

MODELLING AND SIMULATION OF PISTON

S VAMSI ^[1], C NAGAMANI ^[2], K SHARATH ^[3], B SAILESH ^[4]

Dept Of Mech,

Nagole Institute of Technology & Science, Hyderabad, Telangana

Abstract:

Piston is the major part of an internal combustion engine which converts the chemical energy of the fuel into the mechanical energy obtained at the crankshaft through the connecting rod. An internal combustion engine is acted upon by the pressure of the expanding combustion gases in the combustion chamber space at the top of the cylinder. This force then acts downwards through the connecting rod and onto the crankshaft. This paper shows the use of different material from the existing one in the design and analyzed it for better results. A parametric model of piston is modeled using CATIA V5 R20 software and analysis is done by using CAE tools of ANSYS

The piston design is for 150cc 4-stroke petrol engine in which the various dimensions of piston is calculated by analytical method considering maximum pressure condition and the material Aluminium alloy 2024-T361 is used in the design is based on the parameters like Vonmises stress, total

deformation and factor of safety and the weight reduction from the design.

1. INTRODUCTION

A piston is a component of engines. It is the moving component that is contained by a cylinder and is made gas tight by piston rings. In an engine, its transfer force from expanding gas in the cylinder to the crankshaft via a piston rod or connecting rod. As a main part in an engine, piston endures the cyclic gas pressure and the inertial forces at work, and this real working condition may cause the fatigue damage of piston, such as piston skirt wear, piston head or crown cracks and so on. The investigations denote that the greatest stress appears on the upper end of the piston and stress concentration is one of the mainly reason for fatigue failure.

On the other hand, piston over heating-seizure can only occur when something burns or scrapes away the oil film that exists between the piston and the cylinder wall. Understanding this, it's not hard to visually why oils with exceptionally high

film strengths are very desirable. Good quality oils will offer provide a film that stands up to the most intense heat and the pressure loads of a modern high output engine. Thermal analysis is a branch of materials science where the properties of materials are studied as they change with temperature.

Finite element method (FEM) are commonly used for thermal analysis. Due to the complicated working environment for the piston; on one hand, the finite element method (FEM) for the piston became more difficult, on the other hand, though there have many methods which are put forward to apply optimal design, the optimal parameters are not easy to determine. In this study, the piston is used in low idle and rated speed gas engine. In order to enhance the engine dynamic and economic, it is necessary for the piston to implement optimization. The mathematical model of optimization is established firstly, and the FEA is carried out by using the ANSYS software. Based on the analysis of optimal result, the stress concentrates on the upper end of piston has evaluated, which provides a better reference for redesign of piston. The study of the various authors found out that the stress is nothing but force per unit area the stress develop on the mechanical parts of the engine need to be considered for

the safe working of the piston it is possible to find out the various stresses acting on the mechanical parts by analytical and experimentation method

The paper presented by K. Jagdeesh [8] helped to study the various parameters of the piston mainly dealing with the finite element analysis of the vibrating piston who studied the effect of the water waves on the vibrating piston. The acoustic pressure values along the axis of the piston as a function of distance from the piston and frequency are evaluated. The radiation impedance of the vibrating piston is also evaluated. The results of FE analysis are compared with the theoretical values. Finally, he concluded that finite element analysis can be successfully used for simulation of the vibrating piston in the water. Which helped in the design of the solar projection which were important in the naval application also calculations helped to find out useful parameters such as axial pressure, far field pressure, radiation impedance.

F.S. Silva Fatigue on engine pistons – A compendium of case studies- provided valuable information on the first main conclusion that could be drawn from this work is that although fatigue is not the responsible for biggest slice of damaged pistons, it remains a problem on engine pistons and its solution remains a goal for

piston manufacturers. And it will last a problem for long because efforts on fuel consumption reduction and power increase will push to the limit weight reduction that means thinner walls and higher stresses. To satisfy all the requirements with regard to successful application of pistons, in particular mechanical and high temperature mechanical fatigue and thermal/mechanical fatigue there are several concepts available that can be used to improve its use, such as design, materials, and processing technologies.

2. RESEARCH OBJECT – PISTON

A piston is a component of reciprocating CI-engines. It is the moving component that is contained by a cylinder and is made gas-tight by piston rings. In an engine, its main purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod. Piston endures the cyclic gas pressure and the inside mechanical/thermal forces at work, and this operating condition may cause the fatigue damage of piston, like piston side wear, piston head cracks and so on.



