Universiti Malaysia PAHANG
Engineering • Technology • Creativity

JOURNAL OF MODERN MANUFACTURING SYSTEMS AND TECHNOLOGY

Homepage: http://journal.ump.edu.my/jmmst

FINITE ELEMENT ANALYSIS OF THE HEAT TRANSFER IN A PISTON

A. Muhammad[1*] and I.H. Shanono[2]

*Correspondence
ayshermuhd@gmail.com
[1] Faculty of Manufacturing and Mechatronic Technology Engineering, Universiti Malaysia Pahang, 26600, Pekan, Pahang, Malaysia

[2] Faculty of Electrical Engineering, Bayero University Kano, Nigeria.

Articles Info:
Received **15 January 2020**
Received in revised form **26 February 2020**
Accepted **27 March 2020**
Available online **31 March 2020**

Keywords:
ANSYS; Piston; Transient; SolidWorks.

ABSTRACT

The gas expansion process that takes place in a piston cylinder assembly have been used in numerous applications. However, the time-dependent process of heat transfer is still not fully apprehended as the expansion processes are complex and difficult due to the unsteady property of the turbulent flow process. Internal combustion Engines(ICE) designs are conducted with the aim of achieving higher efficiency in the thermal characteristics. To optimize these designs, numerical simulations are conducted. However, modelling of the process in terms of heat transfer and combustion is complex and challenging. For a designer to understand, calculate and quantify the thermal stresses and heat losses at different sections of the structure, understanding the piston-cylinder wall is needed. This study carried out a numerical simulations based on Finite Element Method (FEM) to investigatethe stresses in the piston, and temperature after loading. Appropriate boundary conditions were set on different surfaces for FE model. The study includes the effects of the thermal conductivity of the material of piston, cylinder wall, and connecting rod. Results show the maximum Von-misses stress occurs on the piston head with a value of 3486. 1MPa. The maximum temperature of the piston head and cylinder wall stands at 68.252 and 42.704 degree Celsius respectively. The heat generated in a piston affects the performance as well as fatigue strength of an IC engine. Accurate analysis of the fatigue strength can be possible if the temperature of the piston is determined. The thermal conditions were determined using FEM and behaviour of the piston was explained to some details with the most critical areas determined to be at the pin head

INTRODUCTION

Design of heat engines and thermos-fluid tools have increased the demand for understanding of thermodynamics for the breakthrough of power and green technologies. The process of compression or expansion of a gas in a piston-cylinder system is typical in numerous applications. Nonetheless, the irreversible procedure is not well understood [1]-[6]. The process of gas expansion in a piston-cylinder assembly prevails in lots of applications. The expansion process itself is incredibly intricate. These expansions are defined by turbulent flow, and the procedure is inherently unstable. The complexity makes the characterization of the heat transfer challenges, particularly when time-dependent transfer rates throughout the development are required [7].

For the correct performance of the IC diesel engine, exact piston temperature level circulation was required since piston temperature level has an essential influence on ignition process of the engine, rate, ignition time delay, and efficiency. Understanding heat transfer becomes vital in order to comprehend such systems [8], [9]. Engine components need to be created to hold up against heats, so it works to identify the temperature level circulation of the liquid exposed surfaces, which is experimentally made complex as well as invasive [10], [11]. It is essential in premixed air-fuel fuel engines to figure out hot areas that might cause very early ignition and auto-ignition triggering in irregular combustion.

Simulations, on the other hand, provide inexpensive means for calculating the temperature in the combustion chamber and the heat transfer to the engine wall surfaces. There are nevertheless some problems that may be encountered, as combustion has to be designed expertly, and the transfer models from gas to wall needs robust wall designs, which increases the difficulty of this sort of simulations. In the 1980s, this was accomplished with little success because of the inadequate attributes of ceramic insulation materials used to shield the engine walls [12]. Due to the recent studies in new insulating material, the technology has boosted because new materials of reduced heat capacity and thermal conductivity are being proposed [13]. These studies suggest that by finishing the piston surface area, the instantaneous temperature on it transforms in phase with the gas temperature which further minimises warm losses.

