Thermal Analysis And Optimization Of I.C. Engine Piston Using Finite Element Method

A.R. Bhagat [1],
1, M.Tech,Heat Power,Mechanical Engineering, Department,KITS College of Engineering Nagpur

Y. M. Jibhakate [2]
2, Asso. Professor, Mechanical Engineering Department, KITS College of Engineering Nagpur, India.

Kedar Chimote[3]
3, Research Scholar Nagpur.

Absract: *This paper describes the stress distribution of the seizure on piston four stroke engine by using FEA. The finite element analysis is performed by using computer aided design (CAD) software. The main objectives is to investigate and analyze the thermal stress distribution of piston at the real engine condition during combustion process. The paper describes the mesh optimization with using finite element analysis technique to predict the higher stress and critical region on the component. The optimization is carried out to reduce the stress concentration on the upper end of the piston i.e (piston head/crown and piston skirt and sleeve). With using computer aided design(CAD), Pro/ENGINEER software the structural model of a piston will be developed. Furthermore, the finite element analysis performed with using software ANSYS.*

Keywords :*Ansys, Piston crown, Piston skirt,ProE, stress concentration, Thermal analysis etc.*

1. Introduction

A piston is a component of reciprocating IC-engines. 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.As an important part in an engine, piston endures the cyclic gas pressure and the inertial forces at work, and this working condition may cause the fatigue damage of piston, such as piston side wear, piston head/crown cracks and so on. The investigations indicate 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 overheating-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 see why oils with exceptionally high film strengths are very desirable. Good quality oils can 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. FEM method are commonly used for thermal Analysis. Due to the complicated working environment for the piston; on one hand, the FEA 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 is 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 become evaluate, which provides a better reference for redesign of piston.

2. Piston Design Data

The design data for designing of I.C engine Piston with the help of pro-E is collected from tata motors for Diesel engine vehicle.

Let us assume the other parameters required for desiginig of piston as bellow

Mass of Piston	1.36 kg
Mass Of Conn Rod	0.60 kg
Crankpin Mass	0.25 kg
Crank Radius	39.5 kg

Reciprocating Mass	1.56 kg
Cylinder Pitch	84.0 mm
Weight Of Flywheel	1.00 kg
Rotating Mass	1.12 kg

Engine Specifications

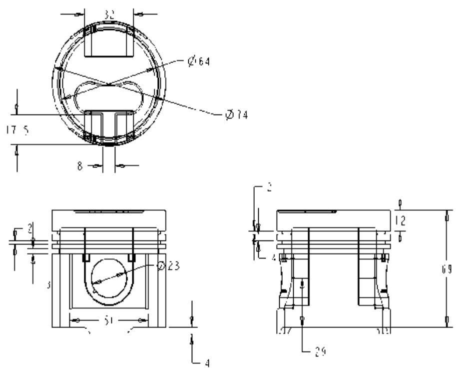
Power @ speed	75ps ,55Kw@4000rpm
Torque@speed	190Nm@1750rpm
Piston assembly	Assume
Stroke length	110mm
Engine Type	C.I Engine
Comp Ratio	17.6:1
Conn Rod Length	Assume

3. Theoretical Foundation

In engine, transfer of heat takes place due to difference in temperature and from higher temperature to lower temperature. Thus, there is heat transfer to the gases during intakes stroke and the first part of the compression stroke, but the during combustion and expansion processes the heat transfer take place from the gases to the walls.

So the piston crown/head , piston ring and the piston skirt should have enough stiffness which can endure the pressure and the friction between contacting surfaces. In addition, as an important part in engine, the working condition of piston is directly related to the reliability and durability of engine. So it is important for the piston skirt and the piston ring to carry out structural and optimal analysis which can provide reference for design of piston.

