

A Study of Steady-State Thermal Distribution on Circular Plate Using ANSYS

F. Jikol¹, M. Z. Akop^{1,*}, Y. M. Arifin¹, M. A. Salim¹ and S. G. Herawan²

¹Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, 76100 Durian Tunggal, Melaka, Malaysia.

²Industrial Engineering Department, Faculty of Engineering, Bina Nusantara University, Indonesia

ABSTRACT

The purpose of this study is to investigate the thermal behavior of a circular-shaped hot plate when certain thermal load is applied. The hot plate is a part of hot surface deposition test equipment, and is placed on top of a heater block. The temperature distribution on the hot plate will indicate the highest temperature, and whether the temperature generated is as per required in the hot surface deposition test. Modelling of the heater block and the steady-state thermal analysis is conducted experimentally using ANSYS Release 16.2 software. The maximum temperature of the hot plate is compared to the applied temperature on the heater block. It is shown that there is dissimilarity of temperature between the hottest spot on the hot plate surface and the temperature applied. When conducting the real deposition test, the exact required temperature of the hot plate can be obtained by altering the temperature setting and measured accurately by using thermometer.

Keywords: Heat flux, hot plate, meshing, steady-state thermal, temperature

1. INTRODUCTION

Hot surface deposition test is a simplified experimental method that is developed to study the effect of type of fuel and temperature on deposits formation in diesel engine combustion chamber [1]. This method is used as a simulation for researcher to gain better and deeper understanding of complex fuel deposits behaviors and properties, and at the same time investigate the negative effects of deposits on engine performance. In the previous researches, most of the experimental studies on fuel and engine performance were carried out by implementing more complicated equipment, especially the real engine. Thus, hot surface deposition test method is an alternative to the real engine test which is more complex, costly, and require longer run time as been investigated by Suryantoro et. al. where he used a real engine test to study the growth and characterization of deposits in the combustion chamber of a diesel engine. In his study, the engine was running for 200 hours on various parameters [2]. Husnawan et. al. has also applied real engine in his research on cylinder head carbon deposits where the engine was operating for 25 hours on various set of test [3]. This longer duration of engine test is obviously required higher cost as the consumption of energy such as fuel and electricity during the test would be higher.

Nevertheless, despite the complexity of real engine test, it could provide more experimental data such as engine emissions, brake specific fuel consumption, and engine brake thermal efficiency [4-7], which is one of the main limitations of hot surface deposition test. Equipment used in the hot surface deposition test consist of heater block which is heated by heater rods and circular hot plate where fuels are dropped to stimulate fuel impingements in engine combustion chamber part, especially the piston. Material used for the heater block and heater

*Corresponding author: zaid@utem.edu.my

rods is structural steel while aluminum alloy is used for the hot plate. The hot plate is fabricated from an aluminum alloy to represent piston, as pistons inside internal combustion engine (IC) are mostly constructed from aluminum alloy [8-11]. However, temperature distribution on the hot plate in hot surface deposition test is not yet described briefly. Information and overview of steady-state thermal behavior of circular aluminum alloy plate especially for hot surface deposition test is still inadequate in the previous literature. This circumstance might be caused by a very limited application on this type of equipment, unlike those experiments using real engine in the earlier study. For a simple solid model that did not involve fluid flow, a steady-state thermal analysis is adequate to stimulate the heat distribution on the solid model as has been investigated by [12-17] in their respective study. In those studies, those researchers focused on thermal behavior within the solid model without external fluid force such as air flow and water impingement.

Steady-state thermal in ANSYS could provide a simulation and data such as temperature distribution and total heat flux. For a work involving fluid flow simulation, such as heat exchanger, cooling system, and steam turbine, Computational Fluid Dynamics (CFD) in ANSYS can be used to analyze complex parameters and produce more precise results [18-21]. With CFD, users could achieve better solution and result of the fluid flow analysis. Whether it is a flow of a liquid or gases, its interaction with certain geometry defined by boundary conditions could be stimulated and analyzed. In this study, the steady-state thermal analysis was conducted by using one of the popular computer aided engineering (CAE) software, which is ANSYS Release 16.2 software. Other than ANSYS, there are program which is used globally especially in engineering field such as AutoCAD and SOLIDWORKS, but these programs are often used for producing designs with more realistic rendering. Contrary, ANSYS software is more focused on providing a more accurate analysis and simulation. For conducting various type of analysis and simulation, ANSYS is capable of producing a more reliable and complete results. Due to its exceptional efficiency and reliability, CAE software such as mentioned before has become a trend in manufacture industry [22].

