Bedajangam SagarKishor et al Int. Journal of Engineering Research and Applications www.ijera.com
ISSN : 2248-9622, Vol. 4, Issue 2(Version 6), February 2014, pp.33-36

RESEARCH ARTICLE OPEN ACCESS

Optimization of Geometrical Parameters of Four Stroke Engine Piston

BedajangamsagarKishor[a], N. P. Jadhav[b]
[a]Mechanical Department, SAOE, Kondhwa, India
[b]Mechanical Department, SAOE, Kondhwa, India

Abstract
The impact of crown thickness, thickness ofbarrel and piston top land height on stress distribution and total deformation is monitored during the study of actual four stroke engine piston. The entire optimization is carried out based on statistical analysis. It has been notified that after reducing the geometrical parameters of piston the stress distribution and total deformation is increased which is within the desirable tolerance limit. FEA analysis is carried out using ANSYS for optimum geometry. The work describes the mesh optimization with using finite element analysis technique to predict the higher stress and critical region on the component.
Key words: FEA, Crown thickness, Thickness ofbarrel, Top land height, Statistical analysis.

I. 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.

The two main requirements of the piston are as follows:

I. It should contain all the fluids above and below the piston assembly during the cycle.
II. It should transfer the work done during combustion process to the connecting rod with minimal mechanical and thermodynamic losses.

The piston is the heart of the internal combustion engine and is subjected to loads such as thermal and structural stress. The piston reciprocates within the cylinder.

Finite Element analysis is a simulation technique which evaluates the behavior of components, equipment and structures for various loading condition including applied forces, pressure and temperatures. Thus, a complex engineering problem with nonstandard shape and geometry can be solved using finite element analysis where a closed form solution is not available. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports for the model. FEA techniques can be used for mesh optimization, design optimization, material weight reduction, and shape optimization.

II. Geometric modeling

ANSYS module Design Modeler was used togenerate (Figure 2.1) and to parameterize (Figure 2.2)geometric model of piston. To avoid regenerationfailures mathematical relations were created betweenparameters and other dimensions by means of theDesign Modeler parameter manager.

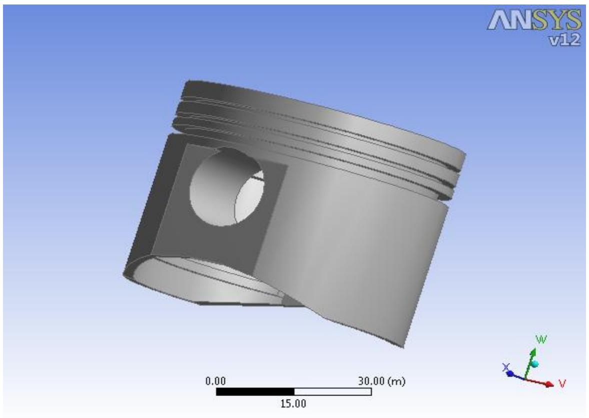

Figure 2.1: Geometry of Piston in Ansys.

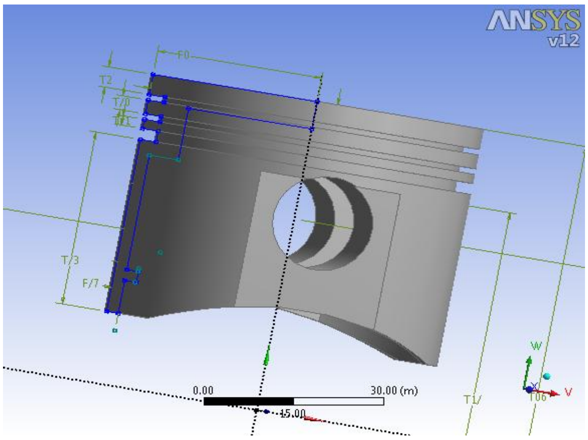

Figure 2.2: Parameterizegeometric model of Piston

III. Material properties and boundary conditions

Piston material was assumed to be aluminum alloy which is homogenous,isotropic and linear elastic with a Poisson's ratio of 0.33 and a young's modulus of 71GPa [2]. The model is loaded with a gasforce on the piston head (piston crown) (Figure 3.2). Fixed support is at the pin bore(Figure 3.1).

