International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014

Thermo Mechanical Analysis and Weight Reduction of Air Compressor Piston using CATIA and ANSYS

Deepak Malhotra[1], Anit Bansal[2]
PG Student[1], Assistant Professor [2]
Department of Mechanical Engineering,
JCDMCOE, Sirsa, India

Sunil Chaudhry[3], PG Student
[3]Department of Mechanical Engineering,
JCDMCOE, Sirsa, India

***Abstract*—the present work investigates the thermal and mechanical strength of a compressor piston using Finite Element Analysis. For this work CATIA and ANSYS software's are used. The CAD model of a Piston is developed in CATIA Software. Thermal and mechanical analysis is performed using ANSYS Workbench. The material used for the piston is aluminum alloy which is light in weight and good thermal and mechanical properties. Aluminum alloy has high heat transfer rate but important thing to take care while using it is, because it expands appreciably on heating so right amount of clearance needs to be provided or else it will lead the engine to seize. The temperature fields, stress and deformation are calculated in different structures. The Finite Element Analysis results are significant to improve the component design at the early developing stage. The main objective of the work is to reduce the weight of the piston without affecting the performance so as to minimize the cost of the material.**

Keywords— Piston, FEA, CATIA V5, ANSYS

I. INTRODUCTION

A piston is a component of reciprocating engines, reciprocating pumps, gas compressors and pneumatic cylinders, among other similar mechanisms. It is the moving component that is contained by a cylinder and is made gas-tight by piston rings. In an engine, its purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod and/or connecting rod. In a pump, the function is reversed and force is transferred from the crankshaft to the piston for the purpose of compressing or ejecting the fluid in the cylinder. In some engines, the piston also acts as a valve by covering and uncovering ports in the cylinder wall.

The piston is connected to the crankshaft via the connecting rod, which allows rotation at both connections. The crankshaft converts the linear motion of the piston into rotational motion. To translate linear motion into rotational motion the crankshaft has an offset rotation. The work carried out the design optimization of piston using the ANSYS software. For optimization it is important to produce an optimized preliminary design. A number of random designs are generated by varying the values of the design variables within the specified limits till optimized design has been reached.

A piston is subjected to stresses caused by

- Combined effect of gas pressure on piston and the inertia of the reciprocating parts,
- Friction of the piston rings and of the piston force and

This leads to stresses and deformation in piston so a structural analysis of piston has been carried out. The stress, strain and deformation contours have been plotted and patterns are studied. The results are compared and verified with available existing results. The optimizations of piston also achieve the reduction in the static and thermal stress thus improving the engine performance.

The main objective of this work is to perform the Finite Element Analysis of Piston using ANSYS Workbench, so as to determine the thermal and mechanical strength in the Piston.

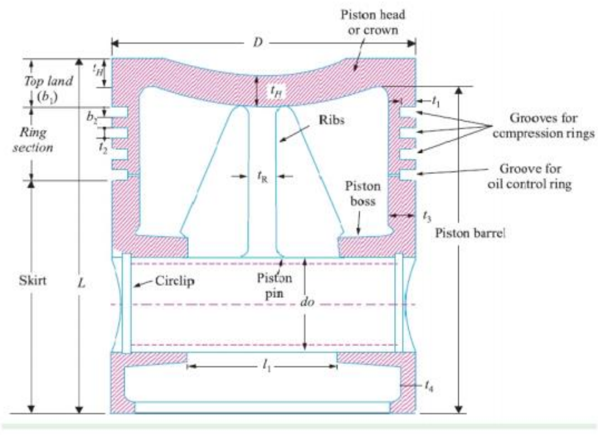

Fig. 1 Nomenclature of Piston.

II. METHODOLOGY

The problem for the analysis of Piston was taken from the paper [1]. Firstly the model of Piston is designed in CATIA V5 software. This model of Piston is then imported in Solid Works software and saved in parasolid file for no data loss. This parasolid format of Piston is then imported in ANSYS. The deformation and stress contours have been plotted after that checks the temperature distribution, stress, strain and deformation on the actual model of Piston. Then random designs are made with modification in dimension and analysis is done. The results obtained are compared with available results in literature survey. The optimization of Piston is carried out with a view to reduce weight and cost. For this study, the model was replicated from the above mentioned paper.

The Temperature and Pressure is kept the same as in Previous Paper is 81.97°C and 3.18 MPa respectively. The model of Piston was analyzed in ANSYS considering the component to be made up of Aluminium alloy. Aluminium alloy has Tensile Yield strength of 2.80E+08Pa and Tensile Ultimate strength of 420MPa. The CAD model of Piston designed in CATIA software and analysed in ANSYS workbench. However, there were some changes made in the geometry of the existing model.

III. MODIFICATIONS

The first step is to prepare a CAD model of Piston in CATIA V5. In this work, the thickness of Barrel is reduced by 1.5mm and fillet of 2mm is provided at the hole from the existing drawing. The fig. 2 shows the 2-D drawing of Piston after modification. CAD model of Piston is generated using CATIA V5 software. The fig. 3 shows the CAD model of Piston.