ANSYS software in particular is widely used by professionals to conduct simulation on interactions of various fields such as physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic [14,23]. It is also used in some higher learning institutes for students especially those whom enrolled in engineering courses. In this paper, the main steps in conducting steady-state thermal analysis using ANSYS software has been presented. The importance of those steps has been highlighted to ensure a proper analysis result. For a circular-shaped aluminum alloy plate particularly, we aimed to study the thermal effect on the plate at certain temperatures. The results obtained and discussions made in this study could be referred in conducting future research regarding steady-state thermal distribution on a solid body using ANSYS. Furthermore, this study aimed to provide a better understanding on thermal effect on aluminum alloy hot plate in hot surface deposition test so that future improvement on the existing experimental setup could be achieved.

2. METHODOLOGY

2.1 Modelling of Heater Block

Solid model could be either imported from other CAD software such as AutoCAD and SOLIDWORKS or created in ANSYS preprocessor. For this study, 3D modelling of the heater block was designed in ANSYS Design Modeler workbench, as shown in Figure 1. The solid model consists of three different part which are hot plate, heater block, and heater rods. Material used for the solid model was structural steel for the heater rod and heater block, and aluminum alloy to represent piston, as pistons inside internal combustion (IC) engine are mostly constructed from aluminum alloy [8]-[11]. Table 1 shows material properties of aluminum alloy. Material

properties of both structural steel and aluminum alloy were imported from ANSYS Engineering Data.

Table 1 Properties of aluminum alloy (Al-Si)[11]

Property	Value
Bulk Modulus	67 GPa
Density	2.0 g/cm ³
Young's Modulus	69 GPa
Elongation at Break	9-25 %
Electrical Conductivity	40-58% IACS
Fatigue Strength	55-97 MPa
Hardness	25-95 MPa
Shear Modulus	34 GPa
Shear Strength	70-207 MPa
Specific Heat Capacity	910 J/kg.°C
Ultimate Strength	310 MPa
Yield Tensile Strength	276 MPa
Thermal Conductivity	155 W/m.K

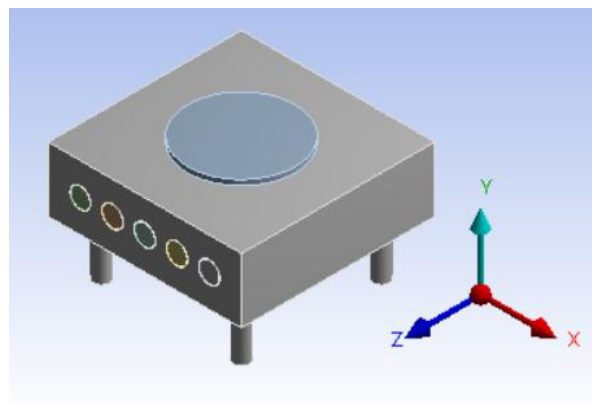


Figure 1. Heater block model.

2.2 Meshing

Meshing is the principal and most important step in ensuring good analysis result. In reference to Noor et. al, he stated that result of an analysis is highly depending on the quality of meshing. Quality of meshing will determine whether the result produced is good or will contain error [18]. Mesh size also influences the responses of the solution analysis and affecting value of heat flux. More complex mesh will increase the computational time and affect final solution [24]. To create meshing element on solid geometry, solid model could be either imported from other CAD software such as AutoCAD and SOLIDWORKS or created in ANSYS preprocessor. There are two common mesh element type, which is hexahedral elements and tetrahedral elements. For regular shaped rectangular type volumes, hexahedral elements or as well as indicated as bricks can be used to mesh the respective volumes. In the other hand, volume with three or four node points in a plane is more suitable to be meshed using tetrahedral element [13].

For this analysis, the combined solid model of hot plate, heater block, and heater rods were used in ANSYS mesh element generation. Figure 2(a) shows initial mesh of the solid model using ANSYS default meshing setting, which is Mechanical Physic Preference. In this geometry

condition, the number of nodes and elements count generated were 11554 and 4697 respectively. It is clear that the mesh elements size generated using default setting is much bigger. Thus, to improve the quality of meshing, remeshing is done by changing the mesh setting to CFD Physic Preference as can be seen in Figure 2(b). In the CFD Physics Preference, the number of nodes has increased to 55305, while for elements count is 204149. Furthermore, the mesh elements size was significantly reduced, and this will lead to better thermal distribution analysis.