In order to predict discharges at different procedure conditions without needing to spend much, scientists have tried to simulate the combustion procedure in engines. Due to the complexity of the burning procedure, several designs have been developed [14]. When emissions are not the focus of a study, it might not be necessary to model the combustion with high accuracy. It was understood that Internal Combustion Engine ICE) efficiency as well as gas performance may be improved if heat losses can be decreased [15]

This paper presents the study on a piston-cylinder system in order to understand the fundamentals of the irreversible process through Finite Element Method in ANSYS software.

METHODOLOGY

2.1 Finite Element Method

Finite Element Analysis (FEA) is one of the most regularly utilised computational method used for testing and changing layout of structures within specific design restriction. It entails diving into small sections described as 'elements' with mathematically specified characteristics for static and dynamic evaluation. With the use of computers, matrix algebra was used to solve complicated structures. The input data include the design constraints on the meshed structures with physical properties of the material [16]-[18] Multiphysics, general-purpose finite element analysis software. A series of CAE products were established by ANSYS Inc, but still, it is best understood for ANSYS Mechanical & ANSYS Multiphysics. The scholastic variations of these products are described as ANSYS Academic Research, ANSYS Academic Teaching Advanced, and so on [19].

Sub-dividing a complicated structure into smaller sized elements is the critical suggestion in FEA. It is one of the practical CAE computational tools as engineering structures can be design based upon the layout criteria within a short period [20]. Computer-aided design software is used to design structures that cannot be done manually. It also provides two-way access to computer-aided style software application Figure 1 below depicts the FEM procedure and steps used to solve engineering problems.

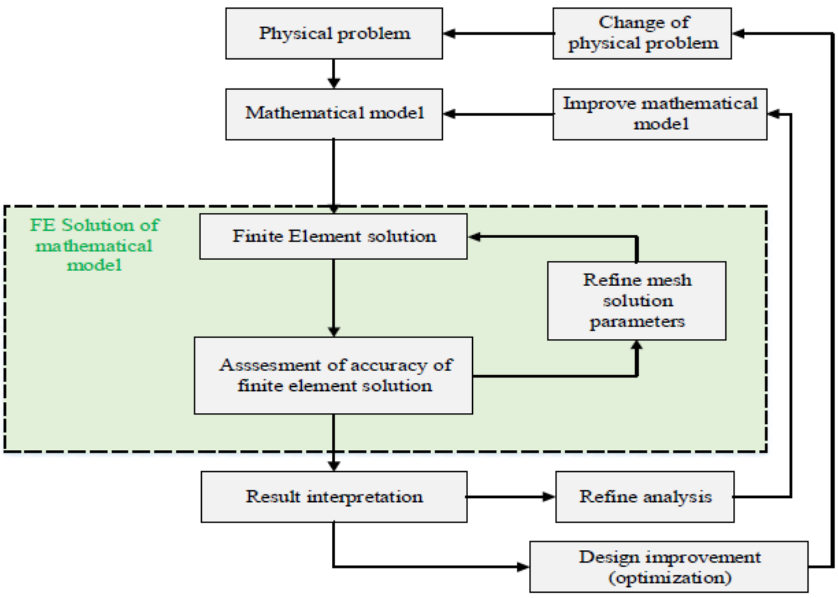

Figure 1 Finite Element Analysis Process flow.

The geometrical structure of the piston was established based upon the geometry of the actual design. Figure 2 shows the CAD structure exported into the ANSYS workbench. Due to the complexity of the structure, meshing sizes were different in particular sections of the structure; an advanced face sizing of 2mm was used on the cylinder, and piston head whereas the other sections were set to a default tetrahedron meshing size.

