tag{12}$$

$$\frac{kA}{\Delta x}(T_{inlet} - T_{outer}) = h_{amb}A(T_{outer} - T_{\infty}) \tag{13}$$

With two equations, which are Equation 11 and 12and having two unknowns, which are temperature of inner wall (T_{inlet}) and temperature of outer wall (T_{outer}) it can be solved. By forming 2 by 2 matrix in MATLAB, and difference between T_{inlet} and T_{outer} are calculated. In this analysis, assuming heat convection coefficient h_{ext} (h_1) which is heat transfer coefficient for exhaust flow and h_{amb} (h_2) which is heat transfer coefficient for ambient condition which are 250 W/m^2 K and 25 W/m^2 K (Typical value of convection heat transfer coefficient for free convection of gasses and forced convection of gasses) respectively. While thermal conductivity of turbine volute is equal to 45 W/m K.

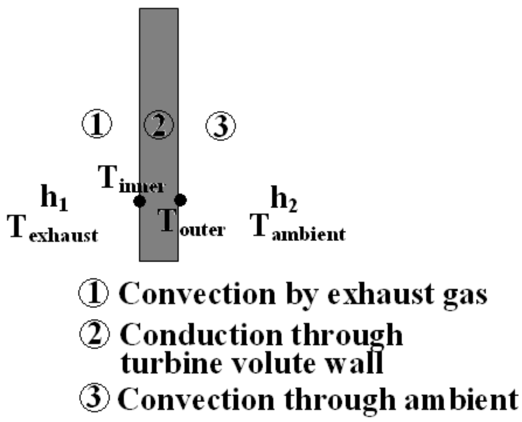


Figure 6: Boundary condition at volute wall for 1-D analysis

The results from the analysis are compared with experimental works by [3]. From the analysis, temperature difference between inner and outer wall is increased when turbine inlet temperature is increased as portrayed in Figure 7. Work by [3] shows that the temperature difference is within the range of 1-7 K, while in this one–dimensional analysis, the temperature difference is 1.25-1.33 K as depicts in Figure 8.

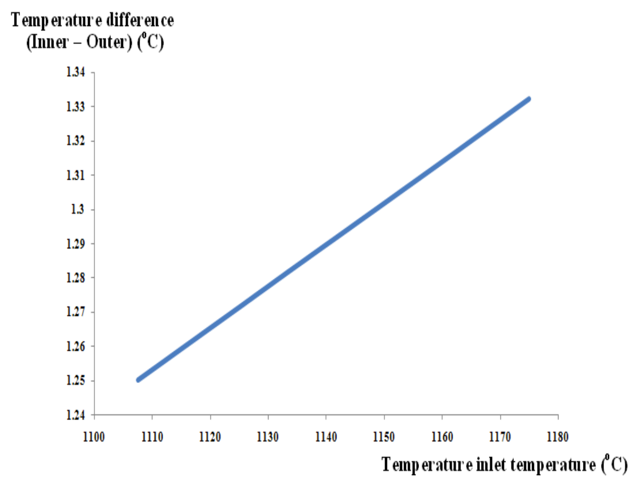


Figure 8: Temperature difference from one-dimensional analysis

7.0 TWO-DIMESIONAL MODEL

Given the converging nozzle model as simplification model, this model can be further extended for two-dimensional analysis with several assumptions. The nozzle circumference is ‘open’ thus, it can be assumed as 2-D flat plate. With Dirichlet and Neumann (at side and up/below surface respectively) boundary conditions, the temperature variation within the flat plate (Nozzle) can be determined. Due to thickness of the nozzle is thin, it can be assume that, turbine inlet temperature and turbine outlet temperature that obtained from experimental value, to be the temperature (Dirichlet B.C) for the flat surface at right and left side respectively as illustrated in Figure 9.

With two-dimensional analysis, temperature contour of volute plate (by thickness) can be seen. From the analysis, turbine inlet temperature is 1175°C, turbine outlet temperature is 1027.5°C and flow temperature (exhaust gas temperature) is taken by taking the average value of turbine inlet and turbine outlet temperature, which is 1101°C. The contour of temperature is illustrated in Figure 10, note that the temperature is in °C (degree Celsius). From the analysis, difference of temperature between inner and outer wall can be calculated. Figure 11 shows temperature difference between inner and outer wall and from the figure it can be seen that the difference is ranging from 1-3°C which still within the range of experimental data from [3].

