ISSN: 0974-5823 Vol. 6 No. 3 December, 2021

International Journal of Mechanical Engineering

Coupled Field Analysis and Design Optimization of Piston

Manu Singhasth
M.Tech (MachineDesign) Student, Dept. of Mechanical Engineering, Vishwavidyalaya College Ambikapur (Approved Vishwavidalaya Engg. College by AICTE and Affiliated to Chhattisgarh

Poonam Diwan
Assistant Professor, Dept. of Mechanical Engineering, Vishwavidyalaya College Ambikapur (Approved Vishwavidalaya Engg. College by AICTE and Affiliated to Chhattisgarh

***Abstract* - The piston is one of the critical parts of IC engine cylinder which is subjected to pressure by hot gases. The objective of current research is to investigate the effect of geometric design variables of piston on its strength. The design points are generated using Taguchi optimization method and stresses are evaluated at each design points using techniques of Finite Element Method. The CAD modelling and static structural analysis of IC engine piston is conducted in ANSYS software. From the static structural analysis conducted on piston, the critical regions of high stresses and deformation are identified. The design of experiments data obtained from Taguchi optimization technique which aided to determine the effect of optimization variables on stresses. The stress analysis results have shown that piston crown is the most vulnerable part followed by piston land region.**

Index Terms - FEA, Piston, Response Surface Method

1.INTRODUCTION

The current automotive industry demands for light weight engines which could enhance fuel efficiency and generate higher power. The weight reduction of engine is possible by using lightweight materials having high strength and thermal resistance. The materials include increased use of "lightweight materials, such as advanced ultra-high tensile strength steels, aluminum and magnesium alloys, polymers, and carbon-fiber reinforced composite materials" [1].

Figure 1:
Piston design [1]

The piston is one of the critical components of engine which is subjected to high pressure from exhaust gases generated from fuel combustion. The major cause of piston failure is due to mechanical stresses or thermal stresses. The piston bears maximum pressure as compared to other parts of engine [1]. The use of lightweight materials can be used to achieve 20% to 40% weight without much compromise in strength of piston.

2. LITERATURE REVIEW

Anil Kumar et al. [2] have conducted static structural analysis of piston using numerical method. Two different pistons are analyzed using ANSYS FEA software to determine equivalent stress and deformation. These cases include piston without coating and piston with coating. The equivalent stress is increased by 16% and deformation is also increased by using optimization technique.
Murthy et al. [3] have conducted investigation on piston design subjected to design optimization for weight minimization. The software used for the analysis is ANSYS FEA. Different design of pistons is evaluated on the basis of stress under thermal and structural loading conditions.
Jatit et al. [4] have conducted FEA analysis of gasoline engine piston using ANSYS software.
The thermal boundary condition, "the stress and the deformation distribution conditions of the piston under the coupling effect of the thermal load and explosion pressure have been calculated, thus providing reference for design improvement" [4].
Elijah et al. [5] have conducted FEA analysis of 2 stroke 6S35ME diesel engine in order to reduce mechanical stresses and temperature by using design optimization. The parameters

used in the optimization are "combustion pressure, piston material and temperature" [5].

Swati et al. [6] has used FEM technique to investigate the strength of engine piston subjected to combustion pressure. The critical regions of high stresses and deformation are identified from the plots of stresses. The mesh setting has significant effect on stresses and deformation generated on piston.

Plamenov et al. [7] has investigated the effect of different design parameters of piston (i.e. barrel thickness and piston top land height) on stress and deformation generated. The optimization studies are conducted using ANSYS FEA software and the effect of barrel thickness on stress generation is established.

3. OBJECTIVE

The objective of current research is to investigate the effect of geometric design variables of piston on its strength. The design points are generated using Taguchi optimization method and stresses are evaluated at each design points using techniques of Finite Element Method. The CAD modelling and static structural analysis of IC engine piston is conducted in ANSYS software.

4. METHODOLOGY

The methodology of numerical analysis using FEM involves basically three stages i.e. preprocessing, solution and post processing. The piston design is developed using revolve tool of ANSYS design modeler. The cross section is initially developed using sketch and revolved around center axis to generate 3D solid model of piston. The dimensions along 3D model of piston is shown in figure 2 below. The dimensions are taken from literature [8].

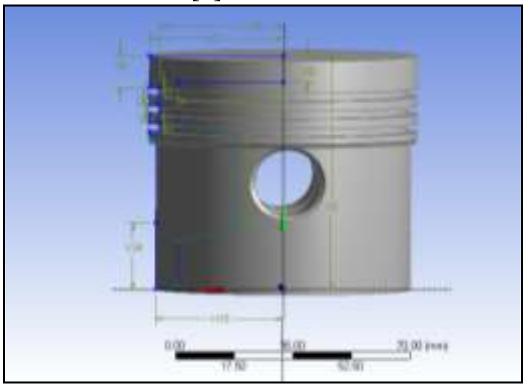

Figure 2:
CAD design of piston with dimensions

For design optimization, two different variables are selected which are "RL 1" and "RL 2" as shown in figure 3 below.

