International Journal of Nanoelectronics and Materials
Volume 15 (Special Issue) March 2022 [147-157]

Transient Thermal Analysis on Convection Process of Circular Plate Used in Hot Surface Deposition Test Rig

F. Jikol[1], M. Z. Akop[2*], Y. M. Arifin[2], M. A. Salim[2] and S. G. Herawan[3]

[1]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
76100 Durian Tunggal, Melaka, Malaysia
[2]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
76100 Durian Tunggal, Melaka, Malaysia
[3]Industrial Engineering Department, Faculty of Engineering, Bina Nusantara University, Indonesia

ABSTRACT

The purpose of this study is to investigate the transient thermal behavior of the convection process of a circular-shaped aluminum alloy plate used in the Hot Surface Deposition Test (HSDT) rig. The temperature distribution and its characteristics over time are investigated by applying certain thermal load and time dependence. As the real deposition test using HSDT involves a process of detaching the aluminum alloy circular plate from the heater block, the transient thermal analysis will provide a base for estimating the time required for the hot plate to cool down to room temperature (25°C-27°C) and safe to be handled. The process of modeling the geometry, meshing, applying the boundary conditions, and evaluating results are conducted experimentally using ANSYS Release 16.2 software. It is indicated in the analysis results that the aluminum alloy circular plate with higher internal temperature undergoes a longer convection process to cool down to room temperature. These results are used to estimate the cooling down time of the hot plate in the real deposition test.

Keywords: Aluminum alloy, convection, deposits, room temperature, transient thermal

1. INTRODUCTION

Combustion chamber deposits formation inside the internal combustion (IC) engine parts is unavoidable. It is one of the main issues that often occur whether in diesel engines or petrol engines. These problems are more severe towards diesel engines as new alternative fuel such as biodiesel is being gradually implemented in a diesel engine to reduce the dependence on the non-renewable petroleum diesel. Deposits existence in the combustion chamber system such as the injector and piston could deteriorate the overall engine operation and performance. Apart from that, a lack of understanding of the deposits mechanism of biodiesel fuel is contributing to the deposits issues in IC engine systems. Thus, investigation on deposits formation is critical to understand the deposits mechanism and finding a solution that is capable of reducing the negative impacts of deposits on engines. In recent years, there was a huge number of experiments conducted to gain a better understanding of the complex mechanism of fuel deposits. Various types of fuel especially biodiesel are often used by researchers to study fuel deposition behavior. Furthermore, the complexity of deposits formation and behavior are making the investigation on this matter more complicated. In the literature, a common method to study deposits formation is by implementing a real engine test, which is more complex, costly, and requires a longer test duration [1], [2]. A fuel deposition test involving an actual engine test bench is performed to stimulate and observe deposits formation under a particular test condition such as running duration or work load applied to the engine. Despite the constraint on the complexity and high cost, one of the most significant advantages of the actual engine test is that the actual engine test

---

*zaid@utem.edu.my

could deliver more experimental data such as brake specific fuel consumption, brake thermal efficiency, and engine emissions [3], [4]. As for deposits investigation, in particular, there is still a lack of a specific experimental method to investigate deposits mechanism and its severity of impacts towards the real engine.

1.1 Existing HSDT model

In order to simplify the experimental method of deposits investigation, a method known as the hot surface deposition test (HSDT) was developed to stimulate the fuel impingements inside a combustion chamber system, mainly the piston. HSDT method involves fuel impingement on a heated surface mainly to investigate the effect of temperature on deposits formation and mechanism. The main equipment in the application of the HSDT method is the aluminum alloy circular plate and a heat source to electrically heat the plate to the desired temperature. Other than that, a counting system that involves sensor utilization is compulsory to record the number of fuel droplets. One of the earliest employment of this experimental method was carried out by Arifin and Arai where they studied the effect of hot surface temperature on diesel fuel deposits formation [5]. In the following years, more researchers applied the same concept as HSDT to carry out their investigation on deposits formation of fuels, especially to stimulate the effect of wall temperature on deposits formation inside the combustion chamber. Pham [6] and Suryantoro *et al*. [7] had developed a model to investigate the deposits formed by diesel and biodiesel fuel on a heated aluminum circular plate. However, in their study, there was no steady-state nor transient thermal analysis on the aluminum alloy circular plate.

