International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 10 | OCT 2018 www.irjet.net p-ISSN: 2395-0072

Transient Dynamic Analysis and Optimization of a Piston in an Automobile Engine

Krupal A[1], Chandan R[2], Jayanth H[3], Ranjith V[4]

[1]*M.Tech Scholar, Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru*
[2,3,4]*Assistant Professor, Dept. of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru*

---***---

Abstract - *Piston sometimes called as heart of the reciprocating engine. In any energy conversation system, piston plays an important role. Piston failure takes place due to mechanical and thermal stresses. Compared to other parts of IC engine the piston works on very high thermal conditions and is the most stressed component of the engine. The main objective is to find out the maximum stresses acting on the piston structure. The project deals with structural and transient dynamic analysis. This is done by generating 3D model using solid edge and then meshed with suitable loads and boundary conditions by importing it to ABAQUS. A finite element model of a piston is established by using the FEM software. After the finite element modelling, natural frequencies, mode shapes and the participation factors are obtained from the modal analysis. By performing this, the maximum stresses acting on the piston structure is found. The stresses obtained from the FEA are used to get different modes of vibrations and the factors are used for determining maximum stresses with respect to time in stress distribution. These maximum stresses acting on the piston are found through the transient dynamic analysis. In this project the aluminium alloy A6061 is used due to its own advantages. Finally optimization is done by reducing the weight of the piston and its size without affecting their characteristics.*

Key Words: *Piston*, Finite element analysis, *modal analysis, transient dynamic analysis*, *optimization*.

1. INTRODUCTION

Piston is one of the principal part in the engine. Its motivation is to exchange constrains from developing gas inside the chamber to the crankshaft by the help of connecting rod. Since the piston is the principle part of an engine, its development creates a change in the system called as imbalances. These irregular characteristics or imbalances for the most part prove itself as a vibration, which makes the engine get harmed. The contact between the dividers of the chamber and the piston rings in the long run outcomes in wear where it diminishes the compelling existence of the component. The sound produced by engine can be irritable therefore, many engines are provided with noise suppression supplies which are utilized to reduce vibrations and noise. Transient means, something that fades with time. The dynamic analysis which is in a time domain is called Transient Dynamic Analysis. If the time history of loading is added then we will get the time history of response. That means, load vs. time will be the input. Response (displacement, stresses, velocities, and acceleration's) vs. time will be the result. In dynamic analysis, there is frequency/modal analysis and transient response analysis. Optimization refers to minimize the shape and size of the masses without affecting their characteristics and is to minimize the stresses occurring on the piston head. To reduce the stress concentration; the deformation must be reduced while the piston head should have enough stiffness. To know about the influence of parameters on piston stress levels, number of iterations should be done by Optimization tool.

2. METHODOLOGY

The basic step is to design the piston using specification. Create a 3D model of piston by using solid works (solid edge) next it is imported in ABAQUS/CAE. And the next step is to mesh the 3D model of piston using the tetra elements. Each and every component is detached finely meshed and then assembled back to its original shape. After the meshing is completed then Perform stress and transient dynamic analysis is done by using ABAQUS/standard. Then the transient dynamic analysis is performed for Piston. The stress distribution, vibration and transient dynamic response of the piston are evaluated. Load conditions for all the tests are as mandated as per recommended standards. The instantaneous response of the piston is evaluated by transient dynamic analysis. The strength and transient dynamic characteristics of the Piston design is evaluated. Further optimization based on the strength of the piston is also evaluated as discussed further.

2.1 Stress Analysis

Linear static stress analysis of piston structure is carried out using ABAQUS software. Initially the component is meshed in Hyper Mesh and is imported to ABAQUS. Hyper mesh is a pre-processor and post- processor, ABAQUS is the solver. Static stress analysis determines the maximum stresses induced in the piston structure to identify maximum compression in the structure. Piston structures with maximum compression loads are considered further analysis.

2.2 Modal Analysis

Modular investigation is the normal attributes of the mechanical structure. Every modular indicates the damping ratio, natural frequency and mode shape during vibration.

