Dynamic Simulation of Piston Motion to Predict the Piston Slap for Single Cylinder Four Stroke Engines

Pavan Kumar H[1] Prashanth N[2]
[1]P.G. Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Adichunchanagiri Institute of Technology, Chikamagalur, Karnataka, India Affiliated to Visvesvaraya Technological University, Belgaum, Karnataka, India

Abstract— In recent years environmental norms have been made mandatory for new design or design optimizations of any combustion based engines and prime movers. The greater demand for quieter engines and hence the technology proved to be the answer for these pollution regulations. Piston slap is still an important mechanical excitation source and noise generator in IC engine. To understand the complete and complex dynamics of piston, connecting rod and crank shaft, a three dimensional modeling and dynamic simulation approach is required. In the present work the study of piston-slap due to mechanical impact with a three-dimensional finite element model is carried out. The results are taken to predict the behaviour of the piston inside the combustion chamber and to predict the piston lateral motion which causes the piston-slap. The particulars utilized in this study of piston belongs to four stroke single cylinder Bajaj pulsur 150 cc bike engine. The geometrical modeling of the piston is done using CATIA V-5 modeling tool, finite element software (ANSYS) is used to implement physics and carryout simulations. The stress and displacement results are viewed and analyzed. These values are interpreted to predict piston-tilt due to the secondary motion of the piston and the piston-slap when piston reciprocates between TDC and BDC.

Key words: Piston Secondary Motion, Piston-Slap, CATIA, ANSYS, Multi Body Dynamics

I. INTRODUCTION

Environmental norms have been made mandatory for new design or design optimization of any combustion based engines and prime movers. The greater demand for quieter engines and hence the technology proved to be the answer for these pollution regulations. There are heaps of exploration works proposing for engine pistons new materials, geometries and manufacturing technologies, and this development has experienced with a consistent change in the course of the most recent decades and needed through examination of the littlest points of interest. One thing that has not changed is the fundamental working principle of piston. Pistons essential configuration is still basically same. So what is changed? The working atmosphere, Today's engine works harder, runs hotter and cleaner than ever before. And also it is expected to work longer with least maintenance. Even though there are an immense number of piston damages. These damage mechanisms have diverse beginnings and are normally due to fatigue, wear, and temperature related. Piston slap is a defect occurs in engine piston due to which noise is developed in the engine. The main reason for this defect is the secondary or transverse motion of piston while moving between cylinders two extreme ends i.e. TDC to BDC. Present work is related upon the analysis of piston damage because of piston slap. This work is study of engine parts viz, the main cause for engine vibration or noise is considered to be the piston. Normally in engine piston we can see two types of piston motion, first is the primary motion of piston and the other is secondary motion of piston. Primary motion of the piston is linear movement of piston from TDC to BDC within the engine cylinder due to combustion pressure. This movement is responsible for producing the power. This movement is schematically shown in the Fig.1 below. Secondary motion of the piston is because of impact load of the combustion and it is transverse movement of piston along with rotational motion about the wrist pin axis during piston travelling between BDC to TDC and vice versa. This movement is schematically shown in the above Fig.2.

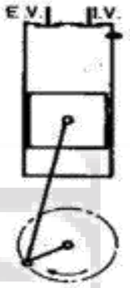


Fig. 1: Primary motion

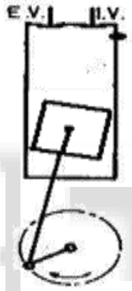


Fig. 2: Secondary motion

A. Reasons for Piston Secondary Motion

- Excessive clearance between cylinder bore and piston.
- Tilting of piston due to insufficient clearance in the piston pin bed.
- Due to miss alignment of connecting rod.
- Excessive big end bearing clearance.
- Axial thrust on piston pin due to alignment error between the axis of the piston pin and the crack shaft axis.
- Correct installation direction of the piston was ignored – piston pin axis offset.

B. Effects of Piston Secondary Motion

- Piston and piston ring fracture.
- Piston pin fracture.
- Damage of piston pin circlips.
- Seizer in the piston pin bores.
- Piston noise/piston slap.
- Cylinder and cylinder liner damage.
- Increased oil consumption.
- Power loss due to friction.