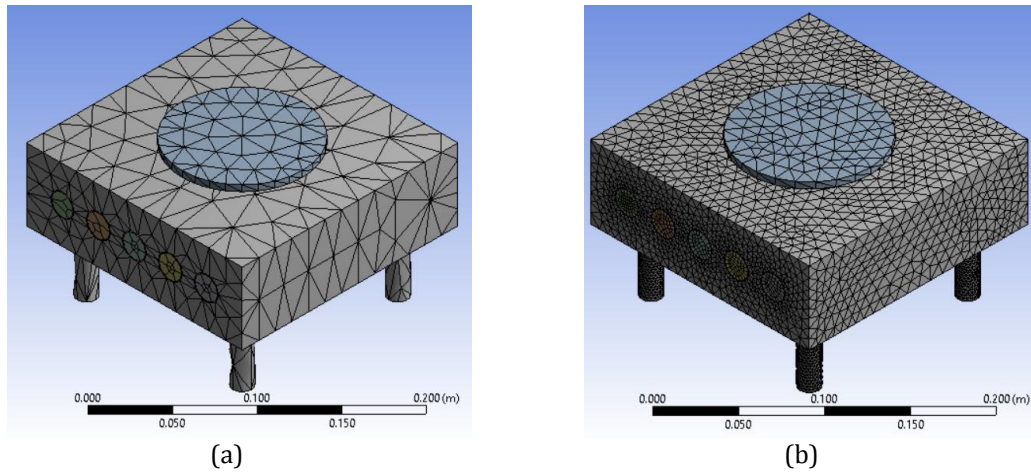


Figure 2. (a) default meshing using Mechanical Physic Preference and (b) improved meshing using CFD Physic Preference.

2.3 Applying Boundary Conditions

Before generating the thermal analysis result, boundary and initial conditions must be set in the ANSYS workbench. Correct and suitable conditions must be applied to avoid error in generating results. Initial condition of the solid model was set at 22 °C, as per default setting in ANSYS. Thermal load (100 °C, 200 °C, 300 °C, 400 °C, 500 °C, and 600 °C) applied on the heater rods. Those heater rods were treated as bonded-contact condition with the heater block. Thus, heat will be conducted from the heater rods onto inner surface of the heater block. Similarly, the heater block is transmitting heat onto the hot plate part. In Figure 3, convection was set to occur on the surface of the hot plate and heater block. Two convection solution is added differently because the heater block and hot plate is different in term of material used. Heat was emitted to the surrounding from these selected surfaces.