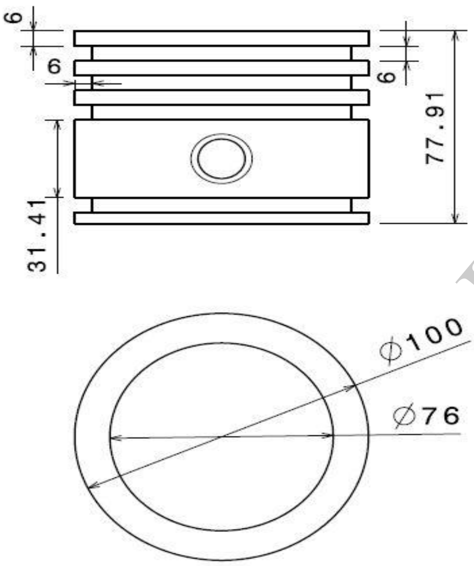

Fig. 2 Drawing of Piston

After generating the CAD models in CATIA V5, it is saved in igs format and this igs file is imported in Solid Works software. Then this imported file is saved in the parasoli (x_t) format to avoid the data loss. And finally the model imported in ANSYS Workbench.

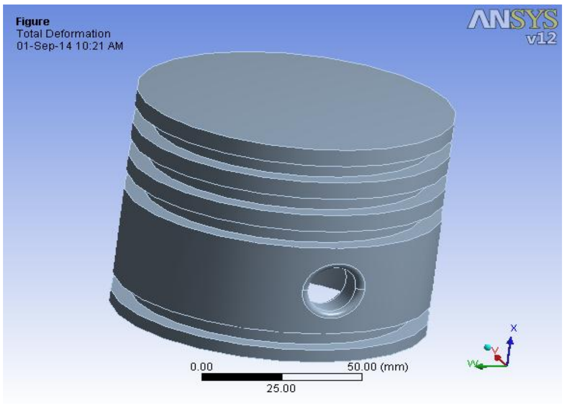

Fig. 3 CAD Model of Piston in ANSYS

IV. TEMPERATURE ANALYSIS OF PISTON

As per the loading condition apply the Temperature of 81.97 °C for Temperature Analysis. Fig. 4 shows the temperature applied on the Piston and Fig. 5 shows the convention on the Piston in ANSYS workbench.

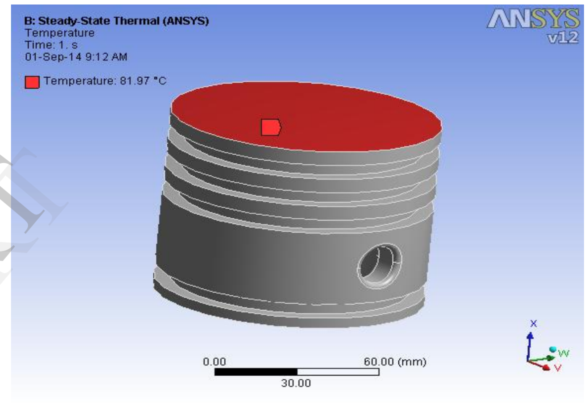

Fig. 4 Applied Temperature

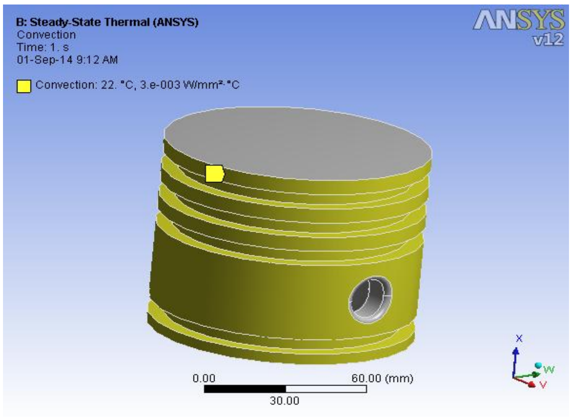

Fig. 5 Applied Convection

Figure 6 shows the Temperature Distribution in the Piston for a Temperature of 81.97 °C. The Average Temperature on the Piston ring is 65 °C.

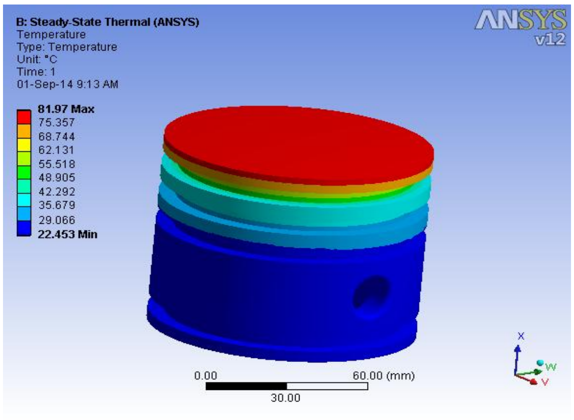

Fig. 6 Temperature Distributions

Figure 7 shows the total Heat Flux in the Piston for a Temperature of 81.97 °C. The maximum total Heat Flux on the top surface of the piston is 0.7475 W/mm^2.

