Steady State Thermal Analysis of IC Engine Piston Head Surface using ANSYS Workbench

[1]Trilok Maithil, [2]Prof. Saumitra Sharma
M. Tech. Scholar, Department of Mechanical Engineering, Oriental College of Technology, Bhopal[1]
Assistant Professor, Department of Mechanical Engineering, Oriental College of Technology, Bhopal[2]

Abstract: The working gas pressure, temperature, and material functions of the piston were utilized in the study. Because damaged or non-working components are so expensive to replace and generally aren't readily accessible, the study was conducted to see whether the top face of the piston might be damaged or fractured because of temperature. Surfaces to design piston is geometry, while ANSYS 14.0 was used for FEM analysis to optimize thermal behaviors of the piston. For the piston's design, aluminium alloy and grey cast iron were utilized. In structural analysis, the piston's stress and displacement are calculated by applying pressure to it. We can evaluate if our planned piston is safe or not under actual load circumstances by looking at the analytical consequences. Temperatures are applied to the piston surface to evaluate the thermal flux and thermal temperature distribution.

Keywords: Internal Combustion (IC) Engines, Performance, Heat Transfer, Piston, ANSYS 14.0

I. INTRODUCTION

The increasing usage of cars is a major demand for automotive mechanics these days. The higher demand is due to improved performance and lower costs for this equipment. In the shortest possible period, R&D and test engineers should enhance key mechanisms to decrease launch time for new goods. “This requires knowledge of new know-how and rapid integration in new product development. A piston is a reciprocal motor. It is the reciprocal component that is controlled by a piston ring and gasproof. Its concept of an engine is to transmit power via a piston rod and connecting rod from increased gas in the cylinder to the crank shaft. In motor cylinders, compressors, and gas pumps, a piston is a component [1, 2]. The piston is constructed in a cylindrical shape with piston rings to make it gastight.

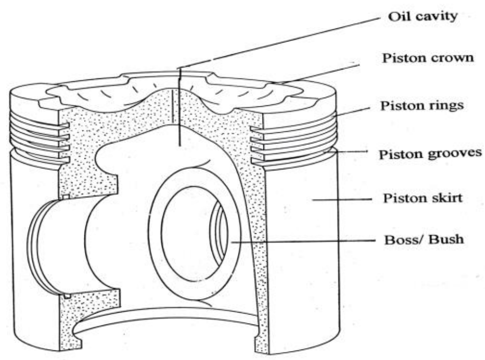

Figure 1: Design of Piston

In an engine, the power of the expanding gas in the cylinders is transmitted to the shaft through a piston rod [3]. When the fluid in the cylinder is pushed or expelled and the strength is passed from the crankshaft to the piston, the pump function is overridden. In certain engines, the piston also serves as a valve for holes on the cylindrical wall and uncovers them”.

Piston materials and design have been changed or enhanced throughout time starting many years ago and will continue until fuel cells are replaced, unique batteries, or anything else replaces the combustion engine [4, 5]. The "heart" of a motor may become a continuous evolving attempt as a result of the column.

II. FINITE ELEMENT METHOD

Finite element analysis is a method that splits a model or element into tiny finite components known as finite elements and then considers the original model or structure as the assembly of those factors connected with various numbers of joints known as nodal points or knots. Because the true fluctuation of subject variables such as displacement, stress, temperature, pressure, or speed over a continuum cannot be determined, a simple function may be used to estimate the change of ground variables within a restricted area. Interpolation models are evaluation functions that are defined based on the values of node field variables. The resolution of the field equations, which are usually in matrix equations, is used to establish the nodal standards for the changing field.

The approximation characteristics characterize the field variable during the assembly of components after the nodal values are known.

The general problems of continuum using the finite element technique typically follow a step-by-step approach.

The following is a step-by-step method for static structural applications:

Step 1:- Explanation of the Design Model (Domain). The partitioning of the final area structure into subdivisions or components is the most important step in the finite element process.

Step 2:- Interpolation of the appropriate kind was employed. Because the explanation of a complex structure (field variable) cannot be anticipated adequately under any particular load circumstances, we assume some appropriate result to estimate an unknown solution in a component. Certain convergence conditions must be met, and the expected outcome must be clear.

Step 3:- Start of the matrices (feature matrices) of the components stiffness and load vectors. The postulated version of the displacement should result in the rigidity matrix [K(e)] and the vector load P(e) of the element 'e' utilising both the balancing scenario and a suitable Variation Precept.