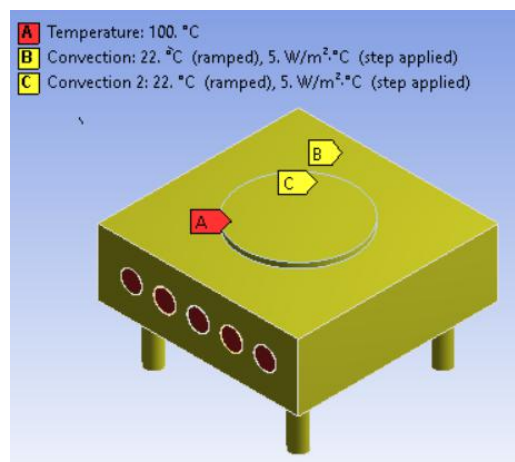
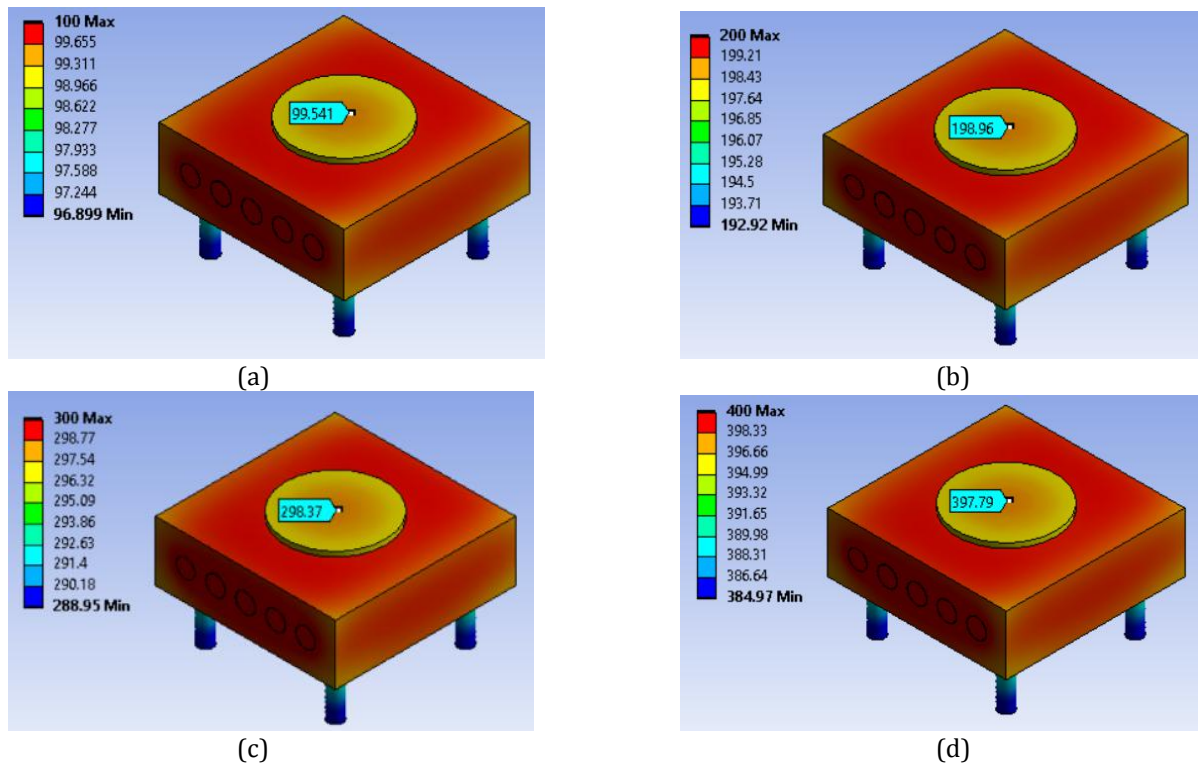


Figure 3. Convection condition applied on heater block and hot plate.

3. RESULTS AND DISCUSSION

3.1 Temperature Distribution Analysis

After meshing and boundary conditions were applied to the heater block model, the thermal solution can be generated. Temperature legend on the left side of the solid model is divided by certain number of color contour. The number of color contour depends on complexity of the solution generated. For this analysis, the red color contour indicates the highest temperature generated while lowest temperature is indicated by dark blue color contour. As can be seen in Figure 4(a), the lowest temperature is dissipated along the base of the heater block. The highest temperature is distributed on the surface of the heater block when 100 °C of thermal load is applied. The maximum temperature generated is same as the temperature applied on the heater rods as the parts are in contact and possesses the same type of material. For the hot plate part, temperature probe is applied to indicate the maximum temperature generated on the hot plate. The temperature probe shows that the maximum temperature is located on the center part of the hot plate. This condition is in agreement with previous researches which indicates highest temperature occurred on center part of piston top [8], [13], [25]. However, the temperature generated is 99.541 °C, which is slightly lower than that of temperature applied. This is due to different material type of the heater block and hot plate. As the analysis is conducted by applying different temperature of 200 °C-600 °C, it is observable that the difference between the hot plate maximum temperature and applied temperature is also increasing, as shown in Table 2 and Figure 5.