Fig;1 piston

In CAE approach, some steps are same as that of conventional method. Here additionally conceptions, concepts are converted into engineering drawing, but it is then modelled on computer. Geometric model of product is made by the utilization of solid work software like CAD which enables better visualization of simple as well as complex models.

These models then further utilized for computerized analysis by using different CAE tools (FEA/CFD software's) depending upon the application before the prototype is made to check whether the components are going to work according to its intended function. After that once congruous results are obtained the final practical testing is carried out [1].

Figure 3 show the CAE approach for design a machine component.

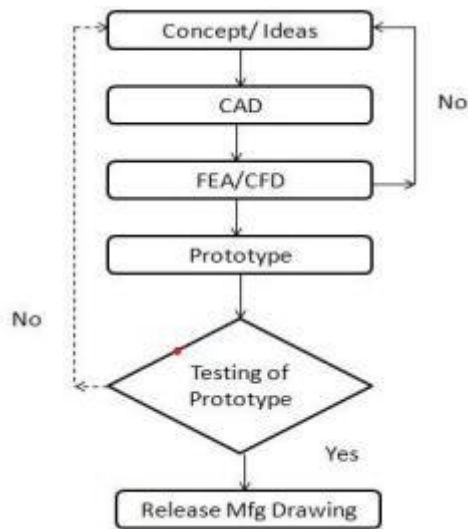


Fig-2 CAE Approach

A. Finite element analysis

FEA implement is the mathematical idealization of authentic system. It is a computer predicated method that breaks geometry into element and link a series of equation to every, which are then solved simultaneously to evaluate the demeanor of the complete system. It is utilizable for quandary with perplexed geometry, loading, and material properties where exact analytical solution are arduous to obtain. Most often utilized for structural, thermal, fluid analysis, however wide applicable for other type of analysis and simulation.

B. Methodology of piston analysis

The Piston during the working condition exposed to the high gas pressure and high temperature gas because of combustion. At the same time it is fortified by the minute terminus of the connecting rod with

the avail of piston pin (Gudgeon pin). Therefore, the methodology for analyzing the piston is considered as; the gas pressure given 180 bar is applied uniformly over top surface of piston (crown) and apprehended all degrees of liberation for nodes at upper moiety of piston pin boss in that piston pin is going to finetune. Considering the type of fit between piston pin and piston is clearance fit. Only the upper moiety of piston pin boss is considered to be fine-tuning during the analysis.

C. Geometry

The below image shows the geometry of piston foreign used for FEA.

A geometrical model of piston is prepared by modelling software's like CREO software or CATIA V5 which can also be modeled in the analysis software's like ANSYS. Figure 3 show the piston engendered by CAD software for further analysis.

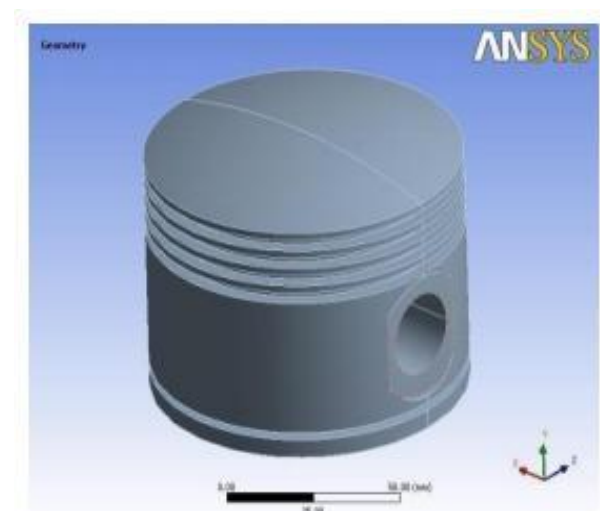


Fig-3 Geometry of piston

D. Finite element model

The element type selected for meshing in the piston model is SOLID187 tetrahedron type of element which is higher order tetrahedral element. The mesh count for the model contains 71910 number of nodes and 41587 number of elements. Figure 4 shows the meshed model of piston.