4. Piston Design



Top view

Front view

Side view

5. Piston Design Procedure and calculations of gas force

Brake Horse Power, BHP = 39 HP

Engine Speed, N= 850 rpm

Mechanical Efficiency,η_m = 80%

Piston Diameter, d=88.90 mm

Indicated Horse Power, IHP = 48.75HP

So the mean effective pressure on the piston crown ,piston top ring and the piston skirt should be restricted at low idle (850rpm) and at rated speed (2700rpm). The formula can be expressed as,

$$IHP = \frac{Pm \times L \times A \times N/2}{60 \times 10^5} \quad \ldots\ldots\ldots\ldots\ldots\ldots (1)$$

Where,
Pm Mean Effective Pressure (bar) ;
L- stroke length(mm) ; A –Area(mm^2) ;
N-speed (rpm) ;
IHP- indicated horse power (watt)
Thus for low idle speed (850RPM) Pm=74.18 bar and for rated speed (2700RPM) Pm= 23.33bar So, Calculation purpose Mean Effective Pressure, Pm = 23.33 bar taken.

Gas Force, Fg = Pm × A

= 2.33 × 6207.16

Gas Force, Fg = 14.49×10^3 N

5.1 Analytical Finite Element Analysis

In this work, due to the symmetry of structure, model of piston has been made in the Pro/E software, and then the FEM is established using ANSYS software. The 3-D 20-node solid element SOLID95 is applied to mesh the whole structure, and 27374 nodes and 14129 elements are obtained. In addition, in order to obtain the better result, the contact pair which is important to carry out the research should be established between the piston crown, piston ring and piston skirt. The contact is highly non-linear and need more computing power. In order to compute effectively, it is important to know the physical property and establish the reasonable model. In this study, the surface-surface contact is applied to the model. The FEA of piston is carried out by using the ANSYS software. The equivalent stress and the deformation of the piston are obtained respectively, as shown in the Fig As shown in the Fig. , the maximum stress appears at the centre and at edge of piston sleeve and the value of the stress is 228N/mm^2. As shown in the Fig. the maximum deformation occurs on piston and the value of deformation is 0.760mm. The analysis of contact is carried out by using

ANSYS workbench software, and the result is shown in the Figures It is correspond with the truth.

5.2 Thermal Analysis Of Piston Using FEA Approach

Piston is treated as Axis-symmetrical object about the centre axis of the piston q_1x, q_1y, q_2x, q_2y, q_3x, q_3y, q_4x, q_4y, q_5x, q_5y, q_6x, q_6y, q_7x, q_7y are the nodes position from 1 to 7 along x and y directions. Each 7245 N force acting vertical down word at node no 6 and node no7respectively.

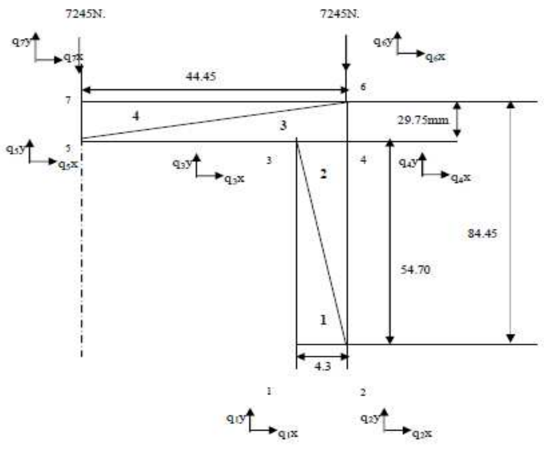


Figure 9.1: Piston Design

E = 72.5 X10^3 N/mm^2

μ = 0.334

ρ=7900 kg/m^3

5.3 Force Vector Matrix, F Radius of Centroid,

$$r_1 = \frac{(x_1 + x_2 + x_3)}{3}$$

$$r_1 = \frac{(40.15 + 44.15 + 40.15)}{3}$$

r1 = 41. 583 mm

and

$$|J_1| = 235.21$$

$$A_1 = \frac{|J_1|}{2} = 117.5\ mm^2$$

Strain Displacement Matrix, B_1

$$B_1 = \frac{1}{J}$$

Material Property Matrix, D_1

$$D1 = \frac{E(1-\mu)}{(1+\mu)(1-2\mu)}$$

Stiffness Matrix K_1

$K1 = t1 \times A_1 \times BT_1 \times D_1 \times B_1$

$K1 = (2 \times \pi \times r_1) \times A_1 \times BT_1 \times D_1 \times B_1$

Material Property Matrix, D1=D2=D3=D4

Similarly for element 2,3,4. Force Vector Matrix, F Piston is treated as Axis-symmetrical object about the centre axis of the piston q1x,q1y, q2x,q2y, q3x,q3y, q4x,q4y, q5x,q5y, q6x,q6y, q7x,q7y are the nodes position from 1 to 7 along x and y directions. Each 7245 N force acting vertical down word at node no 6 and node no 7 respectively.