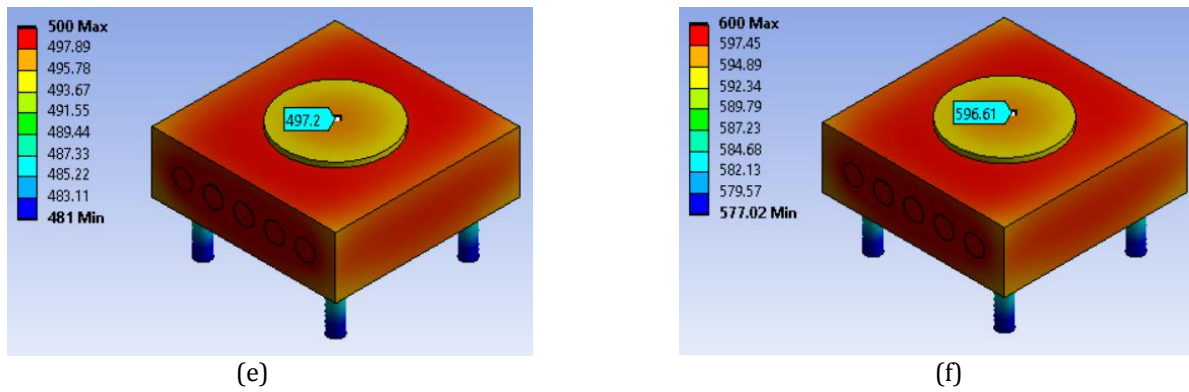


Figure 4. Applied temperature, (a) 100°C, (b) 200°C, (c) 300°C, (d) 400°C, (e) 500°C and (f) 600°C.

Table 2 Temperature difference between hot plate surface and temperature applied

Applied temperature (°C)	Maximum temperature on hot plate (°C)	Temperature difference (°C)
100	99.54	0.46
200	198.96	1.04
300	298.37	1.63
400	397.79	2.21
500	497.20	2.80
600	596.91	3.09

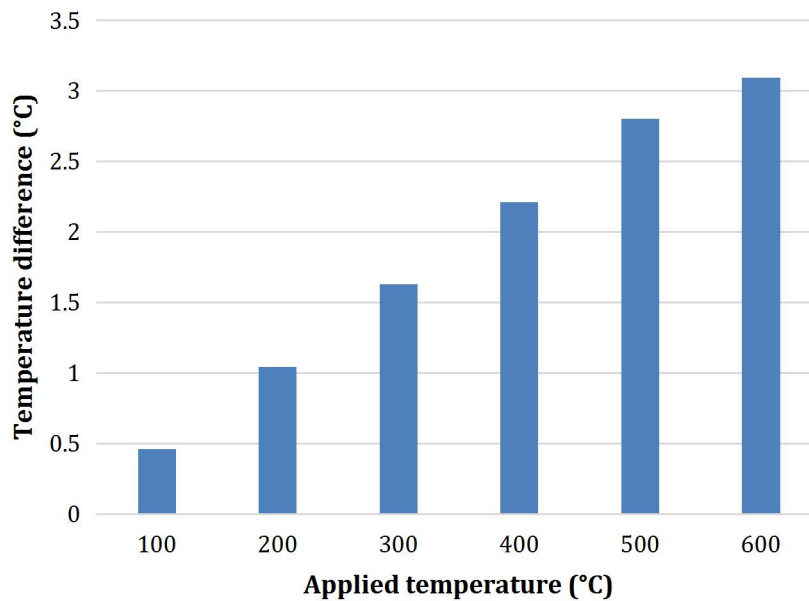


Figure 5. Temperature difference between hot plate surface and temperature applied.

3.2 Heat Flux Analysis

Minimum total heat flux generated is more concentrated on the center part of the hot plate surface. It is indicated by the light blue color contour. On the other hand, maximum total heat flux is indicated by the light green color contour as illustrated in Figure 6(a)-(f). As can be seen in Figure 6(a)-(f), the total heat flux goes on decreasing just a little over from the side of the hot plate towards the center of the hot plate. As being mentioned by Kushwah and Dhakar, higher total heat flux means higher rate of heat transfer [16]. This condition shows that the highest rate

of heat transfer occurred on the higher total maximum heat flux, which is on the light green color contour area. In other words, the temperature will increase quicker on the light green color contour area than that of the light blue color contour area. When the applied temperature is increased, the total heat flux generated on the hot plate also increased as shown in Table 3 and Figure 7.

Table 3 Total heat flux generated on hot plate surface

Applied temperature (°C)	Maximum total heat flux on hot plate (W/m ²)	Minimum total heat flux on hot plate (W/m ²)
100	2754.40	812.17
200	6174.70	1903.90
300	9295.20	3011.40
400	12924.00	4536.60
500	17024.00	5502.00
600	21412.00	6653.00