1.2 Transient Thermal Analysis

Heat conduction, heat convection, and heat radiation are the three basic modes of heat transfer. In most cases, at least two or three heat transfer modes are performed simultaneously. For steady-state thermal, the condition of heat changes with time is negligible. On the other hand, transient thermal analysis involves the calculation of temperature distribution and its characteristics over time [8]. Furthermore, the most notable difference between steady-state and transient analyses lies in the loading and solution process [9]. This paper is the continuation of the previous work [10] that was submitted to this journal where the steady-state thermal analysis were performed on a similar model. However, in a previous study, the focused only concentrated on heat conduction and heat convection without the effect of time-steps control. Thus, the main objective of this study is to investigate the transient thermal analysis of the convection or cooling process of the aluminum alloy circular plate with respect to time when the hot plate is assumed to be heated at a certain temperature. The transient thermal analysis will provide a base for estimating the time required for the aluminum alloy circular plate to cool down to room temperature without the influence of any external force to aid the cooling process.

2. METHODOLOGY

2.1 Geometry Modeling

The main steps in doing element analysis of a particular model consist of modeling the geometry, meshing, applying the boundary conditions, and evaluating results [11]. In this study, all of those steps were performed by using the ANSYS Release 16.2 software. In this transient thermal analysis, the heater block model used and material engineering data are similar to that of our previous study [10]. The solid model created in ANSYS possesses structural steel material properties for both heater block and heater rod while for the circular plate, aluminum alloy material properties were applied to represent piston inside the IC engine, as the aluminum alloy is the common material used to manufacture pistons [12], [13]. For this particular study, the focusing solely on the aluminum alloy circular plate model. This is to reduce the computational times for the ANSYS software to evaluate the analysis results as well as to minimize errors in the analysis. The material properties of the aluminum alloy circular plate are shown in Table 1 and the heater block model used in our previous work [10] is illustrated in

Figure **1**. For this study, the schematic drawing of our aluminum alloy circular plate is shown in Figure **2**.

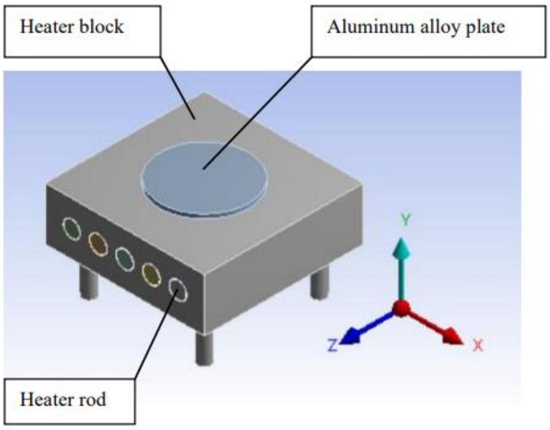


Figure 1. Heater block model.

Table 1 Properties of Aluminum Alloy (Al-Si) [13]

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

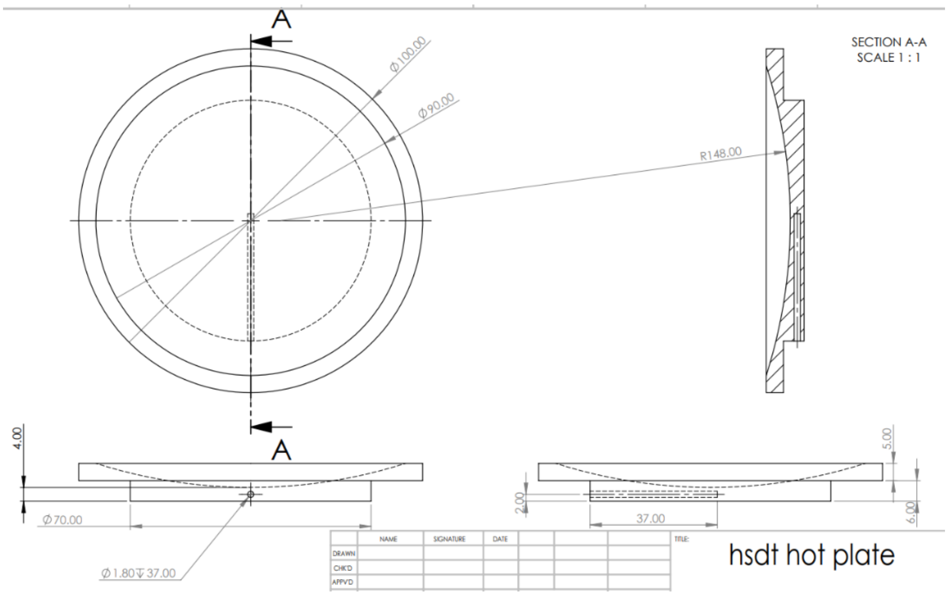


Figure 2. Aluminum alloy circular plate schematic drawing.