	Name	Value	Type
✓	RL1	2.8 mm	Length
✓	RL2	2.8 mm	Length

Figure 3:
Selection of optimization variables

Figure 4:
Meshing of piston

The piston model is discretized with fine sizing, the inflation set to normal and growth rate set to 1.2 with number of layers set to 5. The minimum edge length is set to 1e-3. The meshed model is shown in figure 4 above. The subsequent step involves applying structural loads and boundary conditions as shown in figure 5 below. The side cylindrical faces are applied with cylindrical support and top face is applied with pressure value of 6.5MPa.

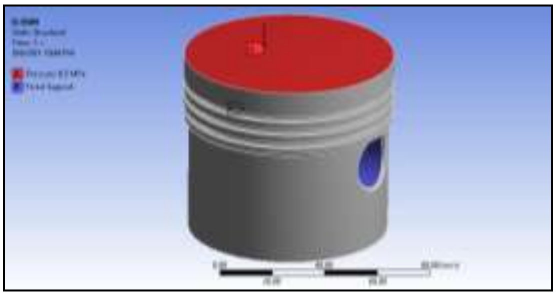

Figure 5:
Structural loads and boundary conditions

After applying loads and boundary condition, the sparse matrix solver is set to solution. The solution process involves "generation of stiffness matrix associated with each element followed by assembly of global stiffness element" [6].

5. RESULTS AND DISCUSSION

The FEA simulation is run to determine deformation, radial stress and tangential stress generated on piston. As it can be observed from figure 6 below, the maximum deformation is observed at the piston crown region with magnitude of nearly .059mm.

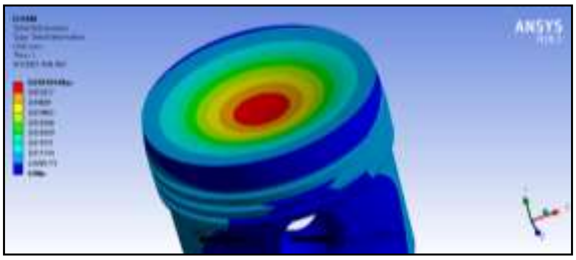

Figure 6:
Total deformation of piston

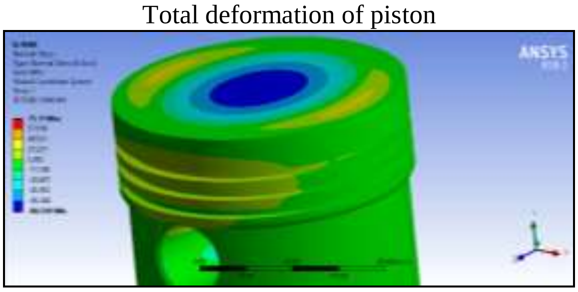

Figure 7:

Radial stress on piston

As it can be observed from figure 7 above, the maximum radial stress is observed at the top face of piston crown region where the stress is tensile with magnitude of 80MPa. The face underneath piston crown experiences compressive stress of 75MPa.

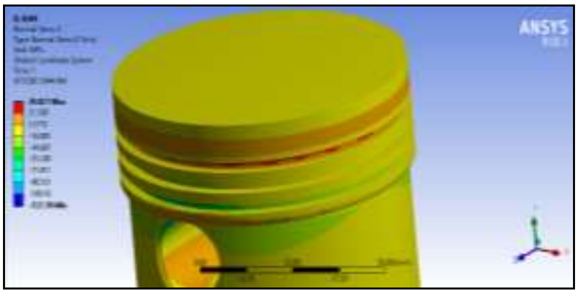

Figure 8:
Tangential stress on piston

The tangential stress plot of piston is shown in figure 8 above. The plot shows higher tangential stress at the piston ring groove region with magnitude of nearly 39.82 MPa and is lower on piston ring land region with magnitude of 6.985MPa. The design of experiment table is generated for radial stress and tangential stress using Taguchi response surface optimization technique.

Table 1:
DOE table for radial stress

Design Points	P4 - radial stress (MPa)
1	75.12269771
2	75.086451
3	75.12130743
4	75.10195314
5	75.16033
6	75.06471111
7	75.095933
8	75.10603029
9	75.11176114