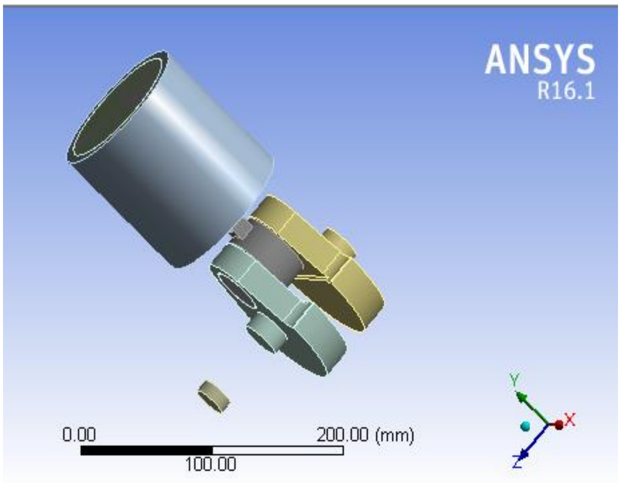

Figure 2 Imported piston-cylinder assembly in ANSYS

After meshing of the structure, the design constraints, a primary step required in an FE analysis are set in a way that represents the piston-cylinder actual real situation [21]. To determine how the contact bodies are going to move relative to each other, a no separation contact is used for the connection contact for frictionless sliding of the faces without separation of the faces in contact. The transient analysis is carried out in the Mechanical APDL solver to determine the transient response of the piston under the design constraints. For the contact setting, the ANSYS software automatically recognises the contact area between the bodies. Using three revolute joint type: (Ground to Crank, crank to Connecting rod and Connecting rod to the piston), a translational joint type (piston to piston head) and to orient and assemble the bodies, ground to the body(piston) fixed joint is added as shown in Figure 3.