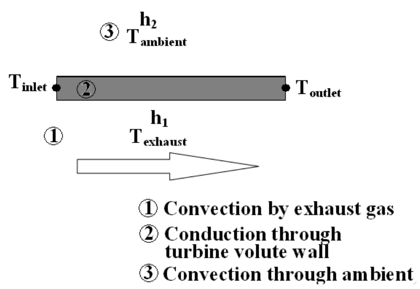


Figure 9: Boundary condition for two-dimensional analysis

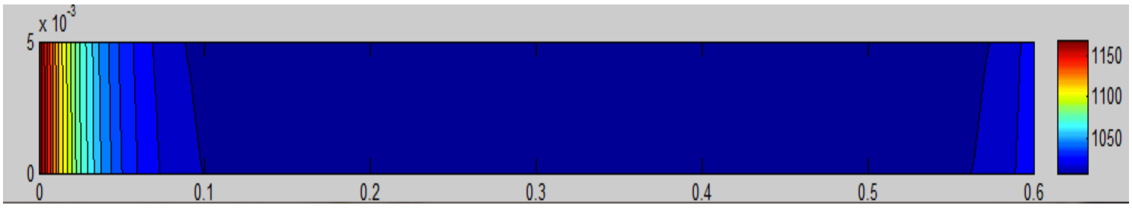


Figure 10 : Temperature contour from 2-D analysis

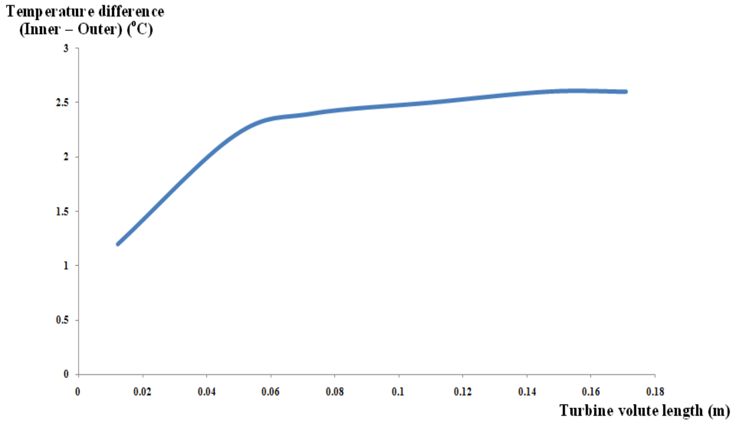


Figure 11: Temperature difference between inner and outer of turbine volute wall

8.0 THREE DIMENSIONAL ANALYSIS

Actual flow physics can be model more accurately in 3D since the working fluid can flow in x, y and z direction in Cartesian grid.Turbine's volute model is drawn based on the given geometry for Holset H3B turbocharger.Based on the dimension given, turbine volute is drawn using SOLIDWORKS. Minor modification has been made to suit meshing criteria in GAMBIT. The model creates using SOLIDWORKS slightly deviates from actual model, which the deviation shows in Figure 12. Model drawn in SOLIDWORKS and meshed in GAMBIT is illustrated in Figure 13 (a) and (b) respectively.

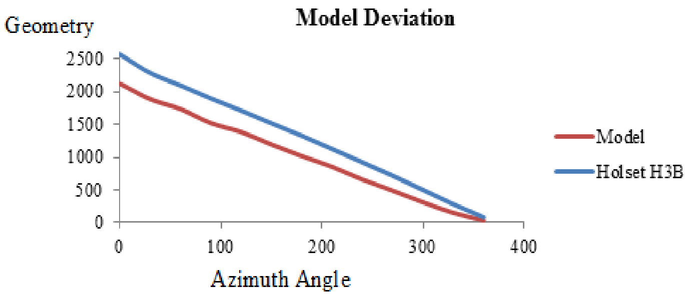
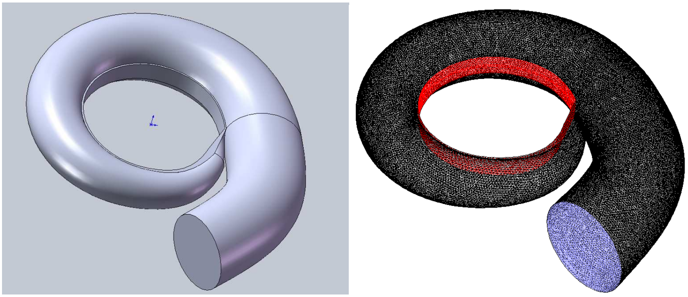


Figure 12 : Model deviation



(a) (b)

Figure 13: (a) Turbine volute model in SOLIDWORKS, (b) Meshed model of turbine volute in GAMBIT

9.0 GRID INDEPENDENT STUDY AND BOUNDARY CONDITIONS

Corrected mass flow parameters and Reynolds number are calculated and plotted over range of number of nodes from sixty two thousands nodes to two hundred and thirty thousands nodes as illustrated in Figure 14 and 15. From the analysis, 199’517 nodes are chosen for further simulation analysis, which is based from grid independent study.

By referring to Figure 14 and 15, the parameters stabilize at two different regions of nodes. Selecting higher region of node, and taking the average of the higher region, is the number of node that is selected for further analysis. Utilizing more nodes will cause longer computation time due to computer needs to resolve more grids. Time for completing computation, depends on computing power.