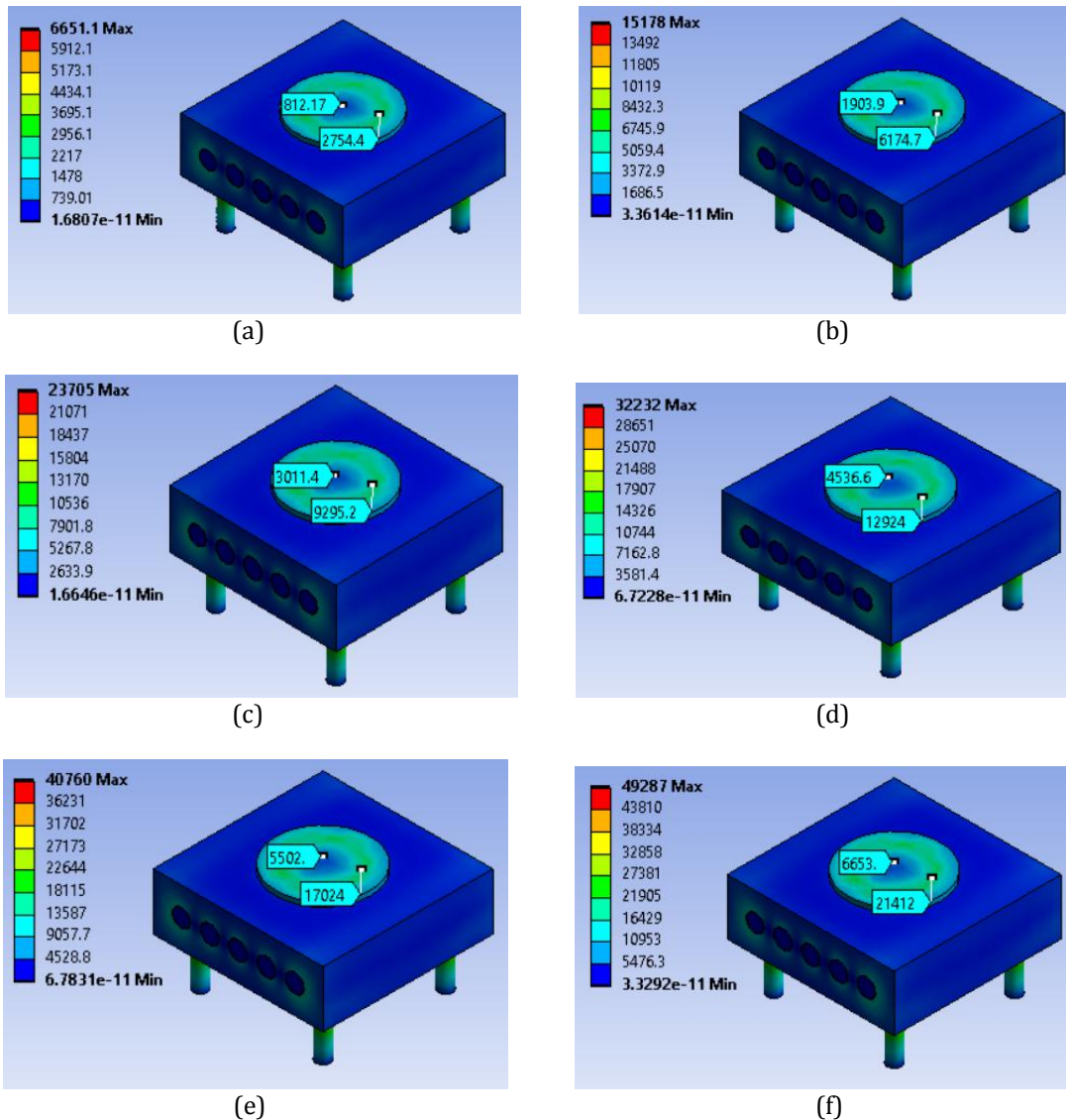


Figure 6. Applied temperature (a) 100°C, (b) 200°C, (c) 300°C, (d) 400°C, (e) 500°C and (f) 600°C.