There are two kinds of modular investigation techniques, experimental modular analysis and calculating modular analysis. In the primary strategy modular parameters are obtained from the securing framework information and yield motions in the test. The following strategy is gotten by the limited component figuring. The reverberation and excitation make the blade to vibrate with single or mixed modular shapes. These conditions couldn't be cleared up utilizing strain gauges. Along these lines, here the ABAQUS is utilized for investigation, to decide the regular recurrence and mode shape.

2.3 Transient Dynamic Analysis

Under the action of time dependent loads the dynamic response of a structure is analyzed. The time varying displacement, strains and forces in a structure as it response to combination of static, transient and harmonic loads are determined. It also determines the response of structure to a time dependent loading considering the inertia and damping effects.

2.4 Material Properties

As shown in the table no [1] above we can assume that the materials properties and the values for aluminum alloy A6061 in a piston are given. Where this is made of aluminum alloy which contains of magnesium along with silicon as its major elements which are hardened, totally this material has outstanding properties which have a weld ability of high range and this is commonly obtained. So due to its application this aluminum 6016 is used to design the piston. Atmospheric conditions and resistance towards sea water for this alloy has excellent corrosion resistant and here this alloy also provides nice finishing applications and reacts well towards corrosion resistance. The above paragraph gives us an idea why this alloy is used in design of the piston.

Table-1: Material properties

SL NO	PROPERTIES	VALUE
1	Density	2700 kg/m3
2	Young's Modulus	70 Gpa
3	Poisson's Ratio	0.33
4	Tensile yield Strength	280 MPa
5	Coefficient of thermal expansion	18e-6 /K
6	Thermal conductivity	134 W/m K

2.5 Geometric Modelling

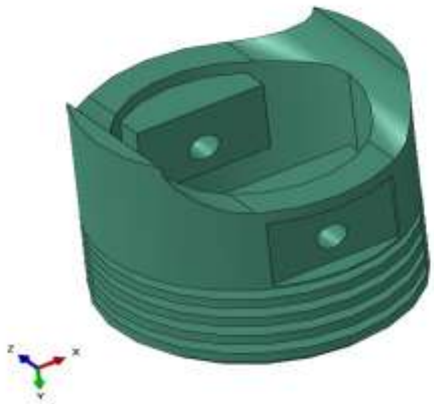

Fig -1: Geometric model

Fig -2: Front view

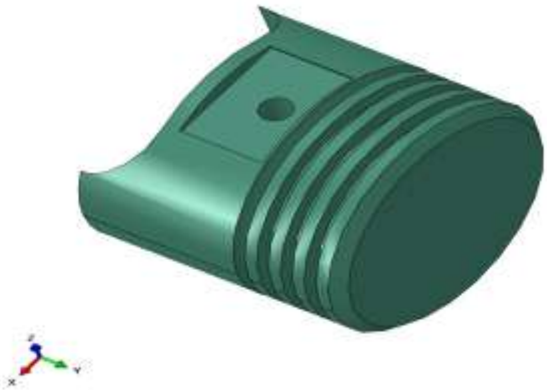

Fig-3: Isometric sector view

Solid works is a modeler which uses a featured based parametric approach to generate model geometry and assembly components. The Constraints are referred as parameters which used to determine the shape of the model geometry or assembly component values. Parameters can be geometric or numeric, numerical may be circle diameter or different lengths of line and geometric parts, such may be parallel, concentric, tangent etc. Model is built in solid works by 2D sketch. Then to define the size and location, dimensions are added to the sketch.

2.6 Meshing

Dividing the component into number of elements with a goal whenever the load is applied on the component it distributes the load uniformly is called as meshing. The figure shows the normal direction for the piston element. The top surface of the piston element has positive normal direction and is referred as positive face for defining the contacts. The bottom surface is negative normal direction along the normal. Both negative and positive surface within the element offsets referred to piston mid-surface.