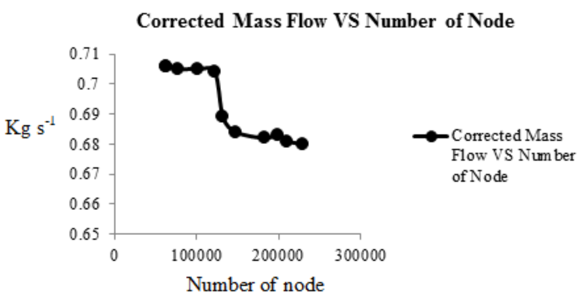


Figure 14: Grid independent study of corrected mass flow

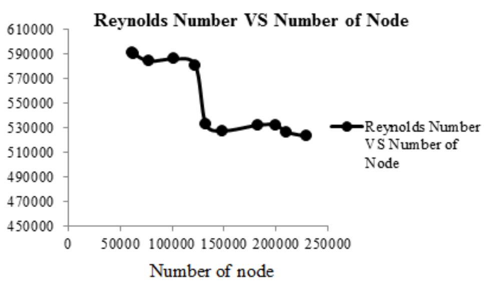


Figure 15: Grid independent study of Reynolds number

Since the value of pressure inlet and outlet of turbine's volute in known from pressure ratio, pressure inlet and pressure outlet boundary condition are used at inlet area and outlet area respectively for simulation in FLUENT. Both boundaries carried inlet and outlet temperature and pressure value with their respective pressure ratio. Boundary condition for inlet and outlet temperature is obtained by experimental data from [2] for analysis of corrected mass flow. For efficiency analysis, experimental data from [11] is utilized. To investigate the heat loss along external turbine's volute surface, convection boundary condition is applied throughout the external surface as illustrated in Figure 16. During the simulation, working fluid (gas) is assumed to be perfect gas (ideal gas), turbulent and in steady state condition [14].

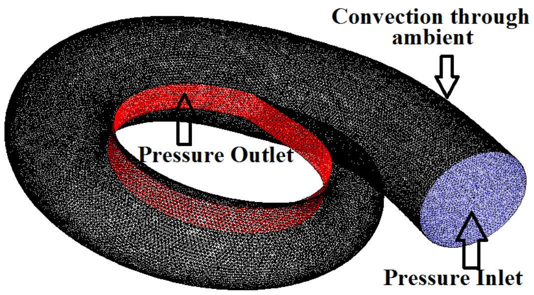


Figure 16: Types of boundary condition for simulation

10.0 CORRECTED MASS FLOW ANALYSIS

Corrected mass flow is one of turbine parameters that reflect turbine performance. It is a parameter that calculates the capability of turbine in swallowing exhaust gas [9]. Different turbine geometry will have different limit of swallowing capacity thus, corrected mass flow parameter will gives the limit of exhaust gas swallowing capability. By keeping fixed value of T_{in} which is 1175K data from [2] and by varying P_{in} based on pressure ratio range (1.1 to 2.0), corrected mass flow is plotted against PR as portrayed in Figure 17. Actual mass flow rate at certain pressure ratio are obtained by FLUENT analysis, is used for calculating corrected mass flow. Non-dimensional corrected mass flow rate is introduced for comparing the trend of the result with other established data as portrayed in Figure 18.

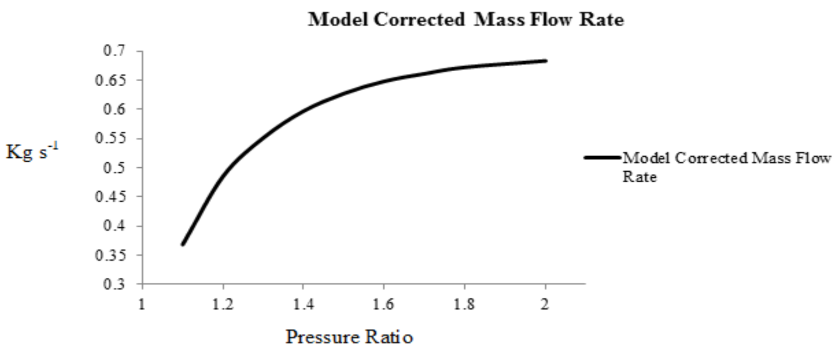


Figure 17: Simulation of corrected mass flow rate

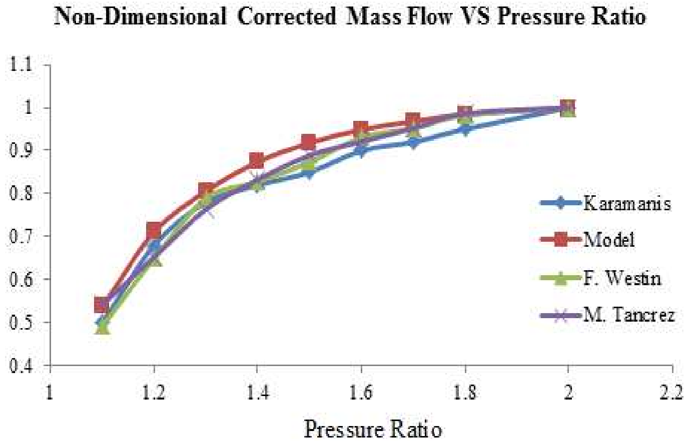


Figure 18: Comparison of non-dimensional corrected mass flow rate

11.0 EFFICIENCY ANALYSIS

For thermal analysis, different experimental data set is used. To obtain realistic simulation for thermal analysis and furthermore will be translated in efficiency analysis, data of turbine inlet temperature and outlet temperature at certain pressure ratio is essential. J. R. Serrano, (2007) gives the best data that can feed into the simulation. Table 1 shows the extracted data from [15].