Table 1: Material Properties of Al

Density (Kg/m^3)	2770
Tensile ultimate strength (Pa)	3.10E+08
Tensile Yield strength (Pa)	2.80E+08
Compressive Yield strength (Pa)	2.80E+08

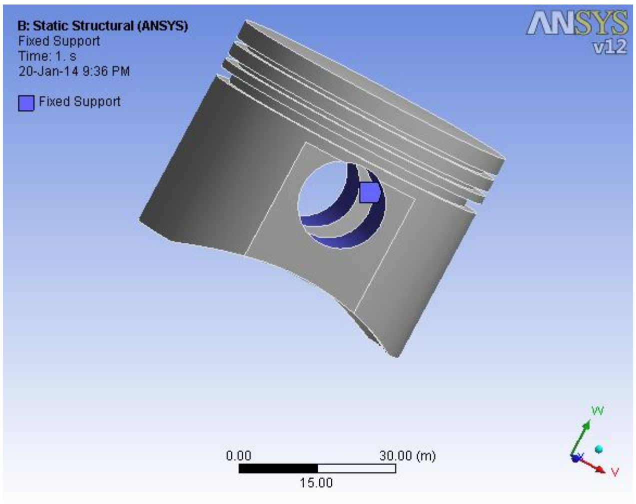

Figure 3.1: Frictional Less fixed support (Boundary Condition 1)

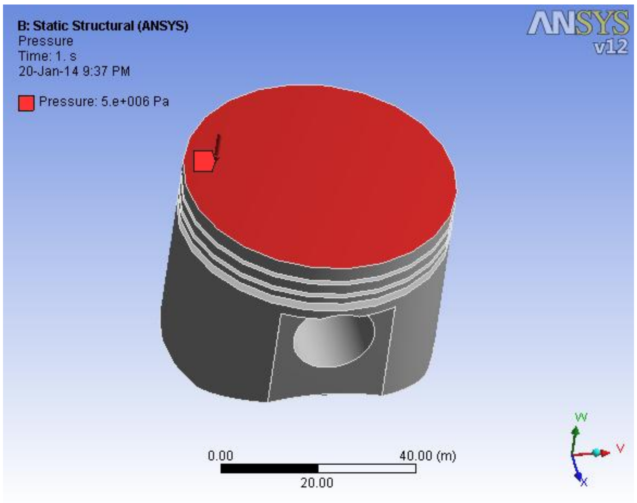

Figure 3.2: Downward gas force acting of Piston crown (Boundary Condition 2)

IV. Meshing

The meshing strategy for the optimization is not exactlythe same as for a single finite element analysis. Themesh for the optimization task has to meet four differentrequirements:

- ✓ Automated meshing must be possible for changing edges, angles and surfaces;
- ✓ The mesh quality must be comparable for every parameter combination;
- ✓ Accurate results for the changing geometry;
- ✓ In light of the expected number of calculations the calculation time should not be too long.

For a single finite element analysis it would be possibleto locate problematic regions with high stress gradientsand to refine the mesh at these specific regions. Butconcerning the second requirement this is not possiblefor the optimization model because critical regions aswell as maximal stress can change the location due toparameter variation. At this point sensitivity analysiswas started to unveil some important features of thefinite element model: are there any regenerationproblems, can the ANSYS Workbench mesher alwaysfind a mesh, where are the maximal stresses located.With ANSYS module Simulation moderate changes inthe geometry within the variation range of parameterswere made, many meshes with different element sizewere generated and finite element analyses were madefor many models with different geometry and differentmesh. Results of these analyses were used to comparenodal solution with element solution to get an idea ofresult quality. At the end the final mesh with thefollowing characteristics was generated: a generalelement size for the model was 4 mm, the relevance forthe model was set to 100 that is the highest level and theshape checking mode was set to aggressive. The finalmodel was meshed with 17636 nonlinear tetrahedralelements with 9431 nodes (Figure 4.1).