Step 4:- Assembly of element equations to reach equilibrium.

Since there are many finite components in the structure, the individual rigidity matrices and the charging vectors have to be properly constructed and the overall balance equation stated as

$$[K]\varphi = P \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (1)$$

When [K] is known as the assembled rigidity matrix, os is known as the nodal displacement vector and P is the vector or nodal pressure for the whole form.

Step 5:- Solve the system equation to get the nodal displacement values (subject variable). The traditional equilibrium equations must be modified to account for the problem's boundary conditions. Once the boundary conditions are considered, the equilibrium equations may be stated as,

$$[K]\varphi = P \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (2)$$

In the case of linear problems, the vector "may be very problem-free. However, if there are non-linear problems, the solution must be achieved in phases, which include modifying the stiffness matrix [K] and the weight vector P.

Step 6:- The tensions and stresses are computed in great detail. If required, the detail lines and stresses may be computed from the nodal displacements using the basic equations of stability or structural mechanics. The words in parentheses in the steps above impose the FEM step by step.

III. MODELING OF PISTON

Aluminium alloy and other grey cast iron are two metals that have been utilized in piston research. For thermal analysis, solids are Piston and ANSYS. Solidwork. To comprehend the true application of Finite Element Modeling, consider it as the FEA when the issue is really solved. The automobile sector has made extensive use of FEA. It is a common tool for design engineers to utilize throughout the product development process. Design engineers may utilize FEA to assess their concepts while they're still in the form of a SolidWork model.

3.1 Layout of ANSYS Workbench

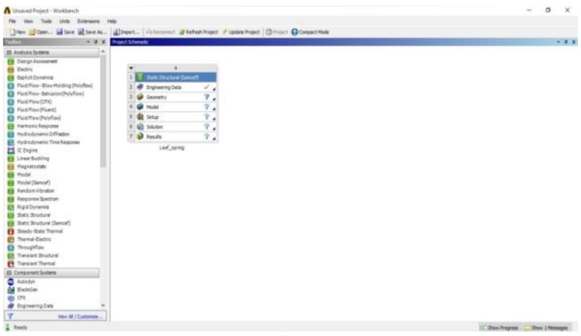

Figure 2: Layout of ANSYS Workbench

3.2 Model Generation

To begin, double-click on the Static Structural option to bring up a little window that we may rename as seen in figure 2. If we wish to change the material, we'll double-click on "Engineering Data." To draw the beam, double-click 'Geometry.' After double-clicking on "Geometry" in Figure 2, a second window called "Design Modeler" will appear, where we may first select the dimensions we want to work on, as seen in Figure 3.

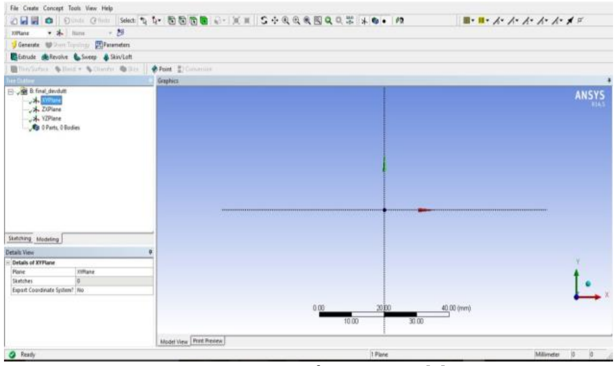

Figure 3: Layout of Design Modeler

3.3 Modeling of Piston

Detailed is an explanation of the modelling of the Piston made using Solidwork. The aim of finite element research is to rebuild the current engineering structures' mathematical behaviour. The model consists of all nodes, elements, material characteristics, actual constants, limits and other characteristics that define the physical system. The model is first created and then certain limits are put to the particular nodes, and finally the final analysis is carried out.

The piston is built in the machine design and in reference data books to the process and specifications.

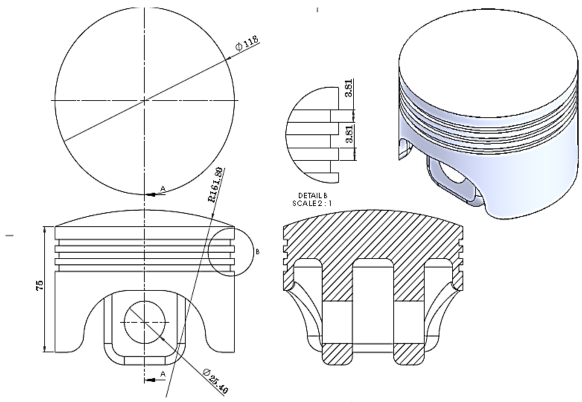

Figure 4: Drawings of the Piston

a) Analyzing the model in ANSYS

The model in Solidwork has been designed. IGS FILE was converted to the format of IGES. The design may be compatible with ANSYS software with this setup.