Table 1: Data extracted from [15]

Pressure Ratio	Inlet Temperature (K)	Outlet Temperature (K)	ΔT
1.9	845	758	87
2.0	850	751	99
2.1	851	742	109
2.2	855	738	117
2.3	850	734	116
2.5	849	730	119
2.7	849	728	121

Turbine's actual work and isentropic work was first analyzed before translating it into turbine's efficiency. Figure 19 shows the difference between actual work and isentropic work. Turbine's efficiency is ratio of turbine's actual power over isentropic power, which is, depicts in Figure 20. Equation (3) is used to calculate turbine's actual power while Equation (6) is used to calculate turbine's isentropic power. Both powers are calculated based on data from Table1 and mass flow value which are obtained from the simulation.

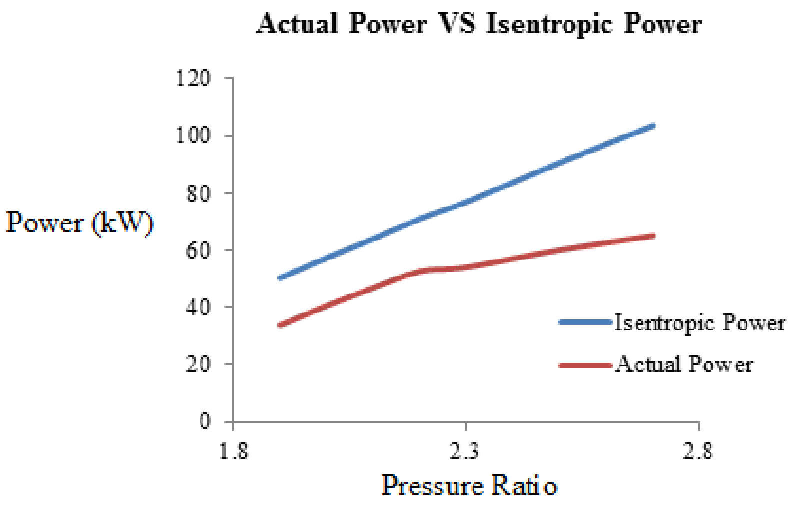


Figure 19: Deviation of turbine actual power with isentropic power

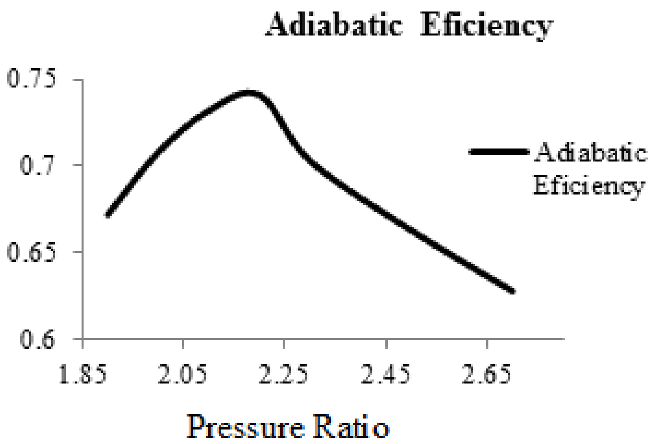


Figure 20: Turbine's adiabatic efficiency

The calculated efficiency is compared with establish data in order to verify the results of simulation. However, established data have different value of efficiency. This is due to different turbine's geometry that is tested and different test condition used in evaluating turbine's efficiency. Figure 21 shows efficiency from simulation and efficiency from several established data at certain range of pressure ratio. In order to investigate similarity between simulation works with established data, non-dimensional efficiency and non-dimensional PR are used as portrayed in Figure 22. Different established data has different set of pressure ratio. They tested turbine at different range of pressure ratio, however with the aid of non-dimensional efficiency and non-dimensional pressure ratio, some degree of similarity can be found.