C. Objectives of Present Work

- Analytical design and analysis of piston.
- Linear and Bi-linear analysis of piston.

- Predict of piston slap in piston.
- Modal and Harmonic analysis.
- High cycle fatigue life evaluation of piston.
- Malty Body Dynamic (MBD) analysis.

II. METHODOLOGY

The design methodology consists of standard methods which specify the design method to be used. Piston is modelled using CATIA V5 by taking the dimensions from analytical design. For analytical design of piston, design procedure is followed as in the design data hand book. After creating the geometric model of piston, finite element software (ANSYS) is used to implement physics and carryout simulations. Next would be the force analysis of piston slap followed by the simulation aspects of the FEM. Once the results of the analysis are obtained, it is validated and design optimization is carried out for the best results.

A. *Material*

In this work the material used is aluminum alloy (A2618). Its chemical composition, physical properties, mechanical properties and thermal properties are shown in tables (1), (2), (3), and (4) respectively.

Constituent	Content (%)
Aluminum, Al	93.3
Copper, Cu	2.55
Magnesium, Mg	1.6
Iron, Fe	1.15
Nickel, Ni	1.08
Silicon, Si	0.19
Titanium, Ti	0.09

Table 1: Chemical composition of A2618

Physical properties	Values
Density (Kg/m^3)	2767.99
Melting point (^0C)	510

Table 2: Physical properties of A2618

Mechanical Properties	Values
Tensile strength	440 Mpa
Yield Strength	420 Mpa
Shear Strength	260 Mpa
Fatigue Strength	125 Mpa
Elastic Modulus	70-80 Gpa
Poisson's Ratio	0.33

Table 3: Mechanical properties of a2618

Properties	Values
Thermal Conductivity ($W/m/^0C$)	147
Coefficient of thermal expansion (1/K)	$25.9X10^{-6}$

Table 4: Thermal Properties of A2618

B. *Engine Specification*

The engine specifications of bajaj pulsur 150cc engine are shown in the table (5).

III. FINITE ELEMENT MODELING AND ANALYSIS

A. *Geometric Model of piston*

Geometric model is created according the calculated dimensions of piston using CATIA V5 software. The CAD model created is shown in the fig.3.

Description	Values
Engine Type	Four stroke petrol engine
Number of cylinders	Single cylinder
Bore	57mm
Stroke	56mm
Maximum power	15.06 KW @ 9000 rpm
Torque	12.5 NM @ 6500 rpm
Displacement volume	150 cc

Table 5: Engine specifications

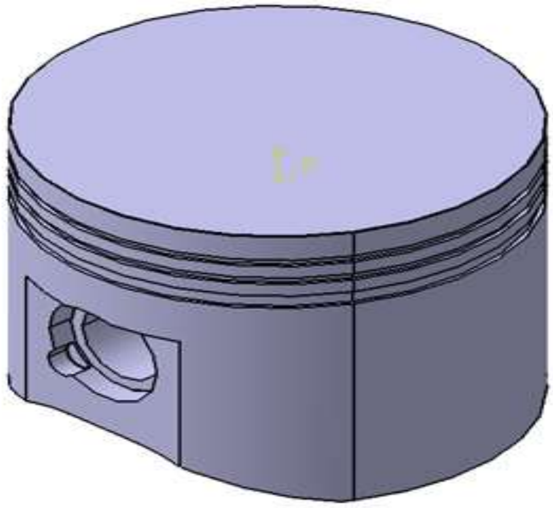


Fig.3 Isometric view of piston

B. *Finite Element Meshing*

The meshed model consists of nodes and elements. The continuum can be meshed using several types of elements. In this work final mesh with the following characteristics is generated: here tetra elements are used with the general element size for the model is 10, patch confirming, edge sizing, body sizing and face sizing methods are used. Fine meshing is done at the discontinuities since it is the stress concentrated location. Coarse meshing is done at other locations. Total number of nodes and elements here will be 66803 and 37840 respectively. The finite element meshed model of piston is as shown in fig.4.