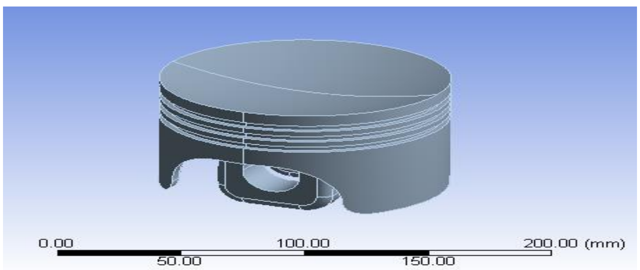

Figure 5: Model of Design 1

The analysis procedure starts after loading the design into ANSYS. When constructing a piston, two types of piston first type is Design 1, and the second kind is Design 2.

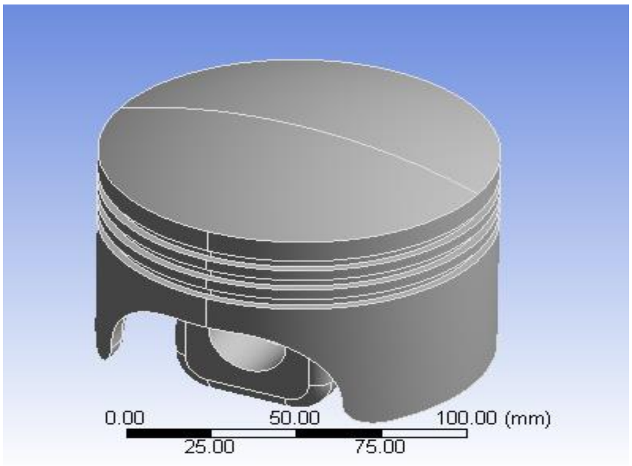

Figure 6: Model of Design 2

b) Meshing the model

The Piston model to be examined is mathematically split into a mesh of finite simple-form components. For each component, basic Polynomial Profile functions and nodal displacements are supposed to compute the displacement difference. Unknown nodal displacements are caused by equations for stresses and strains. The equations of balance are therefore constructed in a matrix that is programmable easily.

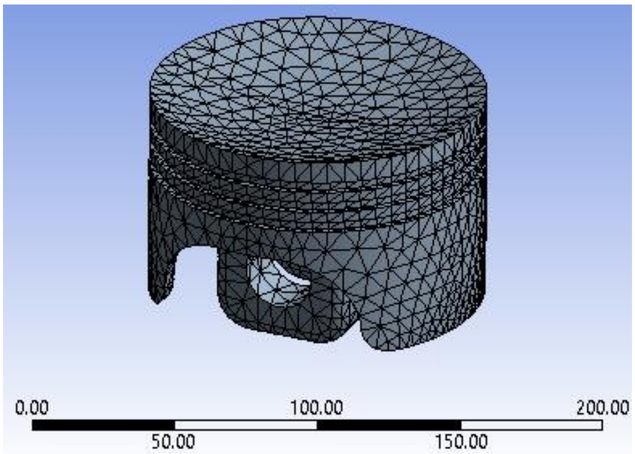

Figure 7: Meshed view of Design 1

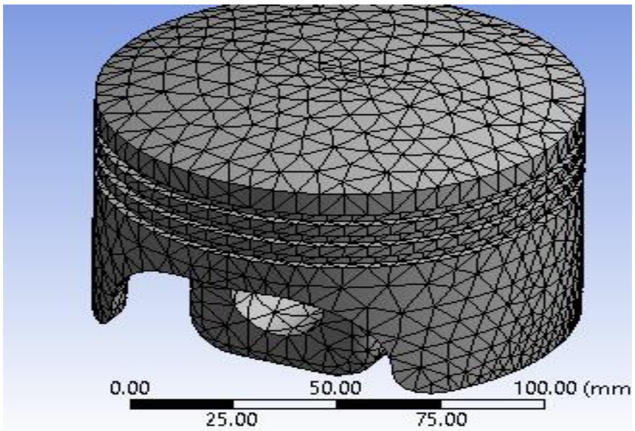

Figure 8: Meshed view of Design 2

3.4 Material used

Material properties of piston material

Piston analysis carried out using piston materials Al 6061 and Gray cast iron alloy. Aluminum alloy composition and grade grey iron cast alloy are given in Table 1.