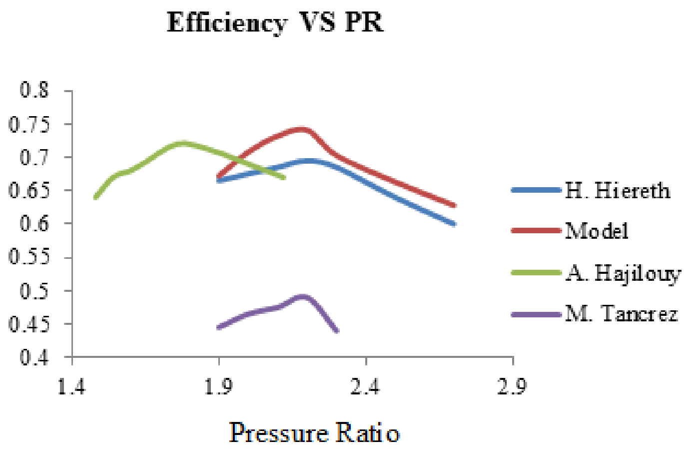


Figure 21: Comparison of adiabatic efficiency

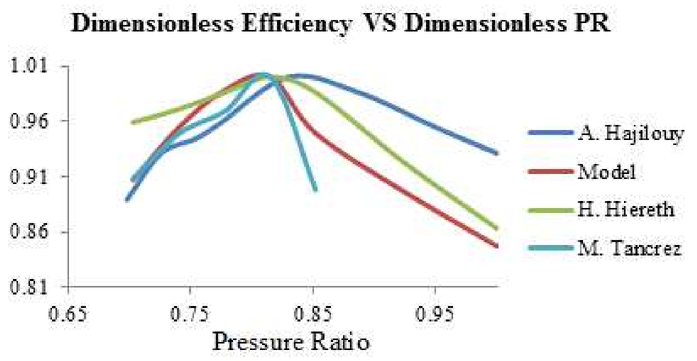


Figure 22: Non-dimensional efficiency versus non-dimensional pressure ratio

12.0 EFFICIENCY ANALYSIS WITH HEAT LOSS

Turbine is simulated with heat loss to mimic the real case condition. Heat transfer through convection to ambient is dominant in turbine heat loss. Since there are no concrete value or serious analysis carried out in determining appropriate heat transfer coefficient for calculating heat loss through ambient, the heat transfer coefficient will be choose based on range given by [16]. In this analysis, 3 different values of heat transfer coefficients are chosen for simulation and to observe the effect towards turbine's efficiency.Table 2 shows case number with their respective value of heat transfer coefficient. The black surface in Figure 23 demonstrates the actual gird of the turbine volute while the red-orange contour represents the temperature value.

Table 2 : Case number with value of heat transfer coefficient

Case 1	250 W/m^2 K
Case 2	2500 W/m^2 K
Case 3	105.486 W/m^2 K

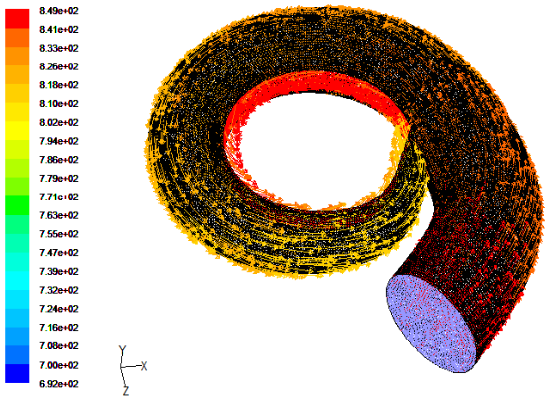


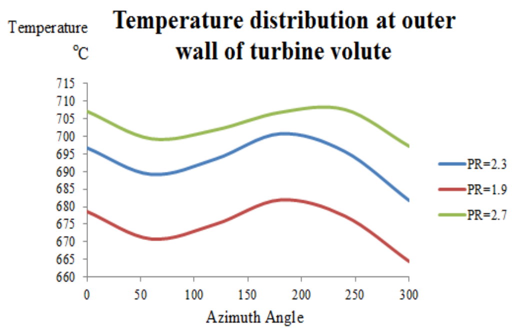
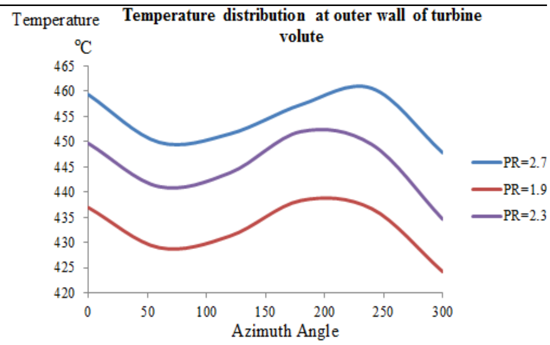
Figure 23:Temperature vector with actual grid (Grid in black colour)

13.0 ANALYSIS OF TEMPERATURE DISTRIBUTION AT OUTER WALL

Temperature at turbine's outer wall is simulated based on Case 1 and Case 2. Maximum temperature obtained from Case 1 and Case 2 are 700^{o}C and 459^{o}C respectively and the comparisons of the temperature distribution between two cases are illustrated in Table 3.It can bee seen that the higher the value of heat transfer coefficient, the lower will be the volute wall temperature. This is due to high forced convection that flows around turbine volute if the coefficient is higher and resulted to more heat been carried away.