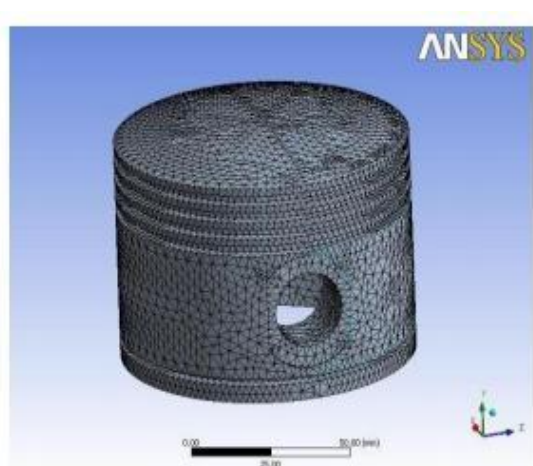


Fig-4 Meshed model of piston

E. Loading & boundary conditions

Figure 5 shows the loading and boundary conditions considered for the analysis. The uniform pressure of 18 MPa is applied on the crown of the piston (red color) and the model is constrained on the upper moiety of the piston pin aperture as shown by violet color.

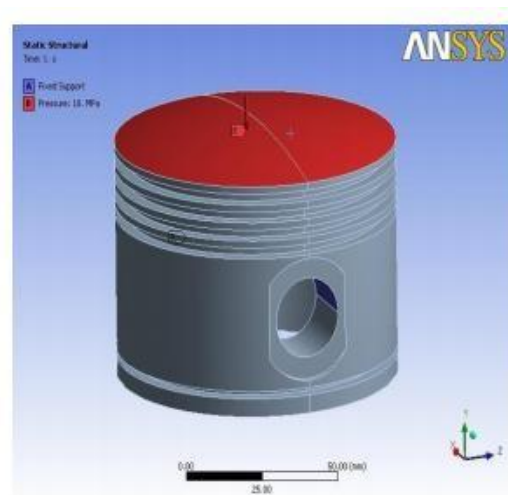


Fig-5 Loading and boundary conditions on piston

3. Result:

Total deflection Figure 6 shows the maximum deflection in the piston geometry due to the application of gas pressure which is 0.29669 mm observed at the central portion of the piston crown.

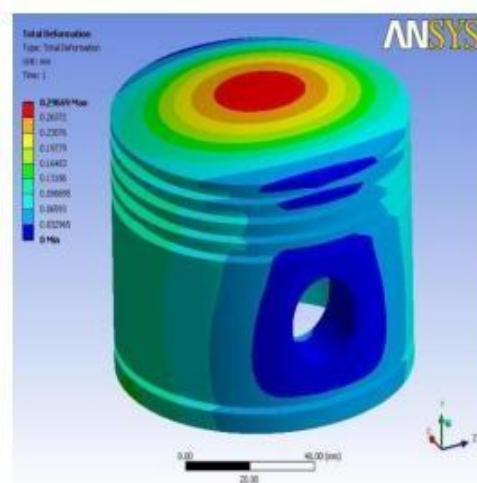


Fig-6 Total deflections on the piston head

Maximum principal stress

Figure 7 shows the distribution of localized and observed at the inner side of the piston pin

boss. The overall maximum stresses in the piston body are of value 231.25 N/mm² at the inner side of piston crown as well as piston boss.

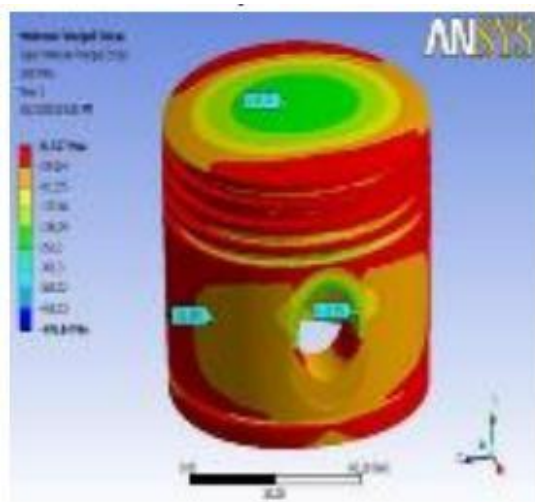


Fig-7 Maximum principal stress on piston

Minimum principal stress:

Figure 8 shows the distribution of the minimum principal stresses induced within the piston body. The most maximum values of equivalent stresses are going up to -376.74 N/mm², which are highly localized and observed at the inner side of piston crown & skirt junction. The overall maximum stresses in the piston body is -250.5 N/mm² at the top of piston crown.