Table 1: Material properties of Piston

Parameters	**Al 6061**	**Grey Cast iron**
Density (Kg/m^3)	2770	7200
Young's Modulus (MPa)	71000	110000
Coefficient of thermal expansion (1/K)	2.3×10^{-5}	1.1×10^{-6}
Poisson's Ratio	0.33	0.28
Elastic modulus (GPa)	70	124
Ultimate Tensile Strength (MPa)	310	240
Thermal conductivity(W/m/^{0}C)	140	52

IV. RESULTS AND DISCUSSION

Static analyses of material deformity, stress and stress were conducted during application Average effective pressure of 0.228 MPa on the piston surface. For aluminum alloy material with iron grey cast.

Table II: Static analysis of Piston head with Design 1 model

Piston head with Design 1 model		
	Materials	
Results	Al 6061	Grey cast iron
Total deformation (mm)	0.00188	0.00123
Stress (MPa)	4.33	4.35
Strain	0.0000619	0.0000402

Table III: Static analysis of Piston head with Design 2 model

Piston head with Design 2 model		
	Materials	
Results	**Al 6061**	**Grey cast iron**
Total deformation (mm)	0.00133	0.000868
Stress (MPa)	3.95	4
Strain	0.0000559	0.0000365

The piston has a significant effect on the engine's performance; the piston material has an impact on the piston's strength. As anticipated, the highest stress intensity is seen on the bottom surface of the piston crown in both materials. The top of the aluminums alloy and grey cast iron piston absorbs the maximum displacement. Because of the thermal conductivity of the materials, the highest value of maximum temperature found in the piston is due to thermal conductivity, and the entire maximum heat flux is absorbed in both piston materials. As a result, with the advanced materials and various designing and analytic techniques, more research may be conducted.

Table IV: Comparison thermal results of Piston with Design 1 model

Piston head with Design 1 model		
Results	**Al 6061**	**Grey cast iron**
Total heat flux	11.24	6.90
Temperature drops on top surface piston	217.71	489.55

Table V: Comparison thermal results of Piston with Design 1 model

Piston head with Design 2 model		
Results	**Al 6061**	**Grey cast iron**
Total heat flux	13.48	7.59
Temperature on top surface piston	309.83	442.52

.

Graph of thermal analysis of piston with Design 1 model

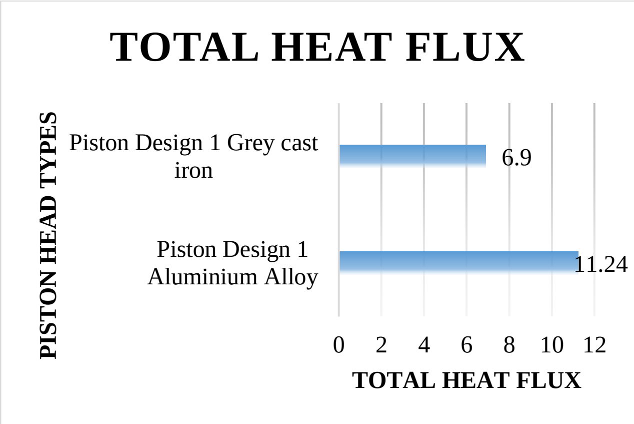

Figure 9: Comparison of total heat flux in Al Alloy and grey cast iron

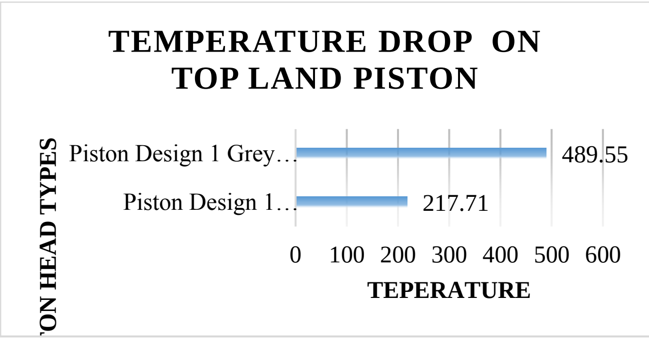

Figure 10: Comparison of Temperature on top surface in Al Alloy and grey cast iron