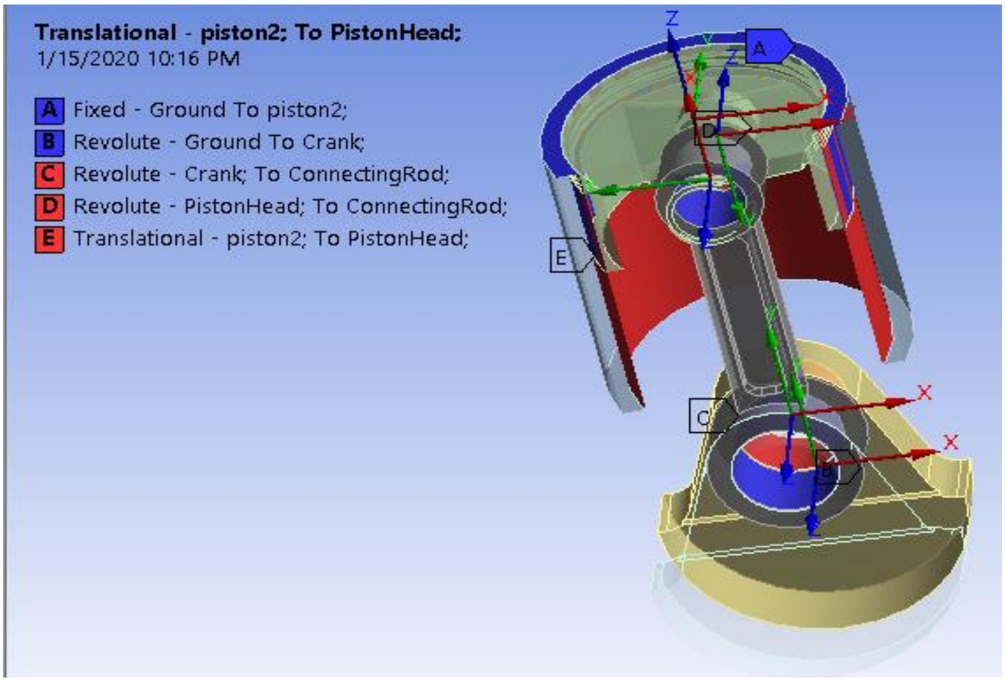

Figure 3 Joint connection

RESULT AND DISCUSSION

Failures of engineering structures make analysis to become very significant to ascertain the safety of the structure because a failure of the whole or part of the structure leads to high risk of life and financial loss [22]. This section presents the analysis result performed. Figure 4 shows the Von-misses stress property of the material having maximum and minimum values of 3486.1MPa and 387.34Mpa, respectively. A value of 1889.1 MPa pressure value was obtained as seen in Figure 5

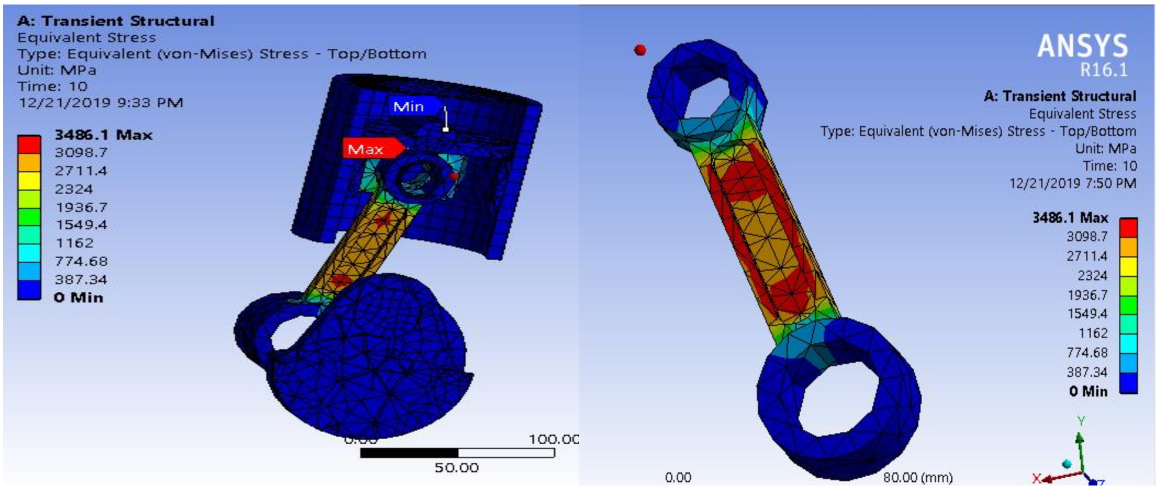

Figure 4 Von-mises Equivalent stress

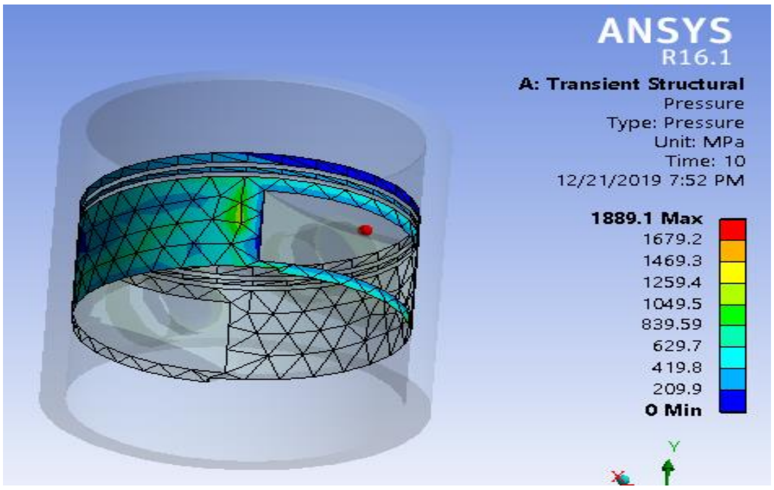

Figure 5: Pressure

The Joint Probe in the *X, Y* and *Z* plane, as shown in figure 6, displays the resultant reaction force for the variation to time. The force at Z-axis has a maximum experienced at 5.2 seconds. The resultant force in the Y-axis remains constant after 1 second throughout the period. While the x-axis takes its peak at approximately 2.5 seconds with a force value of 1.0246E+5N. Hence, the total resultant force is towards the x-axis with a slightly higher magnitude than that of the Y-axis.