2.2 Meshing

To obtain a good computational result in ANSYS, the meshing quality of a designed model should be prioritized as the result of a particular analysis is lying on the quality of the meshing [14]. A good quality meshing usually consists of a smaller size of the mesh elements. However, more complex mesh elements can deliver high accuracy results but often require a longer response time to compute the solution analysis[15]–[17]. Furthermore, a transient thermal analysis will consume more time to be solved compared to that of steady-state thermal because of the time-step features. Thus, to reduce the simulation calculation time, a particular model can be reduced or simplified by focusing the analysis only on the important parts. In this study, the mesh setting used is the CFD Physic Preference, which has an improved mesh quality compared to the default Mechanical Physic Preference mesh setting. However, instead of using the whole heater block model, we used only the aluminum alloy circular plate model to get a more precise analysis. Furthermore, the simplified model could reduce the time required for the ANSYS software to compute the analysis results. The final mesh of the hot plate model comprises 21890 nodes and 114612 elements. In addition, a better thermal distribution analysis can be acquired as the mesh elements size was significantly minimized as can be seen in Figure **3**.

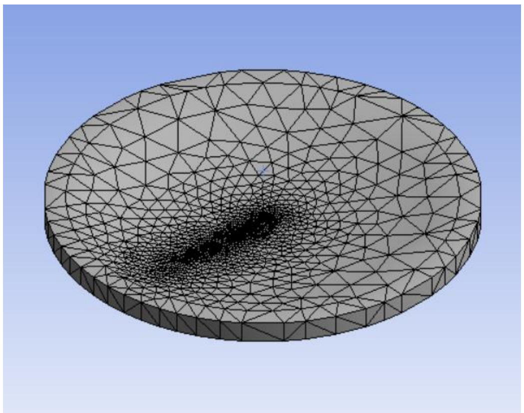


Figure 3. Final meshing generation using CFD Physic Preference.

2.3 Boundary Conditions and Computational Details

Compared to the boundary conditions and computational details in steady-state thermal analysis, a more complicated analysis setting is involved in the transient thermal analysis as it involves time dependence which is represented by the time-steps control feature in the ANSYS software. Furthermore, the computational times will be longer as more features are involved in the analysis. As long as the appropriate boundary conditions are set, the outcomes of the analysis can be used as a significant reference when creating a particular model in real life. In this study, the thermal load is defined as the initial temperature of the hot plate. It is assumed that the hot plate is already heated to the required temperature (100°C, 200°C, 300°C, 400°C, 500°C, 600°C). Convection properties are applied on the top and side parts of the aluminum alloy circular plate as presented in Figure **4**.

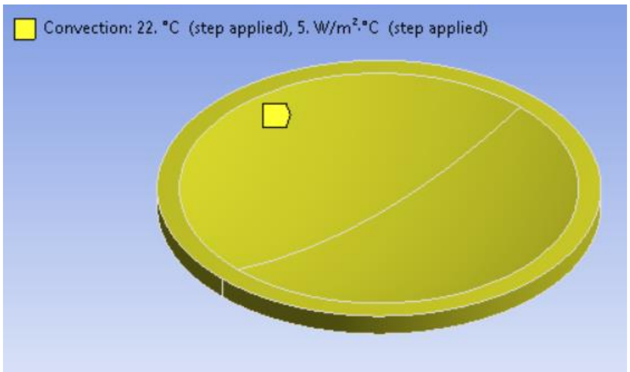


Figure 4. Convection condition applied on the aluminum alloy circular plate.

3. RESULTS AND DISCUSSION

3.1 Thermal Analysis

This transient thermal analysis is conducted to determine the time required for the aluminum alloy to cool down to room temperature after being heated at a certain temperature level. From the analysis, it can be observed that the highest temperature generated on the aluminum alloy circular plate is on the center part of the hot plate model. As shown in Figure **5**, after the hot plate reached room temperature, the cross-section of the hot plate model indicates that the highest temperature distribution is located on the bottom part of the solid model. In addition, the bottom part of the aluminum alloy circular plate also requires the longest time to cool down to room temperature after being heated. As can be seen in Figure **6** (a)-(f), the thermal distribution behavior is identical for all sets of initial temperatures applied on the hot plate model, which can be observed on the generated temperature color contour. The maximum temperature point which is indicated by the red color contour occurred on the center part of the hot plate. On the other hand, the minimum temperature point which is illustrated by the blue color contour occurred on the side part of the hot plate. Therefore, in the real deposits investigation experiment, it is important to make sure that the fuel is dropped exactly at the center part of the aluminum alloy circular plate as it is the location where maximum temperature will be generated.