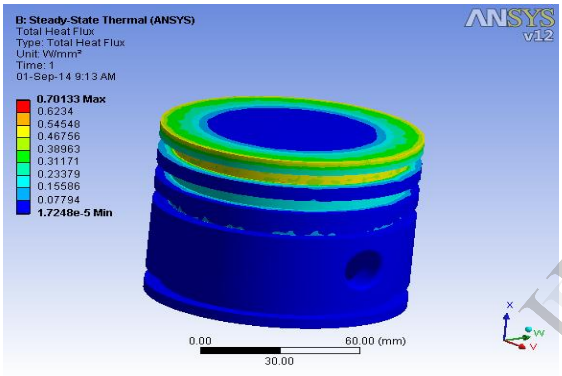

Fig. 7 Total Heat Flux

V. STATIC STRUCTURAL ANALYSIS

Static Structural Analysis is done to find out the total deformation, Equivalent (Von-Mises) stress, Maximum shear stress, Max. Principle Strain respectively in the Piston for a pressure of 3.18 MPa. The Shear Stress is found to be 54.243 MPa, Which is within the safety limits as prescribed in the paper. Moreover, this value is also less than the maximum stress value in the original component. Thus we can conclude that the new modeled component is well within the safe limit.

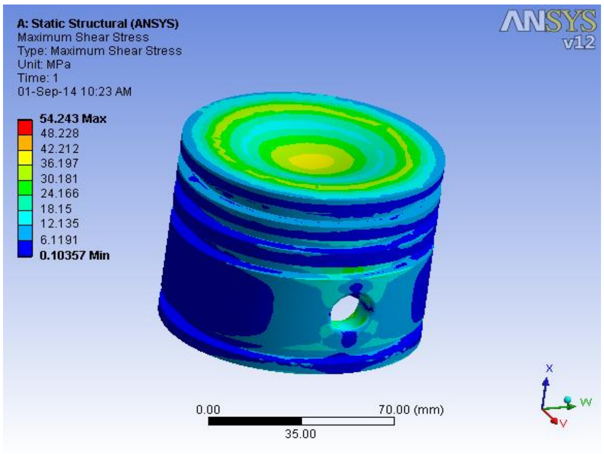

Fig. 8 Maximum Shear stress

The analysis results of Piston after optimization of the existing model are shown in table below.

Table1: FEA results of Piston

Sr. No.	Parameters	FEA Results
1.	Equivalent (Von-Mises) stress	99.09MPa
2.	Maximum shear stress	54.243 MPa
3.	Total deformation	0.128 mm
4.	Maximum principal strain	3.3 e-4

The ANSYS analysis shows that the Equivalent (Von-Mises) stress value obtained is 99.09 MPa,. For the design dimensions utilized in the analysis, the mass of the Piston is about 0.659 Kg. Hence, weight of the Piston is reduced about 9.47 %.

VI. CONCLUSION

Finite Element Analysis of the Piston has been done using ANSYS Workbench. From the results obtained from FE analysis, many discussions have been made. The results obtained are well in agreement with the available existing results. The model presented here, is well safe and under permissible limit of stresses.

1. On the basis of the current work, it is concluded that the design parameters of the Piston with modification give sufficient improvement in the existing results.
2. The average piston temperature beneath the piston ring is about 65 ºC.
3. It is shown in Fig. 7 is that from the top surface of the piston amount of heat transfer from 0.31171 to 0.4675 W/mm^2.
4. The weight of the Piston is also reduced by 9.47 %, thereby reducing the cost of the material.

REFERENCES

[1] Bhaumik Patel, Ashwin Bhabhor (2012) “thermal analysis of a piston of reciprocating air compressor” IJAERS, ISSN: 2249–8974, PP. 73-75.

[2] V. Esfahanian, A. Javaheri, M. Ghaffarpour (2006) “Thermal analysis of an SI engine piston using different combustion boundary condition treatments”, Applied Thermal Engineering 26 , PP. 277–287.

[3] Ekrem Buyukkaya , Muhammet Cerit (2007) “out Thermal analysis of a ceramic coating diesel engine piston using 3-D finite element method”, Surface & Coatings Technology 202 , ISSN: 0257-8972, PP. 398–402.

[4] R. Mikalsen, A.P. Roskilly (2009) “A review of free-piston engine history and applications”, Applied Thermal Engineering 27, ISSN: 2339-2352, PP. 2339–2352.

[5] A. R. Bhagat1, Y. M. Jibhakate (2010) “Thermal Analysis And Optimization Of I.C. Engine Piston Using finite Element Method”, Proceedings in Manufacturing Systems, Vol. 5.

[6] Bhaumik Patel, Ashwin Bhabhor (2012) ”analysis the design and prediction of temperature distribution of piston of reciprocating air compressor”, IJAERS, ISSN: 2249 8974, PP. 172-174.