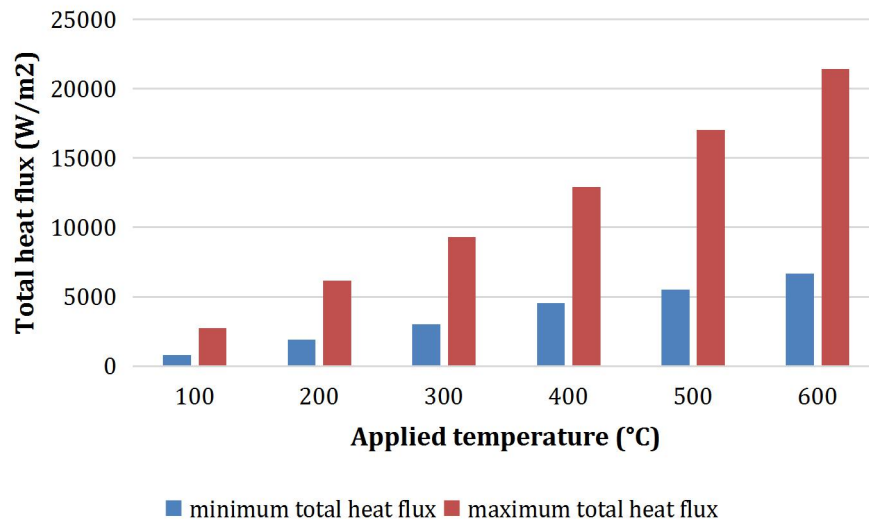


Figure 7. Total heat flux generated on hot plate surface.

4. CONCLUSION

The study of thermal analysis of hot plate was successfully done by using ANSYS Release 16.2 software. Major conclusions of this study are drawn below.

- i. Type of contact between solids and selection of convection area on the solid geometry must be defined correctly in ANSYS to ensure correct analysis result.
- ii. Meshing process is important to produce good analysis result. Selection of mesh setting and parameter must be suitable with the model and type of analysis.
- iii. The hottest spot on the hot plate was indicated, which is at the centre and is in agreement with existing researches where highest temperature occurred on centre part of piston top in combustion chamber.
- iv. As the maximum temperature area has been indicated, it is suitable for fuel to be dropped on the centre area of the hot plate in hot surface deposition test experiment.
- v. There is a difference between the applied temperature and temperature generated on the hot plate. When conducting the real deposition test, the exact required temperature of the hot plate can be obtained by altering the temperature setting and measured accurately by using thermometer.

ACKNOWLEDGEMENTS

This work was supported by Fundamental Research Grant Scheme (FRGS) – FRGS/1/2020/FKM-CARE/F00435. The authors would like to thank the staff of the Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka for their supports.

REFERENCES

- [1] Y. M. Arifin & M. Arai, “The effect of hot surface temperature on diesel fuel deposit formation,” *Fuel*. **89**, 5 (2010) pp.934–942.
- [2] M. T. Suryantoro, B. Sugiarto, & F. Mulyadi, “Growth and characterization of deposits in the combustion chamber of a diesel engine fueled with B50 and Indonesian biodiesel fuel (IBF),” *Biofuel Res. J.* **3**, 4 (2016) pp.521–527.
- [3] M. Husnawan, H. H. Masjuki, T. M. I. Mahlia, & M. G. Saifullah, “Thermal analysis of cylinder head carbon deposits from single cylinder diesel engine fueled by palm oil-diesel fuel emulsions,” *Appl. Energy*. **86**, 10 (2009) pp.2107–2113.
- [4] A. Alahmer, J. Yamin, A. Sakhrieh, & M. A. Hamdan, “Engine performance using emulsified