V. CONCLUSION

According to the results of the research, total heat flux was determined to be at a maximum of 13.48 W/mm^2 in aluminum alloy pistons with Design 2 shapes and a minimum temperature of 217.71 ^{0}C on the top surface of the Design 1 shape pistons with Al 6061. As a result, the Design 1 type of piston is shown to have superior thermal characteristics in this study. As a result of the experiment, a Design 1 head type piston has superior thermal characteristics to build pistons according to material thermal conditions. ANSYS 14.5 analytical software is used to complete the study. The Finite Element Method was used to arrive at these conclusions. As a result, with the advanced materials and various designing and analytic techniques, more research may be done.

REFERENCES

[1] Chunguang Fei, Tong Lei, Zuoqin Qian, "Piston Thermal Analysis of Heavy Commercial Vehicle Diesel Engine Using Lanthanum Zirconate Thermal-Barrier Coating", 15, Energies 2022.

[2] R. Sabarish, M. Prem jeya kumar, "The Design and Analysis of Piston - Steady State Thermal Analysis Using “Ansys”", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Vol. 9, Issue 3, Jun 2019, 197-204

[3] Krishnan S, Vallavi MS, kumarM, Hari Praveen A, 2017, “Design and Analysis of an IC Engine Piston using Composite Material", European Journal of Advances in Engineering and Technology, **4**, pp. 209-215.

[4] Sinha, Sarkar,Mandal, 2017, “Thermo Mechanical Analysis of a Piston with Different Thermal Barrier Coating Configuration", International Journal of Engineering Trends and Technology (IJETT), **48**, pp. 335-339.

[5] G Gopal, L. Kumar, K Reddy, Rao, 2017, "Analysis of Piston, Connecting rod and Crank shaft assembly", Elsevier, **4**, pp. 7810-7819.

[6] Shehanaz, Shankariah, 2017, "Design and Analysis of Piston Using Composite Material",International Journal of Innovative Research in Science, Engineering and Technology, **8**, pp. 16039-16048.

[7] Pandey, Jain, Bajpai, 2016, "Design, Analysis and Optimization of Four Stroke S.I. Engine Piston using Finite Element Analysis in ANSYS software", International Journal of Advance Engineering and Research Development, **9**, pp. 16-27.

[8] koteswaraRao, MansoorAhamed, Raju, 2016, "Fabrication Design and Analysis of Piston Using Metal Matrix Composites", International Research Journal of Engineering and Technology (IRJET), **11**, pp. 448-453.

[9] Vishal, Jain,Chauhan, 2016,"Design And Analysis Of Aluminum Alloy Piston Using CAE Tools", International Journal Of Engineering Sciences & Research Technology, **7**, pp. 332-339.

[10] Reddy, Goud, 2016, "Design and Analysis of the Piston by Using Composite Materials", international journal of professional engineering studies, **7**, pp. 153-162.

[11] Sundaram, Palanikumar, 2016,"Investigation and Analysis of Piston by Using Composite Material”, IJARIIE, **2**, pp. 1447-1454.

[12] Attar, Arora, 2016, “Transient Thermal Analysis of Internal Combustion Engine Piston in ANSYS Workbench by Finite Element Method", international journal of engineering sciences & research Technology, **6**, pp. 805-810.

[13] John, Mathew, Malhotra, Malhotra, 2015, "Design and Analysis of Piston by SiC Composite Material”, IJIRST –International Journal for Innovative Research in Science & Technology, **12**, pp. 578-590.

[14] Devan, Reddy, 2015, “Thermal analysis of Aluminum Alloy Piston", International Journal of Emerging Trends in Engineering Research (IJETER), **6**, pp. 511-515.

[15] Sonar, Chattopadhyay, 2015, "Theoretical Analysis of Stress and Design of Piston Head using CATIA & ANSYS", International Journal of Engineering Science Invention, **6**, pp. 52-61.

[16] Singh, Rawat, Hasan, Kumar, 2015, "Finite Element Analysis of Piston in ANSYS", International journal of modern trends in engineering and research, **4**, pp.619-626.

[17] Srinadh, Babu, 2015, "Static and Thermal Analysis of Piston and Piston Rings", International Journal of Engineering Technology, Management and Applied Sciences, **8**, pp. 51-58.