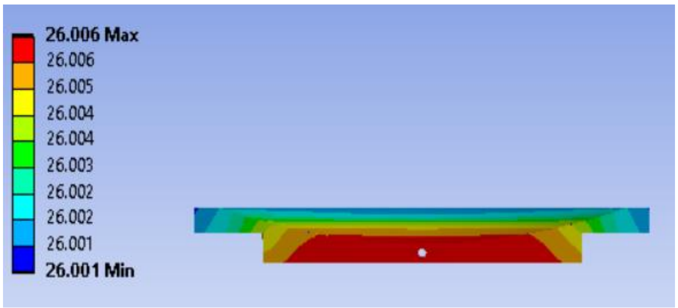
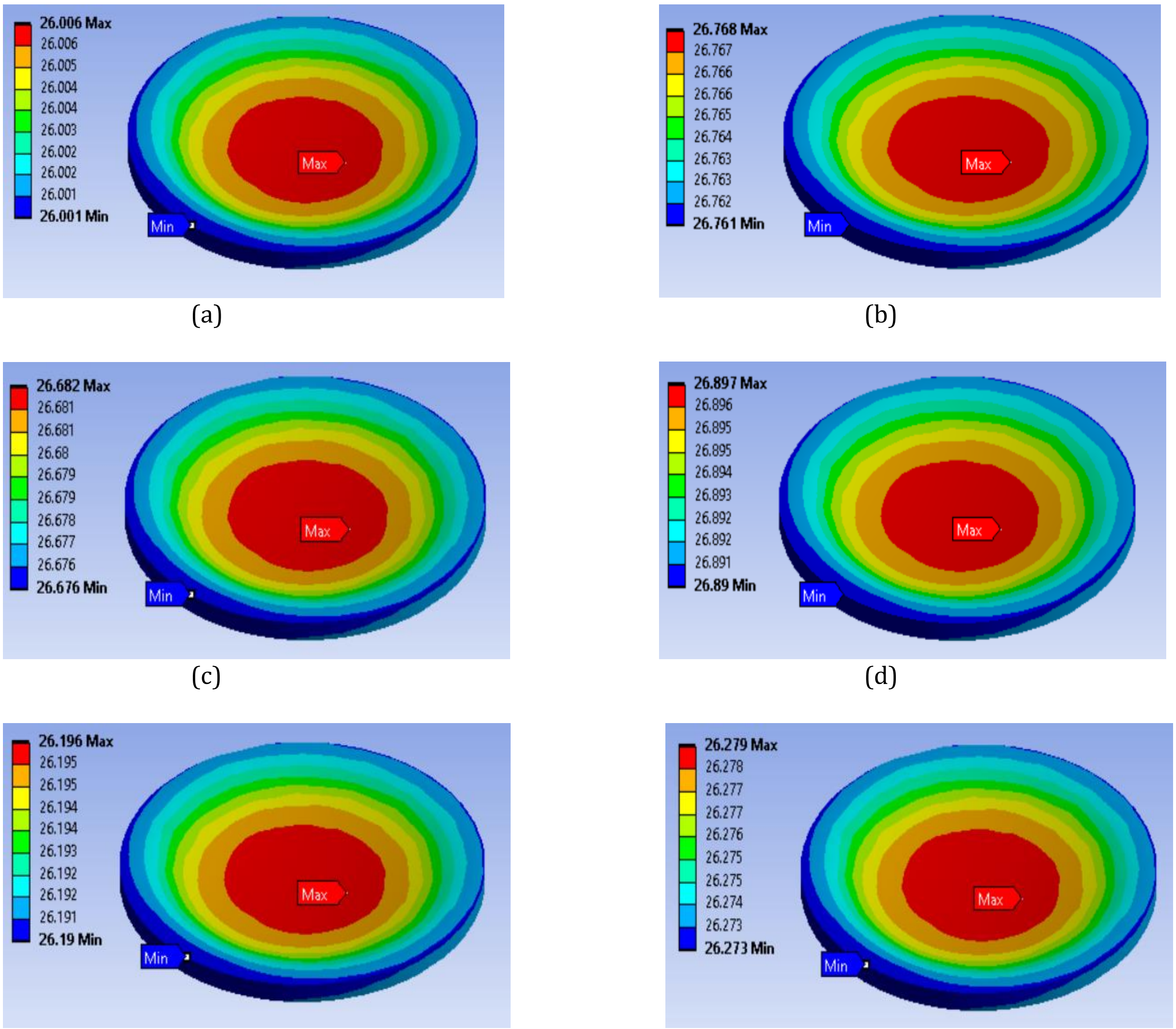


Figure 5. Cross-section of the aluminum alloy circular plate.