5.4 Temperature load Vector Matrix,

$$\theta_1 = i = 2 \times \pi \times r_1 \times A_1 \times BT1 \times D1 \times \varepsilon_{0i}$$

$$\alpha_{Al} = 23 \times 10^{-6}\ W/^{\circ}C$$

ΔT=Element Temp-Room Temp= 150-40= 110^{0}C

Similarly for element 2,3,4.

Global Temperature Load Vector Matrix

$$\theta^G = 1 \times 10^6 [\theta]_{1\times 14}$$

Assembled Global Force Vector Matrix, FG=F+ θ^G

$$[F^G]_{14\times 1} = [K^G]_{14\times 14} = [U]_{14\times 1}$$

Where '**U** ' is Displacement Matrix. By solving the above equations, we get, displacement at various nodes.

Nodes	Analytical values of deformation	Ansys values of deformation
U2X	0.0216	0.805
U4X	0.0538	0.785
U6X	0.4241	0.760
U6Y	0.4366	0.724
U7X	-0.0024	0.077
U7Y	0.4198	0.434

6. Optimization Analysis With Steel By Ansys

Modern engines with variable valve train or different direct injection concepts require pistons with complex crown shapes which would often lead to a higher piston weight. Therefore in every new piston development, the piston geometry is optimized in particular in the ring belt/piston skirt area. Intensive application of numerical simulation methods enables significant weight reductions while increasing at the same time the load-bearing capacity. Newly developed alloys with better castability, but also higher fatigue resistance in the critical temperature and stress region, allow the realization of thinner wall structures. Improved casting methods enable large recesses for the ring belt and hence a considerable reduction in the piston weight.

But also the use of steel pistons in i. c engines is discussed again and again. The advantages of steel pistons such as reduced installation clearances, low fuel consumption figures and long service life would have to be evaluated against customer demands such as low emission levels, lightweight, efficient cooling and a competitive price. But up to now, there are no definite indications that steel pistons would be a viable concept for mass production. The analysis is done in ANSYS Workbench for teo materials aluminium alloy which is existing material and suggested material with different material properties. Following result are obtained as shown in figure.

Structural	
Young's Modulus	72.5e+06 MPa
Poisson's Ratio	0.33
Density	7.9e-006. kg/mm³

Table 1.Alluminium alloy properties

Structural	
Young's Modulus	2.21e+005 MPa
Poisson's Ratio	0.3
Density	7.833e-006kg/mm³

Table 2. steel alloy properties

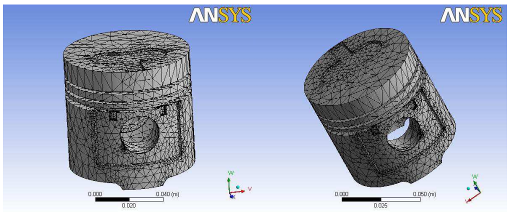


Figure 1:Comparison between meshed

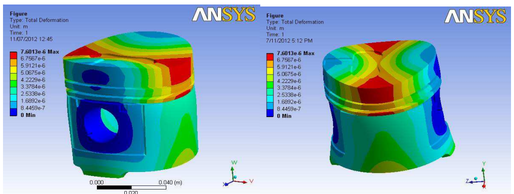


Figure 2: Comparison between total deformation

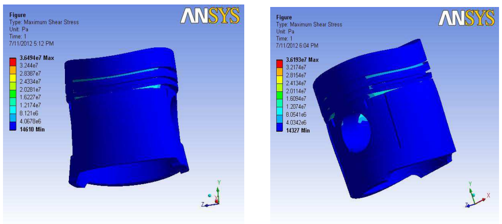


Figure 3: Comparison between Shear Stress

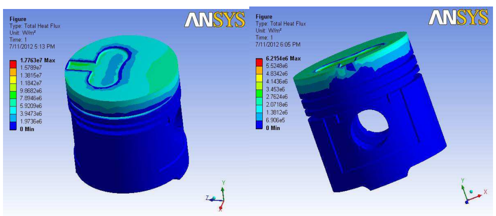


Figure 4:Comparison between total heat flux

7. 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.