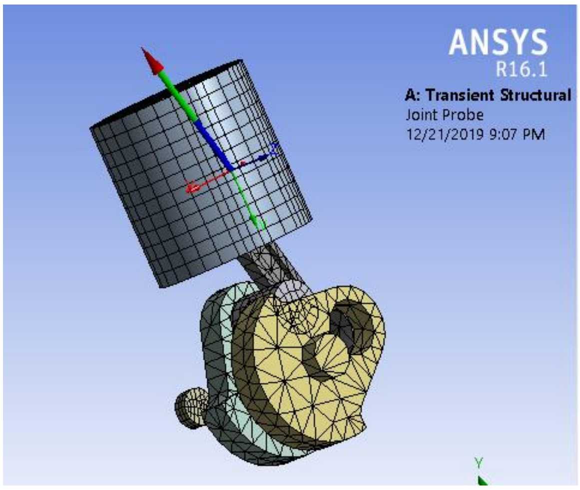

Figure 6 Joint Probe

Result of the transient temperature in the cylinder body and piston is displayed in Figure 7. the piston head as clearly depicted graphically in figure 8 has higher temperature value when compared to the cylinder

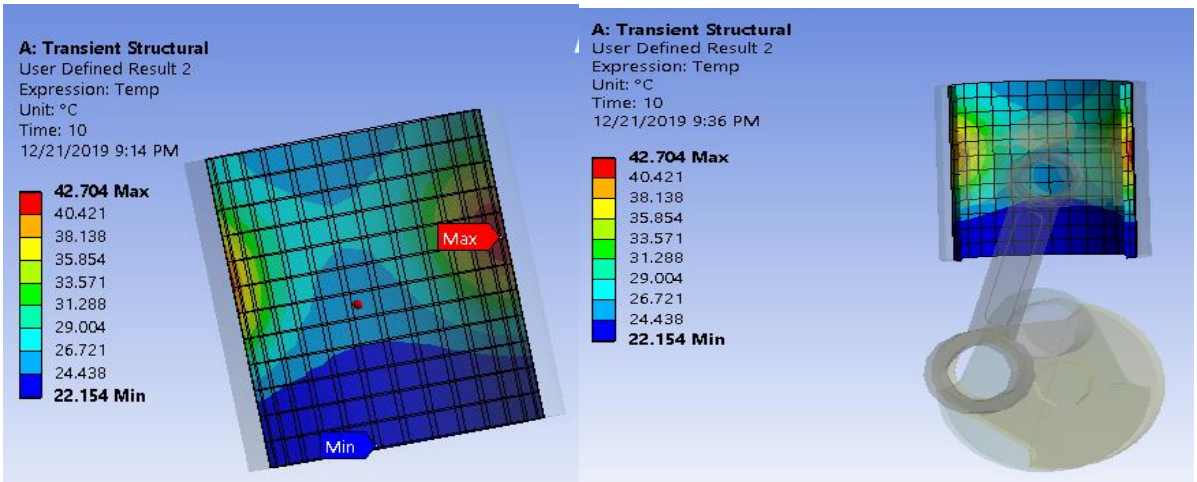

Figure 7 cylinder temperature

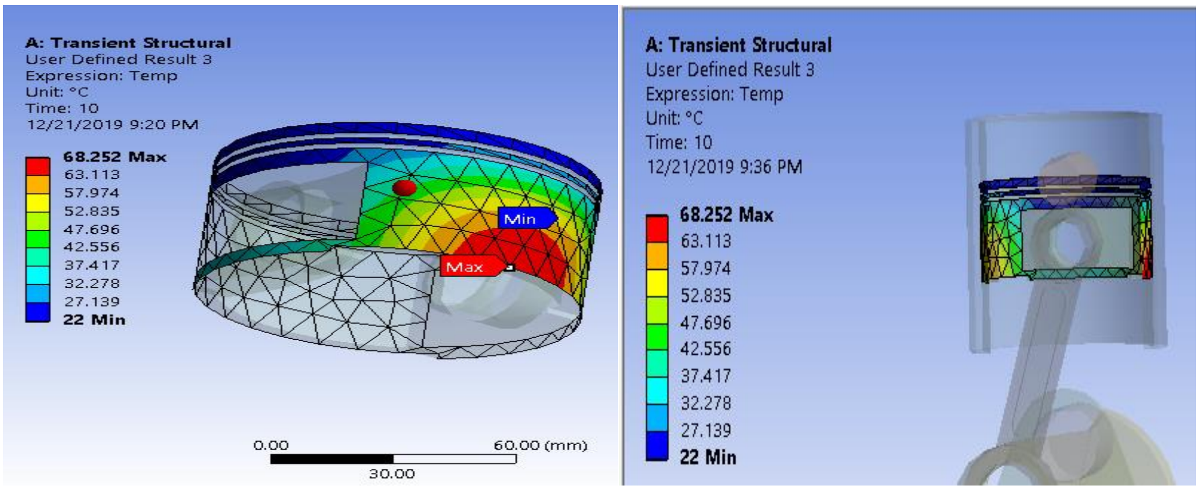

Figure 8 Piston head temperature