Fig. 4 Meshed model of piston

C. *Defining the Boundary Conditions*

In this work boundary conditions used are,

- Frictionless support at the pin bore area and fixed all degrees of freedom.
- Downward pressure (16 Mpa) due to combustion gas load acting on piston head.

Finite element model of piston with the applied boundary conditions is shown in the below fig.5.

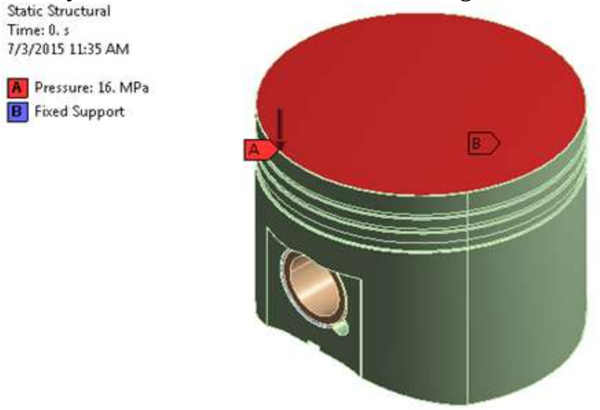


Fig. 5: Finite element model with boundary conditions

IV. RESULTS AND DISCUSSION

A. Linear Static Analysis

The below figure 6 shows the stress tensor obtained for the piston when static analysis is performed by applying the boundary conditions.

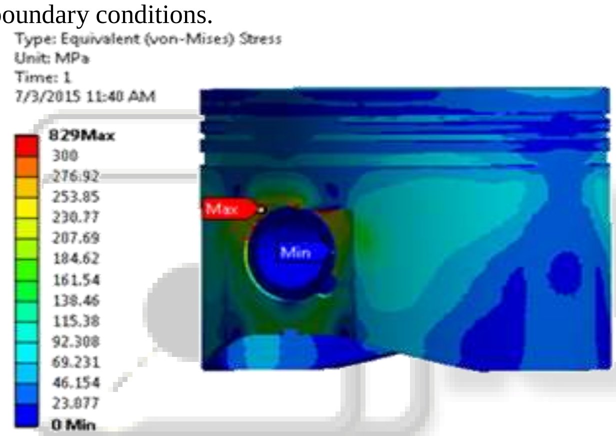


Fig. 6: Equivalent (von-mises) stress

Maximum stress observed in Fig.6 is 829Mpa which is higher than the allowable yield stress of the material i.e.420Mpa. This is due to the sharp edge which is present on the piston and this stress value is coming at one particular node only. By removing that sharp edge stress value can be reduced to a minimum.

B. Proposed Modified Model

To bring the stress value to a minimum, in the base model of piston we made some slight geometrical modifications. As in the previous section we observed, maximum stress value is due to the sharp edge which were present on the piston, this itself acts as a stress riser. So to minimize that, we removed that sharp edge and provided a fillet at that region. The modified CAD model is shown in the fig.7.

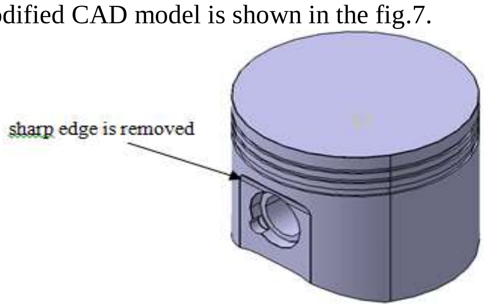


Fig. 7: Modified model of piston

C. Linear Static Analysis of Modified Piston

The below fig.8 shows the stress tensor obtained for the modified piston when static analysis is performed by applying the boundary conditions.