1. The optimal mathematical model which includes deformation of piston crown and quality of piston and piston skirt.
2. The FEA is carried out for standred piston model used in disel engine and the result of analysis indicate that the maximum stress has changed from 228 Mpa. To 89 Mpa. And biggest deformation has been reduced from 0.419 mm to 0.434 mm.

5. References

[1] Richard Mittler , Albin Mierbach, Analysed finite element method can be used for the thermal model in the linear generator engine. Proceedings of the ASME Internal Combustion Engine Division 2009 Spring Technical Conference ,ICES2009 May 3-6, 2009, Milwaukee, Wisconsin, USA.

[2] P.Gustof, A.Hornik, Effects of increase of the load in turbocharged Diesel engine causes the change of value and temperature distribution International Journal of Achievements in Materials and Manufacturing Engineering",Vol. 35 Issue 2 August 2009.

[3] Tulus, Ariffin, A. K., Abdullah, S. Muhamad. N. presented the heat transfer model of piston pin Proceedings Of the 2nd IMT-GT Regional Conference Of Mathematics, Statistics And Applications University Sains Malaysia ,June 13-15 ,2006.

[4] Sanjay Shrivastva,Kamal Shrivastava, Rahul S. Sharma and K Hans Raj, Presented finite element modeling an analysis of automotive piston Journal of scientific & Industrial Research", vol .63, December 2004, pp. 997-1005.

[5] Gunter Knoll, Adrian Rienäcker, Jochen Lang, developed the interpolation of the three dimensional temperature field of a body Lehrstuhl für Maschinenelemente und Tribologie Universität Gh Kassel Germany, McGraw-Hill Book Company, p. 700 f.

[6] Thet T. Mon, Rizalman Mamat, Nazri Kamsah, presented simplified finite element model of spark ignition (SI) engine to analyze combustion heat transfer Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K.

[7]E. Abu-Nada, I. Al-Hinti, A. Al-Sarkhi, B. Akash presented thermodynamic analysis of piston friction in spark-ignition internal combustion engines Department of Mechanical Engineering, Hashemite University, Zarqa 13115, Jordan, Institution of Mechanical Engineers, London, pp. 133–145.

[8]Ashwinkumar S. Dhoble, R. P. Sharma, focused of engine industry research and development was on reducing the engine out emissions CO, HC, NOx and particulate matter (PM) R& D Centre, Mahindra & Mahindra Ltd.,Nashik**,** SAE Project 930797 (1993).

*[9]*D V Bhatt, M A Bulsara and K N Mistry**,** recognized that a large amount of energy is lost in the friction between Piston ring – Cylinder assembly Proceedings of the World Congress on Engineering 2009", Vol II WCE 2009, July 1 - 3, 2009, London, U.K.

[10] Huei-Huang Lee, Finite Element Simulations with ANSYS Workbench 12, SDC, Mission KS, 2010.

[11] J. Zecher, F. Dadkhah, ANSYS Workbench Tutorial, SDC, Mission KS, 2009.

[12] K. Lawrence, ANSYS Tutorial Release 11, SDC, Mission KS, 2007.

[13]Afzal A. and Fatemi A. (2003). A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods. SAE International.

[14]Augugliaro, G. and Biancolini, M. E. (2000). Optimization of Fatigue Performance of Connecting Rod.

[15]Beretta S., Blarasin A., Endo M., Giunti T. and Murakami Y. (1997). Defect Tolerant design of automotive components, Int. J. Fatigue. 19(4) : 319-333.