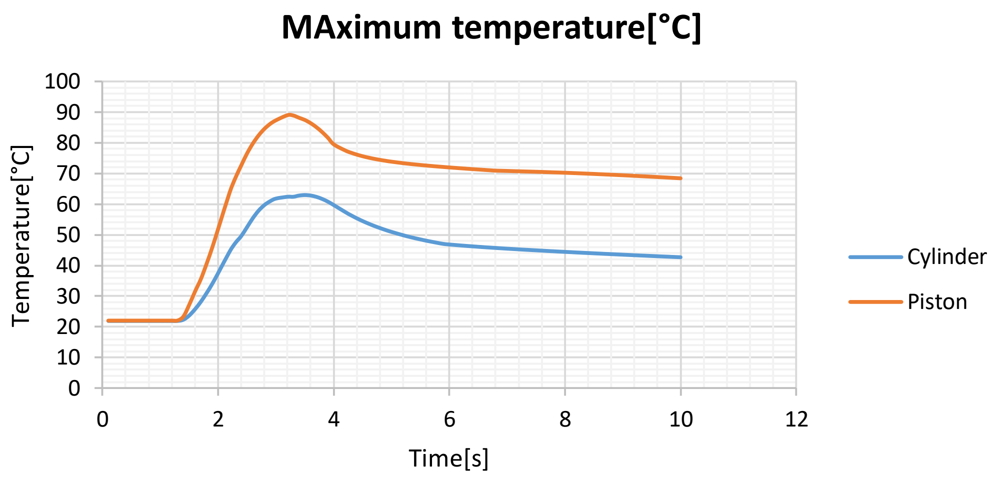

Figure 9 Cylinder-piston temperature comparison

Further repeated refinement of the FEA structure into much smaller elements results in the convergence of the solution to an exact solution of the mathematical problem [23]. In this analysis, the convergence criteria are based on force. Convergence was obtained for the static analysis solution after a maximum number of 10 seconds as shown in figure 10