The higher the value of pressure ratio, the higher will be the value of temperature distributions. At higher pressure ratio, influx of air to turbine's volute is high, thus will lead to high heat convection from the inside turbine's surface to turbine wall. More heat is absorbed by turbine's wall at high pressure ratio. This is the reason why turbine volute is hot at high pressure ratio as compared to lower pressure ratio.

Table 3: Comparison of temperature distribution along outer wall

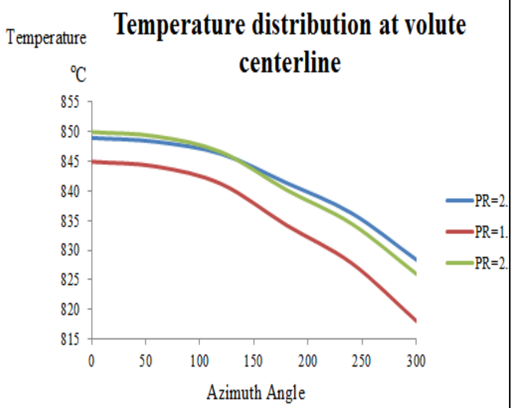
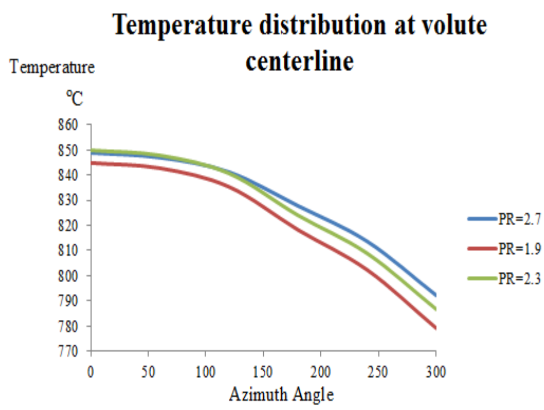
Case 1	Case 2
	

14.0 ANALYSIS OF TEMPERATURE DISTRIBUTION AT TURBINE CENTERLINE

Turbine's centerline temperature is simulated based on Case 1 and Case 2. Moving away from 0^o of Azimuth angle, the temperature drop is steeper due to loss of heat. Temperature drop represents energy is taken out from the control volume (converting heat to power) or energy is lost. Table 4 shows comparison of temperature drop along volute centerline at range azimuth angle (0^o – 300^o).

The temperature drop for both cases is almost similar. However, the final value of temperature drop (at Azimuth angle of 300^o) is different. The final temperature value for Case 2 is slightly less than Case 1. This is due to more heat is rejected to ambient due to high forced convection (high heat transfer coefficient).

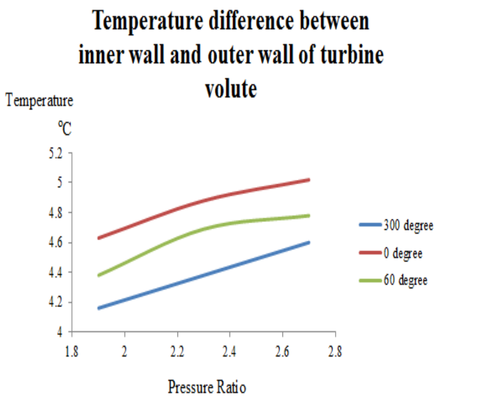
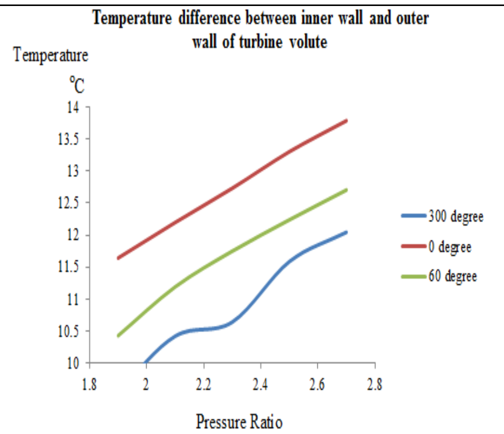
Table 4: Comparison of turbine's centerline temperature

Case 1	Case 2
	

15.0 ANALYSIS OF TEMPERATURE DIFFERENCE BETWEEN INNER AND OUTER TURBINE'S WALL

Difference of temperature is calculated by subtracting inner wall temperature with outer wall temperature. The difference of temperature for Case 2 is higher than Case 1, which is within the range of 10 – 14^oC while for Case 1, 4 – 5^oC. Table 5 shows the comparison of difference between inner and outer turbine's wall temperature.