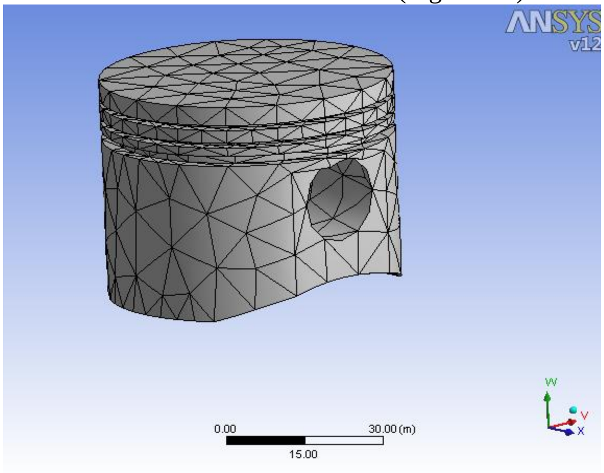

Figure4.1: Meshing Model of Piston.

V. Optimization Analysis of Aluminum Alloy Piston

To study the influence of parameters on piston stress levels, number of iteration are run using optimization tool in Ansys. Through these results it was possible to choose the best value for each parameter taking into account the stress levels on the piston and the mass of the piston. The aim is to obtain an assembly as light as possible and with some safety margin.

Factor of safety = Yield point stress / Working or Design Stress.

Automobile industries use factor of safety between 2.0 to 3.0 [7]. As piston is a critical component we are considering Factor of safety as 2.25. For Aluminum alloy, tensile strength is 280 MPa, Tensile Ultimate strength is 310 MPa. And mass of piston is 0.10194 Kg.

Working or design stress = 280 / 2.5 =124 MPa. Based on above analysis the maximum stress induce in the 67 MPa, which is less than 124 MPa (allowable stress). Hence piston safe and there is a scope for optimization.

So from the optimization results it is clear that the dimension 5.0mm can be reduced to 4.10mm, dimension 7.0mm can be reduce to 6.0mm, dimension 3.7mm can be reduce to 3.6mm. This result in Max equivalent stress of 105.12 MPa which is less than 124 MPa and solid mass is reduced to 0.09498 Kg. So from these result, piston model is modified to new dimensions and static analysis is carried out. The results obtained are well below the working stress and mass of piston is also reduced.