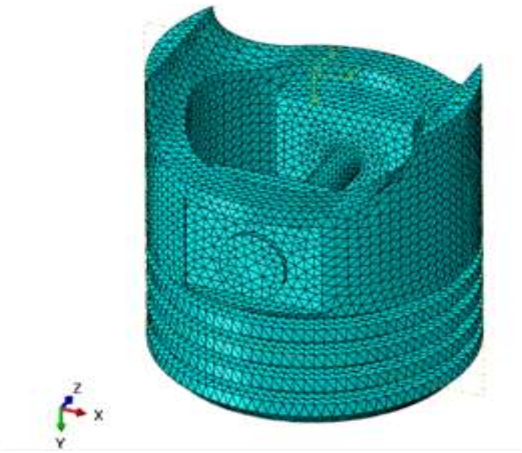

Fig -4: FE Meshed model

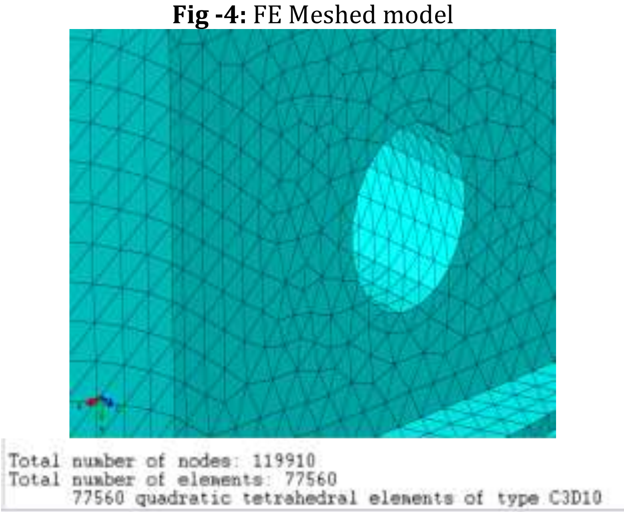

Fig -5: Close view of the meshing

The FE modeling is quadratic tetrahedral elements of type C3D10 as shown in above. Each element has 4 nodes for quadratic and three nodes for triangular elements. These elements provide accurate results when the structure is under constant bending and membrane strain approximations. A suitable finer element size is used to capture the bending and static deflection in the structure. Different elements, can replace to triangles. However, C3D10 element become stiffer than quadratic, hence C3D10 elements are used. Free meshing techniques are common meshing technique in ABAQUS and use hexa shell element to obtain the accurate and DOF.