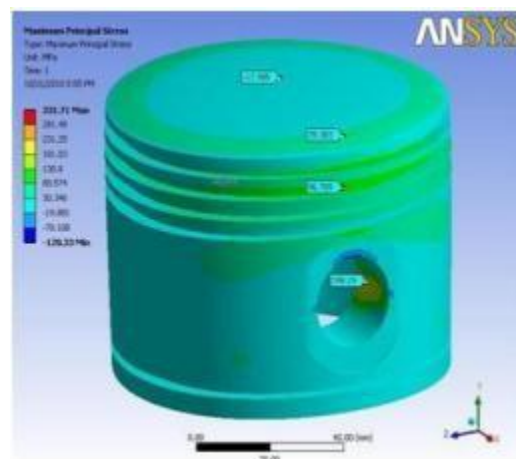


Fig-8 minimum principal stress on piston

Von mises stresses

Figure 9 shows the distribution of Von Mises stresses induced within the piston body. The utmost maximum values of equivalent stresses are going up to 200.97 N/mm².

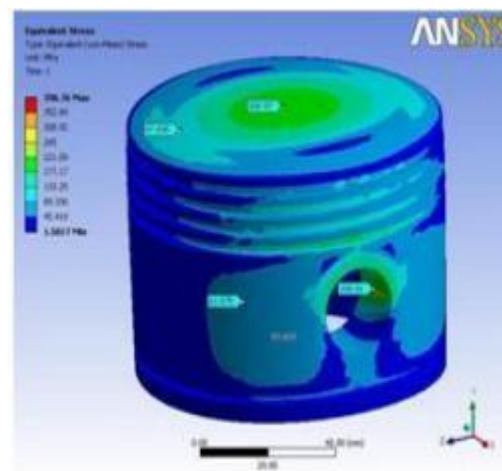


Fig-9 Von mises stresses

4. CONCLUSION

Piston skirt may appear deformation at work, which usually causes crack on the upper end of piston head. Due to the deformation, the greatest stress

concentration is caused on the upper end of piston, the situation becomes more serious when the stiffness of the piston is not enough, and the crack generally appeared at the point A which may gradually extend and even cause splitting along the piston vertical. The stress distribution on the piston mainly depends on the deformation of piston. Therefore, in order to reduce the stress concentration, the piston crown should have enough stiffness to reduce the deformation.

Also from analysis various results are obtained like The maximum deflection occurred about 0.29669mm due to the application of 180bar gas pressure on crown of piston, 231.25N/mm² of maximum principal stress is ascertained, - 250.5 N/mm² of minimum principal stress is ascertained. Also von mises stress of 200.97N/mm² is observed.

5. REFERENCES

- [1] "CAD/ CAM/ CAE "by Coagula N. K. [SciTech Publication]pp1.6, 17.1, 17.2
- [2] "Computer Aided Engineering in Compression Molding" (Antoine Rios,Bruce Davis and Paul Grumman, the Madison Group: PolymerProcessing Research Corporation 505 S. Ronsard. Madison, WI 53719)
- [3] C.H. Li., Piston thermal deformation and friction Considerations,SAE Paper 820086, 1982.
- [4] Properties and Selection: Irons, Steels and High- performance Alloy,ASM Handbook, vol. 1, ASM International, 1990.
- [5] Handbook of Internal Combustion Engines, SAE International.
- [6] Martin, Gary G. Failure of stationary pump engine piston,J Failure Anal Prevention 2004;4(1):37–9.
- [7] "Fatigue on engine pistons–A compendium of case Studies", F.S.Silva *, 31 March 2005
- [8] FE Analysis of a Vibrating Rigid Circular Piston in Water" by K.Jagadeesh and M. S. Vijaya2SASTECH Vol. VI, NO.1, in the yearApril 2007
- [9] F.S. Silva, Fatigue on engine pistons – A compendium of case studies,13 (2006) 480–492
- [10] "Finite Element And MBD Analysis Of Piston To Predict TheEngine Noise" S N Kurbet1, Vinay V Kuppast1 and Vijaykumar NChalwa2*Int. J. Mech. Eng. & Rob. Res. 2013