VI. Results and Discussion

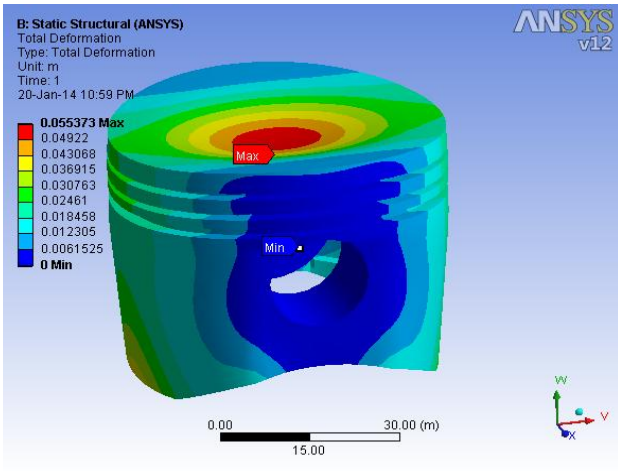

Figure 6.1: Total deformation before optimization

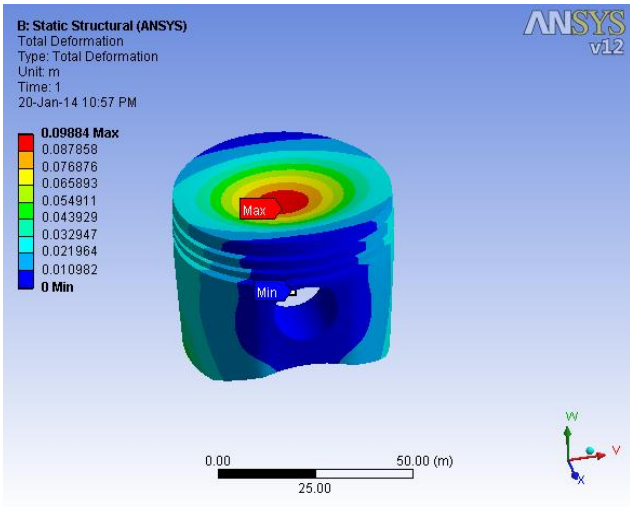

Figure 6.2: Total deformation after optimization

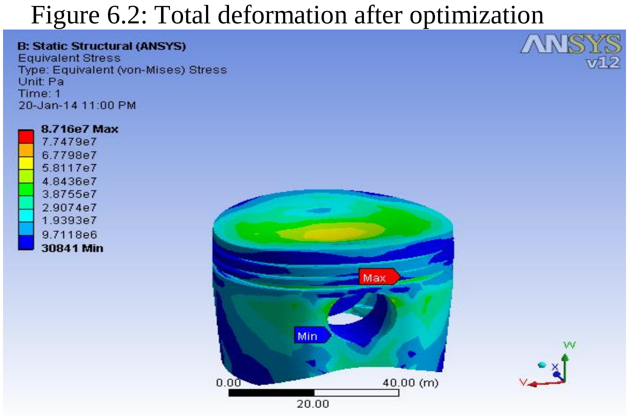

Figure 6.4: Equivalent stress before optimization.

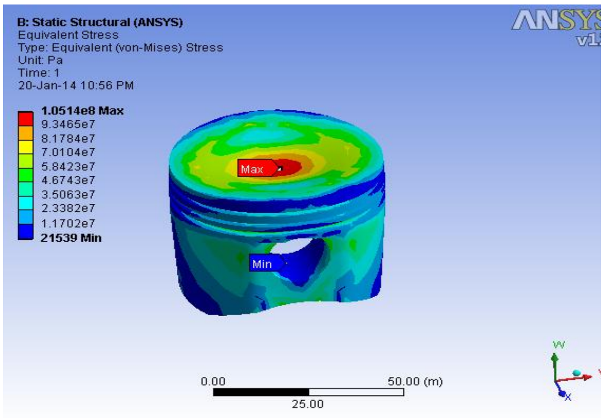

Figure6.3: Equivalent stress after optimization.

In this paper work analysis is done on actual piston which is used in 150cc engine. The result shows that the parameter crown thickness is mainly affecting on change in stress acting at piston head. The optimize result are shown in Table 6.1 below;

Table 6.1 Optimization Result

Parameters	Before Optimization	After Optimization
Thickness of crown (mm)	5	4.10
Thickness of barrel (mm)	7	6
Equivalent stress (MPa)	67.01	105.12
Total Deformation (mm)	0.05537	0.09884
Mass of piston (Kg)	1.0158	0.94643
Volume of Piston (mm^3)	36803	34291

VII. Conclusion

The FEA is carried out for actual piston which is used in 150cc four stroke petrol engine and results of analysis indicate that the stress and total deformation are increase in same rate.

- ✓ The stress is increase within the allowable limit i.e. below 124 MPa.
- ✓ The thickness of piston crown is optimizing at 16% because of this stress is increase at 36.25% and deformation increase at 44%.
- ✓ In this optimization found that mass is reduced at 7%.
- ✓ From optimization results it is clear that there is a scope for reduction in the thickness of piston skirt, piston crown wall thickness and piston crown thickness.

References

[1] **Mi Yan, Wang Tao,"** Finite Element Analysis of Cylinder Piston Impact Based on ANSYS/LS-DYNA", MEMS 2012.

[2] **Ch.VenkataRajam, P.V.K.Murthy, M.V.S.Murali Krishna, G.M.PrasadaRao,** "Design Analysis and Optimization of Piston using CATIA and ANSYS", January 2013, issue 2 volume 1.

[3] **Anil kumar, Kamaldeep Grover, BalvinderBudania,"** Optimization of Connecting Rod Parameters using CAE Tools", Vol. 1 Issue 3 September 2012.

[4] **Thet T. Mon, RizalmanMamat, NazriKamsah,** "Thermal Analysis of SI-Engine using Simplified Finite Element Model",Vol III WCE 2011, July 6 - 8, 2011.

[5] **Ghodake A. P., Patil K.N, "Piston** Design and Analysis by CAE Tools ", ISSN: 2250-3021 ISBN: 2878-8719 PP 33-36 National Symposium on engineering and Research.

[6] **Pravardhan S. Shenoy and Ali Fatemi,** "Connecting Rod Optimization for Weight and Cost Reduction", 2005-01-0987.