Table 5: Comparison of temperature difference between inner and outer turbine's volute wall

Case 1	Case 2
	

From the analysis taken for case heat transfer coefficients equal to 250 W/m^2 K (Case 1), it can be can seen that increasing pressure ratio, will increase the temperature of outer wall. When turbine volute sustains high temperature at high pressure ratio, while ambient temperature remains the same, this will creates high gradient of temperature region thus the rate of heat transfer will increase and reduce turbine's efficiency. This is the reason why turbine efficiency is lower at high pressure ratio. The temperature difference is compared with experimental works by [3]. The experimental work shows difference of temperature between inner and outer wall is increased if there is external forced convection flow around turbine volute, which is illustrated in Figure 24. The higher the mass flow rate, the higher will be the temperature difference and external ventilation (forced convection) is another factor that caused significant increase in temperature difference.

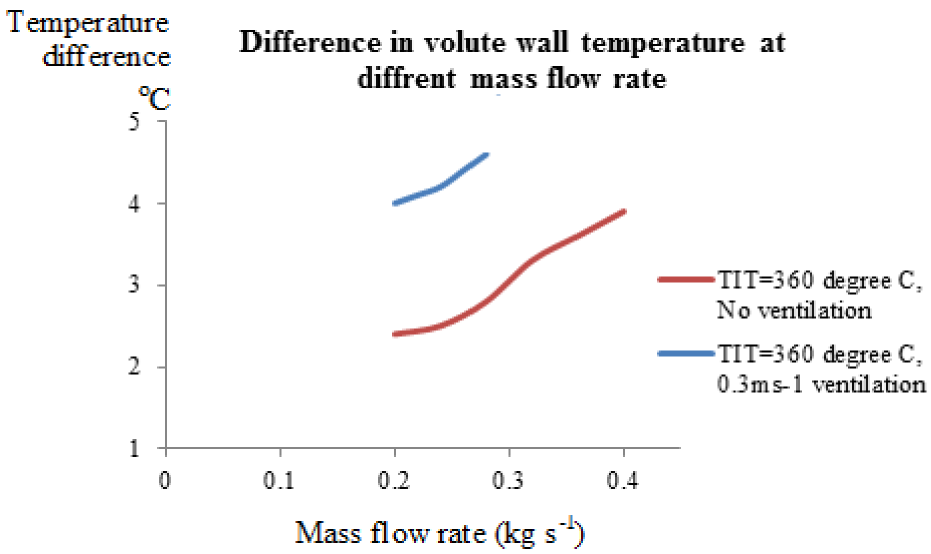


Figure 24 : Difference of temperature between inner and outer turbine's volute wall [3]

Based on Table 5, the lower Azimuth angle the higher will be the temperature difference. At lower Azimuth angle, flow of gas inside turbine's volute posses high temperature due to less heat loss and less energy taken out from the flow. Since the flow having high temperature at early stage of Azimuth angle while the ambient temperature is still the same, this will creates high temperature gradient. Good agreement was found based on the results which is main heat loss occur before exhaust gas strike turbine's blade or upstream of turbine rotor [15].

16.0 ANALYSIS OF TURBINE'S EFFICIENCY

Adiabatic efficiency is the highest efficiency due to no heat loss or heat interaction takes into account during its calculation and it served as benchmark for turbine characteristics, which included in manufacturer turbine's map. However the efficiency is differ based on tested condition. Turbine operates at hostile environment which is to be its actual operating condition, thus adiabatic efficiency will gives information with certain degree of error.The actual powers are analyzed with heat loss effect based on Case 1, 2 and 3. Different cases have different value of efficiency. Figure 25 compares the efficiency for different cases.

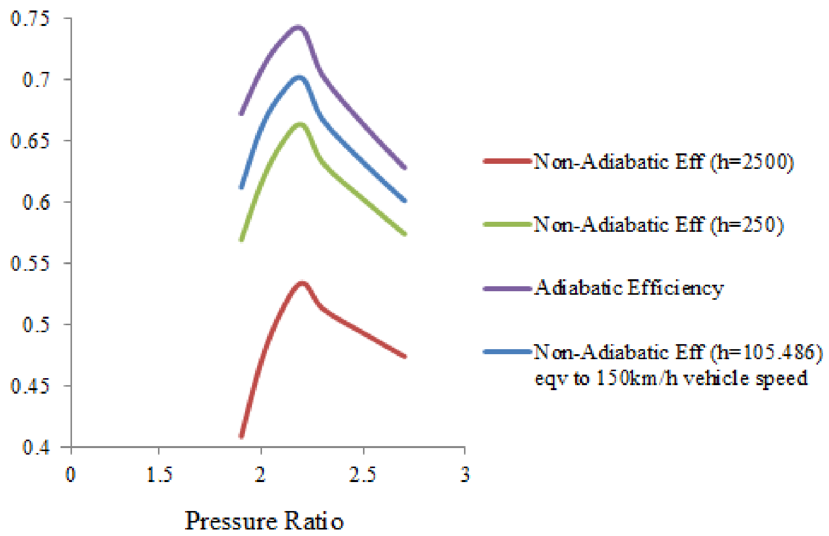


Figure 25: Comparison of efficiency for different cases

The higher the value of heat convection coefficient, the lower will be the efficiency. This is due to, the coefficient that represent how strong is the forced convection occur along turbine volute. Stronger forced convection, will caused high temperature gradient and since the volute thickness is thin, high temperature gradient will cause more heat loss furthermore deteriorate turbine's efficiency. Higher heat convection coefficient will caused high temperature gradient at turbine wall furthermore will cause heat loss and deteriorate turbine's efficiency.