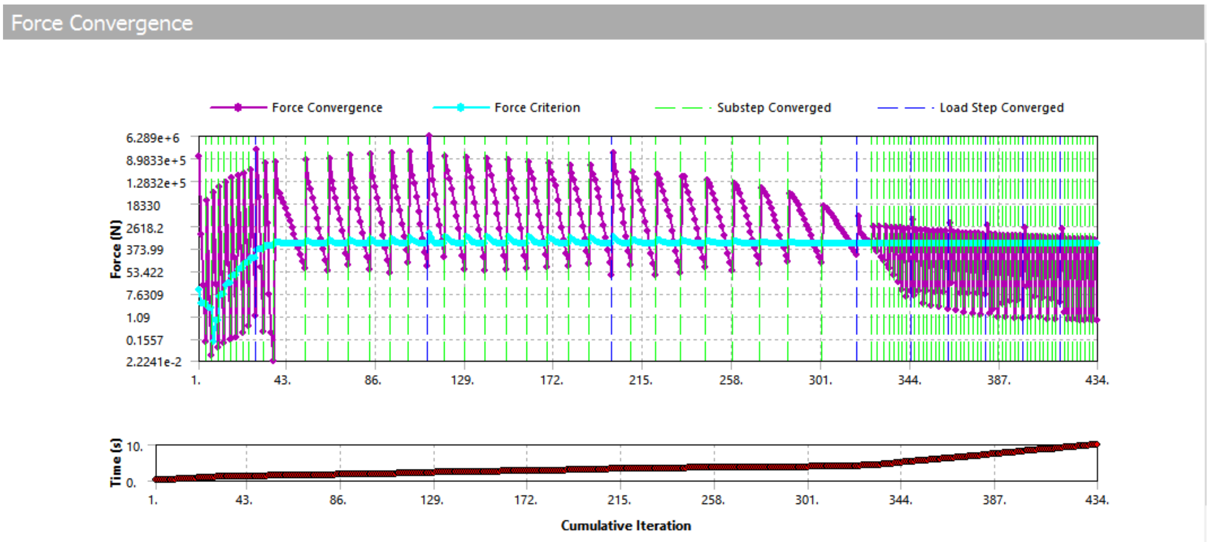

Figure 10 Convergence plot

CONCLUSIONS

In this paper, transient analysis of a piston-cylinder assembly using Finite Element Method (FEM) in ANSYS workbench is carried out in order to observe its stress and temperature response under the design loading conditions. The piston assembly structure was designed using SolidWorks software, saved as an IGES file and then imported into the ANSYS software. From the transient analysis result of the piston, the maximum Von-misses stress occurs on the piston head with a value of 3486.1MPa. The maximum temperature of the piston head and cylinder wall stands at 68.252 and 42.704 degree Celsius respectively. The red colour indicates this at the base-tip of the head while the stress distribution level across the connecting rod two ends joint appeared to have no overstressed visible region.

This paper provides an introduction to FEM for engineering systems with similar models and gives away for future research for structural optimisation of the structural design.

In order to accurately determine the thermal conditions of an IC engine, many details are required. A more comprehensive investigation would certainly be extremely beneficial. As a future recommendation, combining simulations and experimental could be used to get a more accurate understanding of the thermal operating conditions of the piston.