3. RESULTS

3.1 Loads and Boundary Conditions

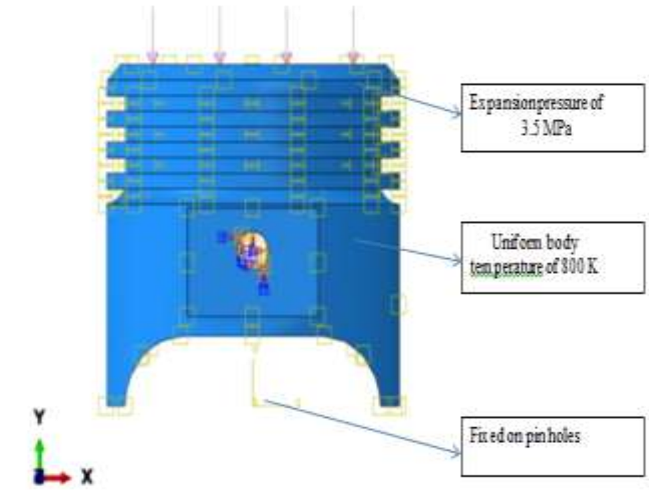

Fig -6: 3-D model Boundary conditions of piston

3.2 Static Stress Analysis

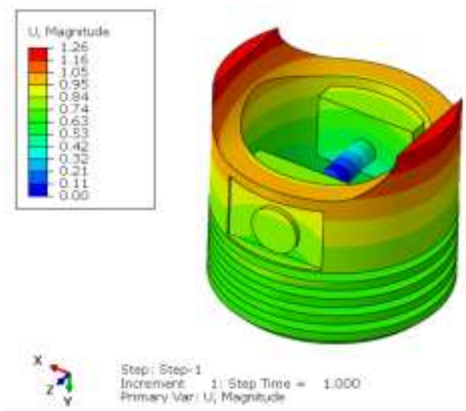

Fig -7: Radial growth plot of isometric sector model

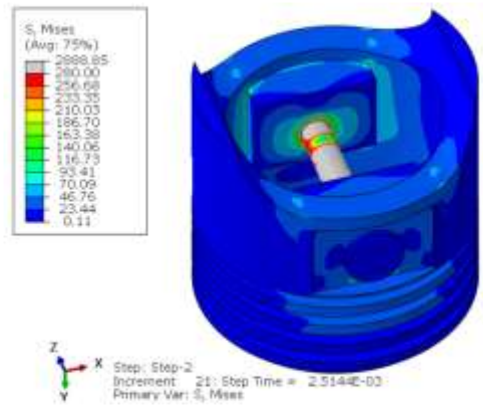

Fig -8: Elemental stress analysis of isometric section view of piston

Maximum displacement is found to be 1.27 mm which is within the acceptable limit and the maximum stress is found to be 280 MPa in the piston, where the piston pin experiencing a higher stresses.

3.3 Modal Analysis

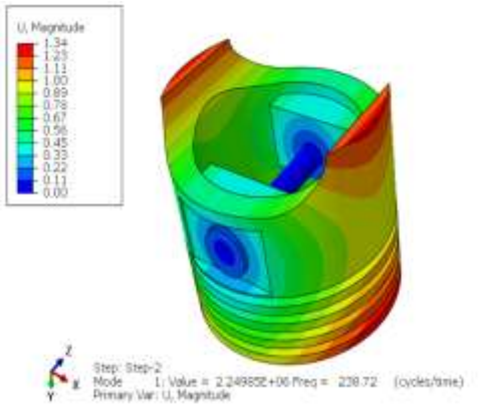

Fig -9: 1st mode of vibration

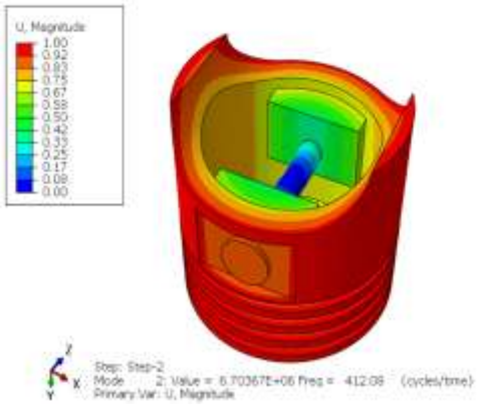

Fig -10: 2nd mode of vibration

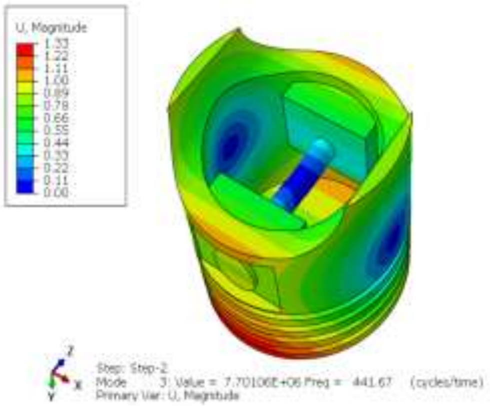

Fig -11: 3rd mode of vibration

The created model incorporated into ABQUAS solver to obtain better results, the 1st modes of vibration to obtain independent from applied load the nature frequency obtained is 238.7 Hz with corresponding to displacement 1.34 mm, the 2nd mode of vibration the natural frequency is 412.08Hz with corresponding displacement of 1mm, in 3rd mode the displacement is 1.33mm for natural frequency of 441.67Hz.From the above results, finally concluded that the displacement decreases with increasing natural frequency, The maximum displacement obtained is 1.34mm and the maximum natural frequency from this result is 238.7Hz.

3.4 Transient Dynamic Analysis

The [fig 12] indicates the transient dynamic analysis of a piston structure using ANSYS, it indicates that for the step time of 4.9042E-3 the maximum dynamic displacement obtained is 1.66 mm, which shows dynamic amplification factor of 1.3 which is under the safe limit. The [fig 13] indicates the plot of displacement v/s mode number, where the tests are conducted and plotted above. Here the displacements for particular modes excitation based on mass participation in independent directions are shown, among that the maximum displacement obtained is 1.6 mm for a particular mode number 0.3.