17.0 CONCLUSIONS

Heat transfer that occurs in actual turbine's operation is something that cannot be neglected since it gives significant effect towards turbine's efficiency. Turbine maps given by turbocharger manufacturer are made by running the turbocharger in laboratory condition or in turbocharger test benches [8]. Without consideration of pulsating flow and heat loses during mapping of turbine map, its actual condition cannot be predict correctly [17].

External heat transfer or heat loss through turbine volute via convection and radiation accounts 70% of the total heat loss is agreed by several authors. Simulation from Case 2 predicts the heat loss approximately 68.7% on average. It was a good result. However to come out with such value of coefficient it is quite generic assumption. Convection heat transfer coefficient for forced air ranging from 25 – 250 W/m^2 K, while convection heat transfer coefficient for forced liquid, the value is above 2000 W/m^2 K. Thus, the coefficient for Case 2 only valid for forced liquid. However, since radiation also plays significant role in turbine external heat loss, by assuming Case 2 coefficient is a lumped value for both convection and radiation, the model still can be used to evaluate the condition of 70% of total heat loss, is the heat loss through turbine's volute that was mentioned by several authors. The model made for Case 2 is for further investigation on how heat transfer coefficient affecting turbine's efficiency.

The key attribute that affecting turbine's efficiency is temperature difference between inner wall and outer turbine's wall. Since turbine thickness is only few millimeters (in the model is 5mm), small value of temperature difference would tend to deteriorate turbine's efficiency.

ACKNOWLEDGEMENTS

The authors wish to thank Universiti Teknologi Malaysia and the Malaysian Government for supporting this research activity.

REFERENCES

1. Crouse and Anglin, 1993. *Automotive Mechanics*, Mc Graw-Hill International Editions.
2. Fredrik Westin, Jorgen Rosenqvist and Hane-Erik Angstorm, 2004. Heat losses from the turbine of a turbocharged SI-Engine – Measurement and Simulation, *SAE Technical Paper Series.*
3. Nick Baines, Karl D. Wygant and Antonis Dris, 2009. The analysis of heat transfer in automotive turbocharger, *Proceedings of ASME Turbo Expo 2009 : Power for land, Sea and Air.*
4. A. Romagnoli, Ricardo Martinez-Botas and S. Rajoo, 2011. Steady state performance evaluation of variable geometry twin-entry turbine. *International Journal of Heat and Fluid Flow.*
5. Bohn D, Tom Heuer and Karsten Kusterer, 2005. Conjugate flow and heat transfer investigation of a turbocharger, *Journal of engineering for gas turbines and power*
6. Mickael Cormerais, Pascal Chesse and Jean-Francois Hetet, 2009. Turbocharger heat transfer modeling under steady and transient conditions, *International Journal of Thermodynamics.*
7. N. Watson and M. S. Janota, 1982. *Turbocharging the internal combustion engine*, Wiley
8. M. Tancrez, J. Galindo, C. Guardiaola, P. Fajardo and O. Varnier, 2011. Turbine adapted maps for turbocharger engine matching, *Experimental thermal and fluid science* 35 (146-153).
9. A. Romagnoli and Ricardo Matinez-Botas, 2009. Heat transfer on a turbocharger under constant load points, *Proceedings of ASME Turbo Expo 2009 : Power for land, Sea and Air.*
10. Hermann Hiereth and Peter Prenniger, 2007. *Charging the internal combustion engine*, Springer Verlag, Wein
11. http://www.pipeflow.uk December 2011.
12. Jurij Sodja, 2007. *Turbulence models in CFD*. Master thesis University of Ljubljana, Faculty for Mathematics and Physics.
13. Rafael Cavalcanti de Souza and Guenther Carlos Kriger Filho, 2011. Automotive turbocharger radial turbine CFD and comparison to gas stand data, *SAE Technical paper series.*
14. G. Descombes, J. F. Pichouron, F. Maroteaux, N. Moreno and J. Jullien, 2002. Simulation of the performance of a variable geometry turbocharger for diesel engine road propulsion, *Int. J. Applied Thermodynamics*
15. Srithar Rajoo, 2007. *Variable geometry-active control turbocharger* , PhD thesis at Imperial College London.
16. Yunus A. Cengel and Afshin J. Ghafar, 2011. *Heat and mass transfer fundamentals and applications – Fourth edition in SI units*, McGraw Hill Higher Education
17. M. S. Chiongg, S. Rajoo, R. F. Martinez-Botas, A. W. Coastall, 2011. Inlet boundary condition study for unsteady turbine performance prediction using 1-D modeling, *Jurnal Mekanikal* 33, 89 – 104.