REFERENCES

[1] Joaquim, A., Fereira, J. M., Soares, A.A. (2009). When an adiabatic irreversible expansion or compression becomes reversible. European Journal of Physics, 30, 487–495.
[2] Eric, A. G., Norman, C. C. (2002). First law of thermodynamics; irreversible and reversible processes. Journal of Chemical Education 79,193–199.
[3] Plotnikova, L.V., Zhilkin, B.P. (2017). The gas-dynamic unsteadiness effects on heat transfer in the intake and exhaust systems of piston internal combustion engines. International Journal of Heat and Mass Transfer, 115B, 1182-1191.
[4] Stanislaw, S. (2008). Variational setting for reversible and irreversible fluids with heat flow. International Journal of Heat and Mass Transfer, 51(11–12), 665-2675.
[5] Yingru, Z., Jincan, C. (2007). An irreversible heat engine model including three typical thermodynamic cycles and their optimum performance analysis. International Journal of Thermal Sciences, 46(6), 605-613.
[6] D. N. John. (2016). The Impossible Process: Thermodynamic Reversibility. Modern Physics, 55, 43–61.
[7] Matthew, J. R. (2014). Heat Transfer During the Piston-Cylinder Expansion of a Gas. Oregon State University, US.
[8] John, B. H. (1999). Internal Combustion Engines Fundamentals, NY, USA: McGraw-Hill New York.
[9] Richard, S. (1999). Introduction to Internal Combustion Engines, Macmillan Press.
[10] Iqbal, O., Arora, K., and Sanka, M. (2014). Thermal Map of an IC Engine via Conjugate Heat Transfer: Validation and Test Data Correlation. SAE Int. J. Engines, 7(1), 366-374.
[11] Seokhwan, C., Choongsik, B. (2008). Design of a heat exchanger to reduce the exhaust temperature in a spark-ignition engine. International Journal of Thermal Sciences, 47 (4), 468-478.
[12] Woschni, G., Spindler, W., and Kolesa, K. (1987). Heat Insulation of Combustion Chamber Walls — A Measure to Decrease the Fuel Consumption of I.C. Engines. SAE Technical Paper 870339, Germany.
[13] Kosaka, H., Wakisaka, Y., Nomura, Y., Hotta, Y. et al. (2013). Concept of “Temperature Swing Heat Insulation” in Combustion Chamber Walls, and Appropriate Thermo-Physical Properties for Heat Insulation Coat. SAE Int. J. Engines, 6 (1), 142-149.
[14] Mohammadia, A., Yaghoubi, M. (2010). Estimation of instantaneous local heat transfer coefficient in spark-ignition engines. International Journal of Thermal Sciences. 49(7), 1309-1317.
[15] Som, S., Longman, D., Aithal, S., Bair, R. et al. (2013). A Numerical Investigation on Scalability and Grid Convergence of Internal Combustion Engine Simulations. SAE Technical Paper, US.
[16] Aisha, M., Ibrahim, I. H. (2019). Static Analysis and Optimization of a Connecting Rod. International Journal of Engineering Technology and Sciences (IJETS), 6(1), 24-40.
[17] Subodh, K. S., Saini, P. K., Samria, N. K. (2015). Experimental Thermal Analysis of Diesel Engine Piston and Cylinder Wall. Hindawi Publishing Corporation, 10.
[18] Aisha, M., Ibrahim, I. H. (2018). Strength analysis and structural optimisation of an l-shaped bracket. Nigerian Journal of Technological Research (NJTR), 13(2), 14-19
[19] Aisha, M., Mohammed, A. H. A., Ibrahim, I. H., (2020). ANSYS – A bibliometric study. Materials Today: Proceedings, 4,(2), 179-186. DOI: 10.1016/j.matpr.2020.01.192
[20] Aisha, M., Ibrahim, I. H. (2018). Weight Optimisation of a laboratory stool using ANSYS workbench. In 1st International Civil Engineering Conference (ICEC 2018), Minna.
[21] Aisha, M., Ibrahim, I. H. (2019). Transient Analysis and Optimization of A Knuckle Joint. KINETIK, 4,(2), 179-186.
[22] Aisha, M., Mohammed, A. H. A., Ibrahim, I. H. Fatigue and Harmonic Analysis of a Diesel Engine Crankshaft Using ANSYS. M. N. Osman Zahid et al. (Eds.): iMEC-APCOMS, 2019, LNME. 1–6, 2020.. DOI: 10.1007/978-981-15-0950-6_56
[23] Xiaolin, C., Yijun, L. (2014). Finite Element Modeling and Simulation with ANSYS Workbench, London NewYork: CRC Press.