(a) (b)

(c) (d)

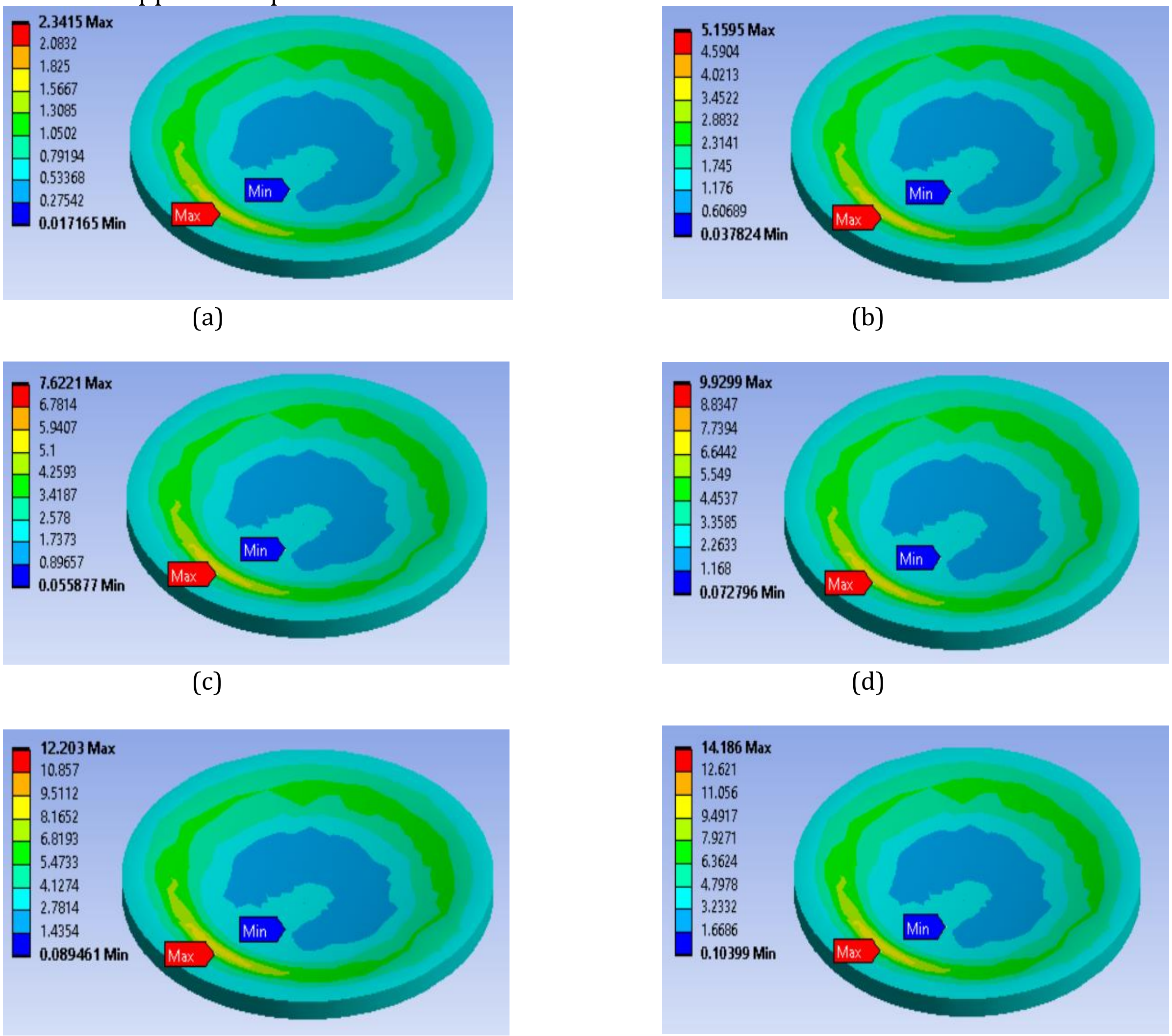
(e) (f)

Figure 6. Maximum and minimum temperature point at room temperature for applied temperature, (a) 100°C, (b) 200°C, (c) 300°C, (d) 400°C, (e) 500°C, (f) 600°C.