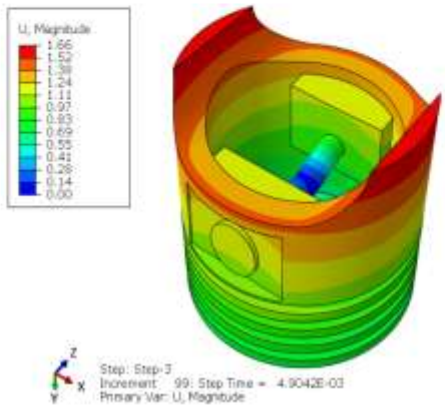

Fig -12: Transient dynamic analysis

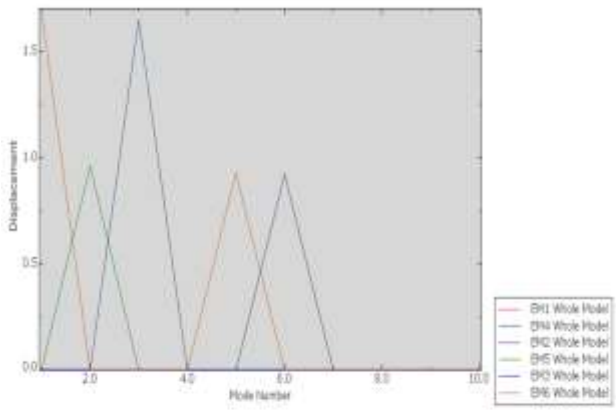

Fig -13: Displacements for particular modes excitation based on mass participation in independent directions

3.5 Optimization

Optimization refers to minimize the shape and size of the masses without affecting their characteristics. Optimization is to minimize the stresses occurring on the piston head. To reduce the stress concentration; the deformation must be reduced while the piston head should have enough stiffness. To know about the influence of parameters on piston stress levels, number of iterations should be done by Optimization tool. From these results it will be possible to choose the best value for each and every parameter taking into considerations, the stress levels on the piston and the mass of the piston. The aim is to minimize the stress present on the piston head of non-optimized model with some safety margin

4. CONCLUSION

The transient dynamic analysis proves to provide greater insight into the dynamic behavior of the piston. The dynamic amplification factor calculated for the 5 milliseconds loading is about 1.3. This is very much correlating to the established theory that the dynamic amplification factor for any structure under dynamic loading could by up to about 1.44. The static analysis result shows that the stress levels of 270 MPa for the piston is within yield limit. The deflections are also acceptable at 1.27 mm.

The modes excitations are captured through the modal analysis which shows a first natural frequency at 238 Hz. The mass participation plot shows the displacement excitation in individual directions at different frequencies as plotted .Finally the optimization is done in order to change the size and shape but not the characteristics.

REFERENCES

[1] Y.X. wang, Y. Q. Liu, H.Y. shi "Finite Element static and Dynamic Analysis For a piston" volume 97, pp. 3323-3326.

[2] Shiva yogi S Hiremath and I G Bhavi "Multi-Body Dynamic Analysis of an IC Engine Piston For shape optimization" volume 3, pp. 315-356, Oct 2014.

[3] Vaibhav V. Mukkawar, Abhishek, D.Banale, Nitin d, Bhusale, ganesh M, Surve "Design analysis and optimization of piston using CAE tools" volume 2, pp. 122-198, April 2015.

[4] M.Praveen kumar, Dr. Adinarayana "Design optimization of piston of an IC engine and IVestigation Its Influence on overall Assembly" Volume 7, Issue 4, 2017.

[5] G,Shiva Prasad, K. Dinesh Achan, E. Dileep Kumar Good, M. Nagaraju, k. srikanth "Design and analysis of piston of internal combination Engine on different materials using CAE tool ANSYS" Volume 2, Issued 3, May 2016.

[6] Gedala Chomu, Srinivasa Rao, I.Abilash, C.Rammohan Rao "Design and Analysis of piston by using different Material" Volume 6, no 2, pp. 63-71, Mar 2017.