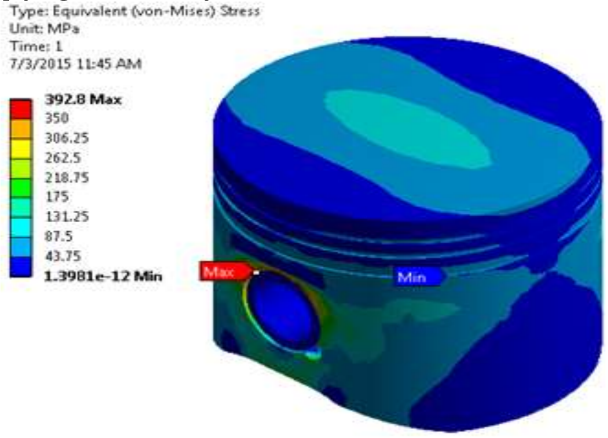


Fig. 8: Equivalent stress for modified piston

From stress tensor after geometric modification of piston we observe that the maximum stress is reduced from 829Mpa to 392.8Mpa. By observing the stress tensor it is clear that stress values over regions of piston are within the allowable yield stress of the material therefore modified piston design is safe.

D. Bi-Linear Static Analysis

Below figures shows the stress tensor obtained when bi-linear static analysis is performed by applying definite boundary conditions.