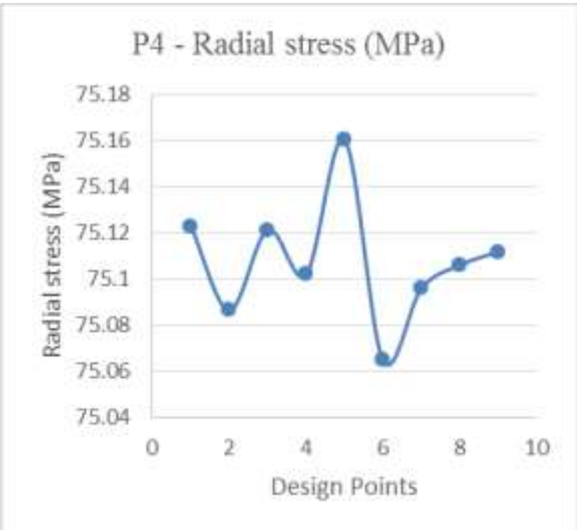

Figure 9:
Radial stress vs design points

The variation of radial stress vs design points is shown in figure 9 above. The maximum radial stress is obtained for design point number 5 and minimum radial stress is obtained for design point number 6. The dimensions corresponding to design point number 5 is 2.786mm for “RL1” and 2.782mm for “RL2”. The dimensions corresponding to design point number 6 is 2.795mm for “RL1” and 2.791mm for “RL2”.

Table 2:
DOE table for tangential stress

Design Points	P3 - tangential stress (MPa)
1	38.37746129
2	39.126027
3	38.064567
4	39.38210622
5	38.90718547
6	42.25720283
7	39.08462905
8	39.21840924
9	42.47176467

The DOE table for tangential stress is shown in table 2 above. The maximum tangential stress is obtained for design point number 9 and minimum tangential stress is obtained for design point number 3. The dimensions corresponding to design point number 9 is 2.78mm for “RL1” and 2.8mm for “RL2”. The dimensions corresponding to design point jnumber 3 is 2.79mm for “RL1” and 2.80mm for “RL2”.

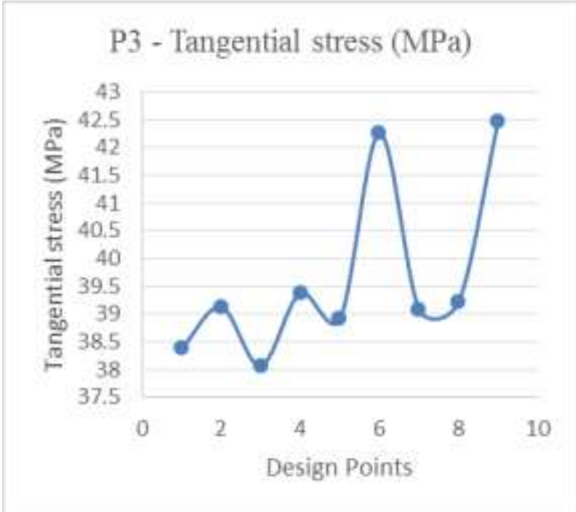

Figure 10:
Tangential stress vs design points

The variation of tangential stress vs design points is shown in figure 10 above. The maximum tangential stress is obtained for design point number 9 and minimum tangential stress is obtained for design point number 3. The dimensions corresponding to design point number 9 is 2.78mm for “RL1” and 2.8mm for “RL2”. The dimensions corresponding to design point number 3 is 2.79mm for “RL1” and 2.80mm for “RL2”.

6. CONCLUSION

The FEM is a viable tool in determining structural characteristics of IC engine piston. From the static structural analysis conducted on piston, the critical regions of high stresses and deformation are identified. The design of experiments data obtained from Taguchi optimization technique which aided to determine the effect of optimization variables on stresses. The stress analysis results have shown that piston crown is the most vulnerable part followed by piston land region.

REFERENCES

[1] Gadde Anil Kumar "Design and Analysis of an IC Engine Piston and Piston Rings by Using Three Different Materials" International journal of advances in mechanical and civil engineering", volume-4, issue-2, aprl.-2017

[2] P. V. K. Murthy "linear static structural analysis of optimized piston for bio-fuel using ansys" International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN 2249- 6890 Vol. 3, Issue 2, Jun 2013

[3] M. V. S. MURALI KRISHNA "Design Analysis and Optimization of Piston using CATIA and ANSYS" International Journal of Innovative Research in Engineering & Science ISSN 2319-5665 (January 2013, issue 2 volume 1)

[4] Jatit & LLS "An Analysis to Thermal Load and Mechanical Load Coupling of a Gasoline Engine Piston" Journal of Theoretical and Applied Information Technology 20th February 2013. Vol. 48 No.2© 2005 - 2013

[5] Elijah Musango Munyao "Simulation of Thermal-Mechanical Strength for Marine Engine Piston Using FEA" Journal of Engineering Research and Applications www.ijera.com ISSN : 2248- 9622, Vol. 4, Issue 3(Version 1)

[6] Swati S Chougule "Piston Strength Analysis Using FEM" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 3, Issue 2, March - April 2013, pp.1724- 1731

[7] Radoslav Plamenov Georgiev "Design a four-cylinder Internal Combustion Engine" International Journal of Mechanical Engineering Research & Applications Vol. 1 Issue 5, October – 2013. ISSN: 2347-1719

[8] V Alexa "Mechanical and thermal analysis of the internal combustion engine piston using Ansys" IOP Conf. Series: Materials Science and Engineering 163 (2017) 012043