- diesel fuel,” *Energy Convers. Manag.* **51**, 8 (2010) pp.1708–1713.
- [5] O. M. Ali, R. Mamat, N. R. Abdullah, & A. A. Abdullah, “Analysis of blended fuel properties and engine performance with palm biodiesel-diesel blended fuel,” *Renew. Energy.* **86**, (2015) pp.59–67.
- [6] C. S. Cheung, L. Zhu, & Z. Huang, “Regulated and unregulated emissions from a diesel engine fueled with biodiesel and biodiesel blended with methanol,” *Atmos. Environ.* **43**, 32 (2009) pp.4865–4872.
- [7] O. A. Elsanusi, M. M. Roy, & M. S. Sidhu, “Experimental Investigation on a Diesel Engine Fueled by Diesel-Biodiesel Blends and their Emulsions at Various Engine Operating Conditions,” *Appl. Energy.* **203**, (2017) pp.582–593.
- [8] E. Buyukkaya & M. Cerit, “Thermal analysis of a ceramic coating diesel engine piston using 3-D finite element method,” *Surf. Coatings Technol.* **202**, 2 (2007) pp.398–402.
- [9] I. Doetein, L. Muremi, & P. Bokoro, “An Analysis of geometry characteristics of metal oxide blocks on heat flow rate using Finite Element Analysis,” 2020 IEEE Electr. Insul. Conf. EIC 2020, (2020) pp.163–166.
- [10] S. Ganapathy Pandian, S. Puzhuthivakkam Rengarajan, T. P. Babu, V. Natarajan, & H. Kanagasabesan, “Thermal and Structural Analysis of Functionally Graded NiCrAlY/YSZ/Al₂O₃ Coated Piston,” *SAE Int. J. Mater. Manuf.* **8**, 2 (2015) pp.578–585.
- [11] P. V. Srikanth, B. V. V. P. Rao, K. M. Laxmi, & N. HariBabu, “Material coating optimization and thermal analysis of a four stroke C I engine piston,” *Int. J. Mech. Eng. Technol.* **8**, 8 (2017) pp.988–997.
- [12] A. Tariq, K. Altaf, A. M. A. Rani, M. Baharom, & A. R. A. Aziz, “Study of heat transfer attributes of custom fins for Crank-Rocker engine block using ANSYS,” *J. Adv. Res. Fluid Mech. Therm. Sci.* **62**, 2 (2019) pp.235–243.
- [13] S. A. Havale & P. S. Wankhade, “Design, thermal analysis and optimization of a piston using ansys,” *Int. Res. J. Eng. Technol.* **4**, 12 (2017) pp.1311–1317.
- [14] J. Jaivignesh *et al.*, “Study of one dimensional conduction heat transfer for constant thermal conductivity through composite plane slab and in cylinder at steady state condition,” *Int. J. Mech. Eng. Technol.* **8**, 11 (2017) pp.457–466.
- [15] R. Kumar, D. Singh, & A. K. Sharma, “Static Thermal Analysis of Fins Models Using Ansys,” *Int. J. Mech. Eng. Technol.* **11**, 2 (2020) pp.10–21.
- [16] B. Kushwah and P. S. Dhakar, “Temperature Distribution Analysis of Cylinder Block with Fins for Various Materials and Different Geometry,” *Int. J. Res. Anal. Rev.* **6**, 2 (2019) pp.552–558.
- [17] Y. Nietiadi, C. Vidal, R. Luís, & P. Varela, “Thermal analyses of the in-vessel frontends of the ITER plasma position reflectometry system,” *Fusion Eng. Des.* **156**, January (2020) pp.2–8.
- [18] T. Yusof, M.M.Noor, & Andrew P.Wandel, “Detail Guide for Cfd on the Simulation of Biogas Combustion In Bluff-Body Mild Burner,” *Int. Conf. Mech. Eng. Res.* July (2013) pp.1–3.
- [19] A. Raza, H. Mehboob, S. Miran, W. Arif, & S. F. J. Rizvi, “Investigation on the characteristics of biodiesel droplets in the engine cylinder,” *Energies.* **13**, (2020) pp.1–14.
- [20] U. J. Ali, O. A. Mohamad, M. S. Khan, & M. S. Gadala, “Cooling of hot steel plate on run-out table facility; A simulation,” 2020 Adv. Sci. Eng. Technol. Int. Conf. ASET 2020, (2020) pp.51–56.
- [21] S. Silvester, “Excellence in Engineering Simulation (ANSYS SPORTS),” *ANSYS.* **6**, 2 (2012) pp.10–14.
- [22] K. Dou, E. Lordan, Y. J. Zhang, A. Jacot, & Z. Y. Fan, “A complete computer aided engineering (CAE) modelling and optimization of high pressure die casting (HPDC) process,” *J. Manuf. Process.* **60**, October, (2020) pp.435–446.
- [23] G. Grebenisan, I. E. RADU, & I. Anton, “Finite Elements Analysis of Thermal Steady State, Using Ansys,” *Ann. ORADEA Univ. Fascicle Manag. Technol. Eng.* **2** (2014) pp.108–112.
- [24] S. L. M. Ribeiro Filho, M. O. Gomes, C. H. Lauro, & L. C. Brandão, “Definition of the Temperature and Heat Flux in Micromilling of Hardened Steel Using the Finite Element Method,” *Arab. J. Sci. Eng.* **39**, 10, (2014) pp.7229–7239.

- [25] Y. Lu, X. Zhang, P. Xiang, & D. Dong, "Analysis of thermal temperature fields and thermal stress under steady temperature field of diesel engine piston," *Appl. Therm. Eng.* **113**, November (2016) pp.796–812.