3.2 Total Heat Flux Analysis

A higher magnitude of total heat flux indicates that the rate of heat transfer occurring in an element is higher [18]. As can be observed in Figure 7 (a)-(f), the highest rate of heat transfer possessed by the aluminum alloy circular plate after it cooled down to room temperature is indicated by the light yellow color contour area.

Furthermore, the analysis also shows that the total heat flux generated on the hot plate increased when the applied temperature is increased.



(a) (b)

(c) (d)

(e) (f)

Figure 7. Maximum and minimum total heat flux point at room temperature for applied temperature, (a) 100°C, (b) 200°C, (c) 300°C, (d) 400°C, (e) 500°C, (f) 600°C.

3.3 Convection/Cooling Down Process

Figure 8 shows that when a 100°C of temperature is applied as the initial temperature of the aluminum alloy circular plate, it will take 7000 seconds which is equivalent to 117 minutes for the hot plate to cool down to room temperature without the mean of any external forces such as wind and liquid. As the applied temperature is increased to 200°C, 300°C, 400°C, 500°C, and 600°C, the duration of the convection process to reach room temperature also increase to 8500 seconds (142 minutes), 9500 seconds (158 minutes), 10500 seconds (175 minutes), 10800 seconds (180 minutes), and 11100 seconds (185 minutes) respectively as can be seen in **Figure 9**, **Figure 10**, **Figure 11**, **Figure 12**, and **Figure 13**. Furthermore, the time taken also indicate that the hot plate reached room temperature at 25°C-27°C range as shown in **Table 2**. In the actual deposition test experiment, when the hot plate reached room temperature, it will be detached from the heater block. In this way, it will be able to handle the cooled hot plate in a much safer manner to measure the weight and take a photograph of the generated deposits.

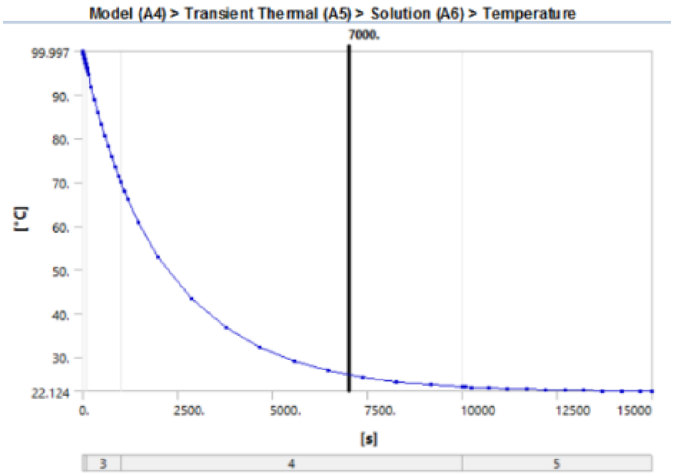


Figure 8. Time of cooling down to room temperature for 100°C hot plate.

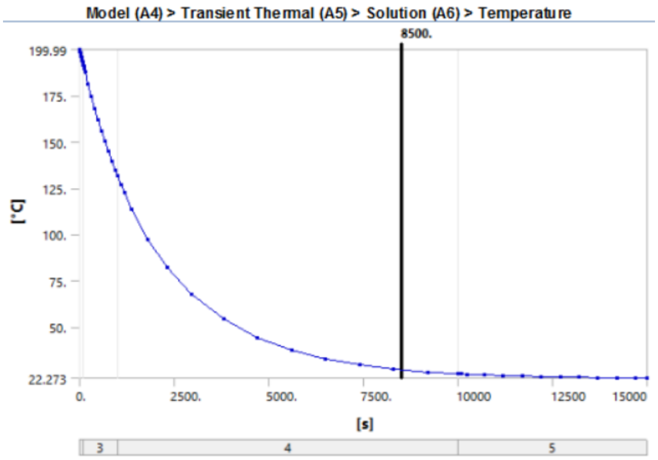


Figure 9. Time of cooling down to room temperature for 200°C hot plate.

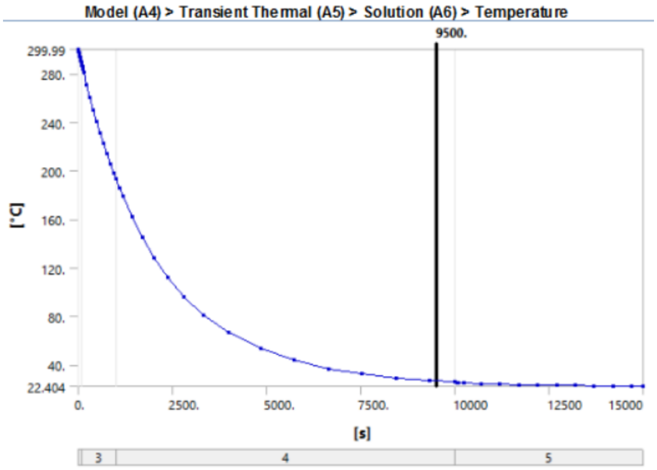


Figure 10. Time of cooling down to room temperature for 300°C hot plate.

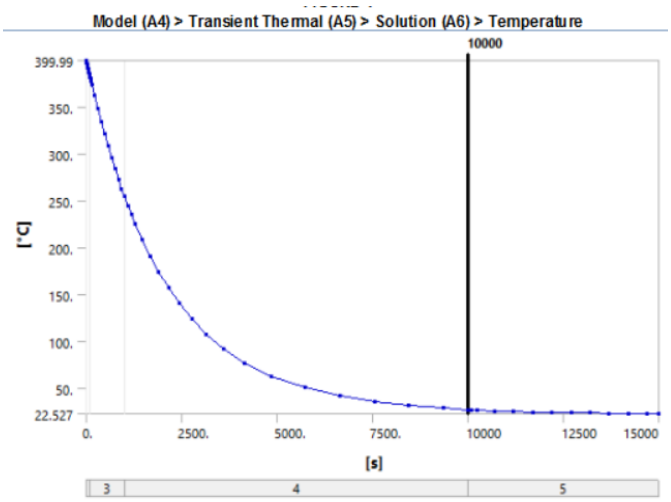


Figure 11. Time of cooling down to room temperature for 400°C hot plate.

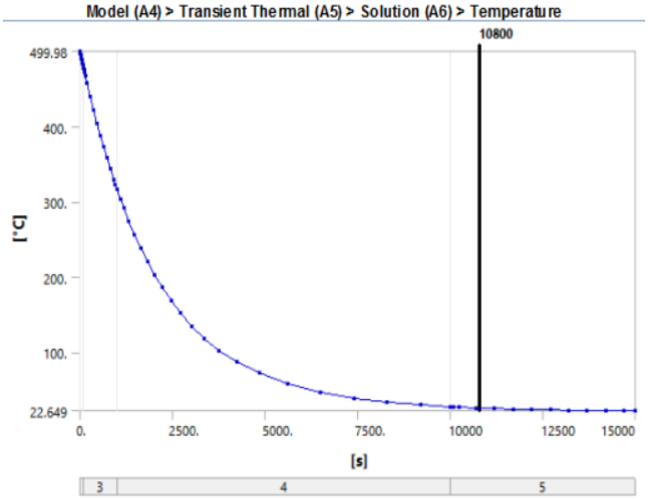


Figure 12. Time of cooling down to room temperature for 500°C hot plate.

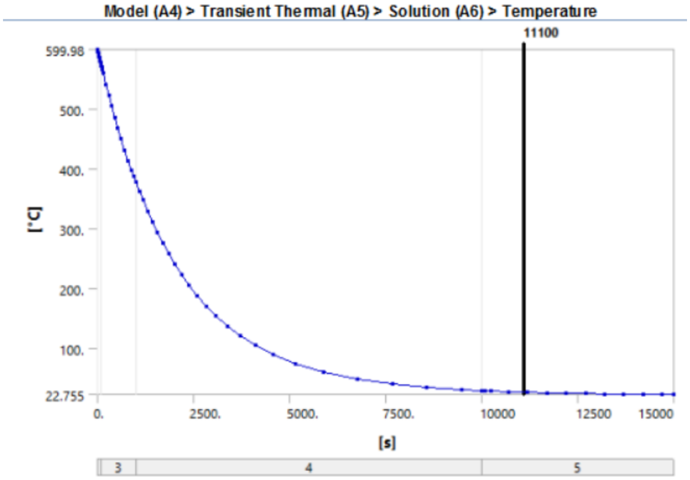


Figure 13. Time of cooling down to room temperature for 600°C hot plate.

Table 2 Time of Cooling Down to Room Temperature for Hot Plate

Applied Temperature (°C)	Time to Reach Room Temperature (s)	Average Temperature (°C)
100	7000	26.006
200	8500	26.767
300	9500	26.681
400	10500	26.896
500	10800	26.196
600	11100	26.278

4. CONCLUSION

The transient thermal analysis study on the convection process for aluminum alloy circular plate was successfully conducted by using ANSYS Release 16.2 software. From this work, it can be concluded that the highest temperature generated is at the center of the hot plate. The center area of the hot plate also needs a longer time to cool down to room temperature compared to other parts and this is caused by the thicker dimension on the bottom of the plate. For the time taken for the aluminum alloy circular plate to cool down to room temperature after being heated to a certain temperature, the results show that the convection process duration is longer for a hot plate with a higher internal temperature. However, the maximum total heat flux generated after the hot plate reached room temperature is different regarding the temperature applied on the analysis, which is higher maximum total heat flux for higher applied temperature and vice versa. Thus, this study is able to provide the data that could be used in estimating the time taken for an aluminum alloy circular plate to cool down to room temperature before it is safe to be handled in the actual deposit investigation using the HSDT method.

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 support.

REFERENCES

[1] 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 Research Journal. **3**, 4 (2016) pp.521–527.

[2] 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,” Applied Energy. **86**, 10 (2009) pp.2107–2113.

[3] 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,” Renewable Energy. **86**, (2015) pp.59–67.

[4] 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,” Applied Energy. **203**, (2017) pp.582–593.

[5] Y. M. Arifin & M. Arai, “The effect of hot surface temperature on diesel fuel deposit

formation," Fuel. **89**, 5 (2010) pp.934–942.

[6] V. V. Pham, "Research and design an experimental model for the determination of deposits formation mechanism in the combustion chamber," International Journal on Advanced Science, Engineering and Information Technology. **9**, 2 (2019) pp.656–663.

[7] M. T. Suryantoro *et al.*, "Effect of temperature to diesel (B0) and biodiesel (B100) fuel deposits forming," AIP Conference Proceedings. **2062**, January (2019) pp.1–6.

[8] C. Zhang, Q. Guo, J. Sun, & C. Liu, "Comparative analysis for heat transfer performance of heat exchanger single tube model with and without plug-in," Chemical Engineering Transactions. **66**, (2018) pp.301–306.

[9] ANSYS Inc, "ANSYS Mechanical Heat Transfer Module 06 : Transient Thermal Analysis," (2019) pp.1–31.

[10] F. Jikol, M. Z. Akop, Y. M. Arifin, M. A. Salim, & S. G. Herawan, "A Study of Steady-State Thermal Distribution on Circular Plate Using ANSYS," International Journal of Nanoelectrics and Materials. **14**, August (2021) pp.479–488.

[11] A. Bhat, B. Pal, & D. Dandotiya, "Structural Analysis of A Two-Wheeler Disc Brake," IOP Conference Series: Materials Science and Engineering. **1013**, 1 (2021) pp.1-8.

[12] 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 Electrical Insulation Conference, EIC 2020. (2020) pp. 163–166.

[13] 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," International Journal of Mechanical Engineering and Technology. **8**, 8 (2017) pp.988–997.

[14] T. Y. Yusaf, M. M. Noor, & A. P. Wandel, "Detail Guide for Cfd on the Simulation of Biogas Combustion In Bluff-Body Mild Burner," International Conference on Mechanical Engineering Research. July (2013) pp.1–3.

[15] L. T. Wei *et al.*, "A Simulation Study on Performance of Photovoltaic Thermal Water (PVTW) Collector Under Different Loading of Mass Flow Rate Using ANSYS Fluent," International Journal of Nanoelectrics and Materials. **14**, August (2021) pp.467-478.

[16] 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," Arabian Journal for Science and Engineering. **39**, 10 (2014) pp.7229–7239.

[17] A. Belhocine & O. I. Abdullah, "A thermomechanical model for the analysis of disc brake using the finite element method in frictional contact," Journal of Thermal Stresses. **43**, 3 (2020) pp.305–320.

[18] B. Kushwah & P. S. Dhakar, "Temperature Distribution Analysis of Cylinder Block with Fins for Various Materials and Different Geometry," International Journal Of Research And Analytical Reviews (IJRAR). **6**, 2 (2019) pp.552–558.