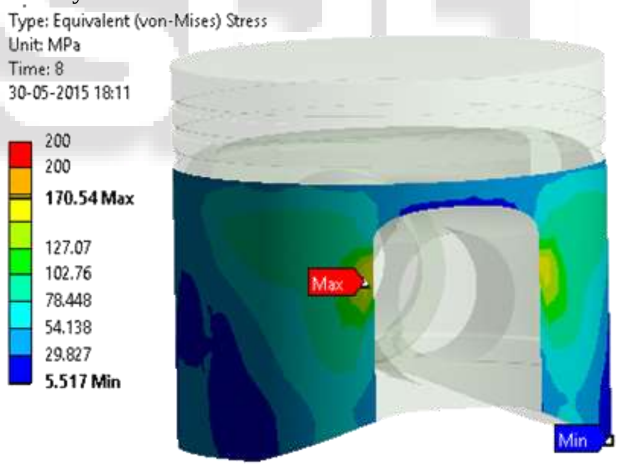


Fig. 9: section stress at skirt

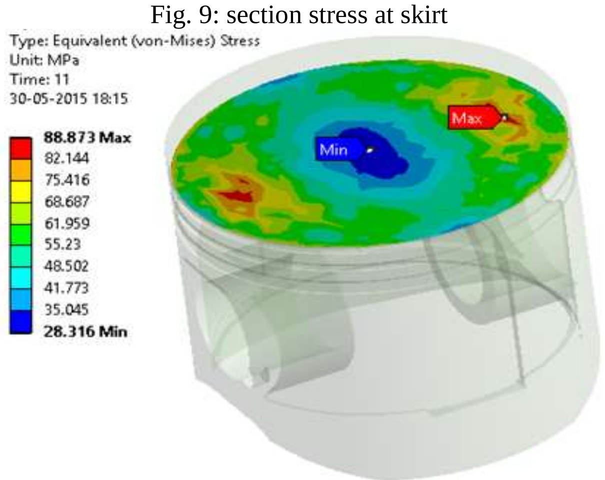


Fig. 10: Section Stress at Piston Head

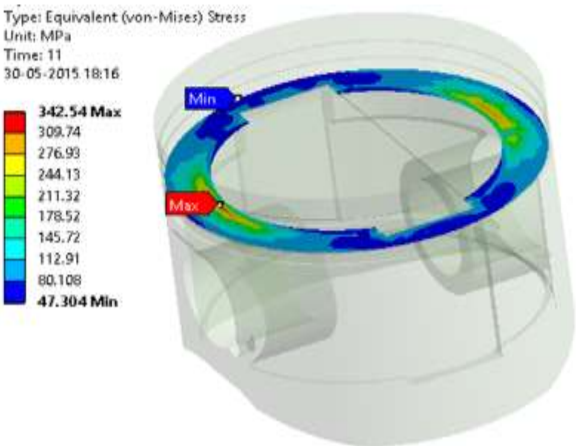


Fig. 11: section stress at ring grove

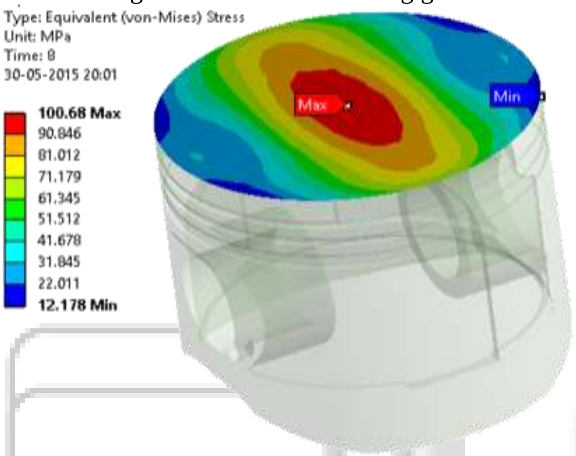


Fig. 12: section stress at top land

By observing the stress tensors, section stress at ring grove is maximum (342.54Mpa) this is because of the presence of sharp edge at ring groves. However yield strength of material is 420Mpa against which I have 342.54Mpa as the section stress, so I still have a margin of 1.33. At different sections of piston, i.e. maximum stresses at skirt (170.54Mpa), maximum stress at piston head (88.873Mpa) and maximum stress at piston top land (100.68Mpa) we have an optimum margin of safety so the design is considered to be safe.

E. Modal Analysis

Table 6 below shows all the six frequency values obtained for six mode shapes in ANSYS.

Tabular Data

	Mode	Frequency [Hz]
1	1.	7293.2
2	2.	7968.4
3	3.	8841.4
4	4.	10795
5	5.	12168
6	6.	12749

Table 6: Frequency values for six modes

Fig.13 shows the Campbell diagram drawn for modal analysis. In which engine speed is constructed on abscissa and frequency values on ordinate. The condition for resonance in Campbell diagram is, all the three parameters such as frequency, speed and modes must coincide at a point. But in this study we verified for modes up to 45X, and we are not come across the resonance. Hence with this we conclude that as per modal analysis the piston design is safe.

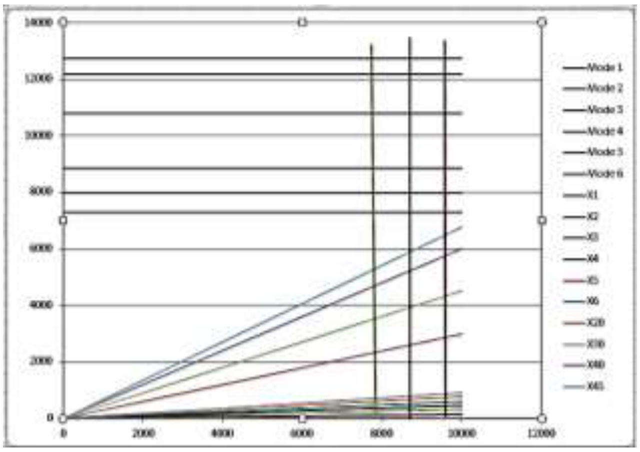


Fig. 13: Campbell diagram

F. Harmonic Analysis

Harmonic analysis is done by using stimulus approach. Here the load considered is 0.16Mpa, i.e. 1% of combustion gas pressure (16Mpa). Below fig.14 shows the finite element model with applied boundary conditions for the harmonic analysis.

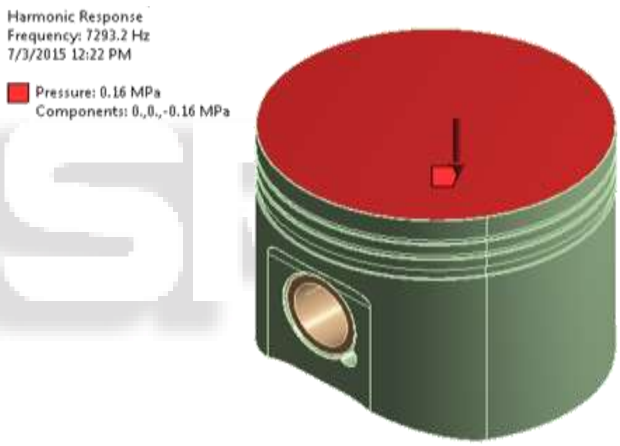


Fig. 14: FEM with boundary conditions for harmonic analysis

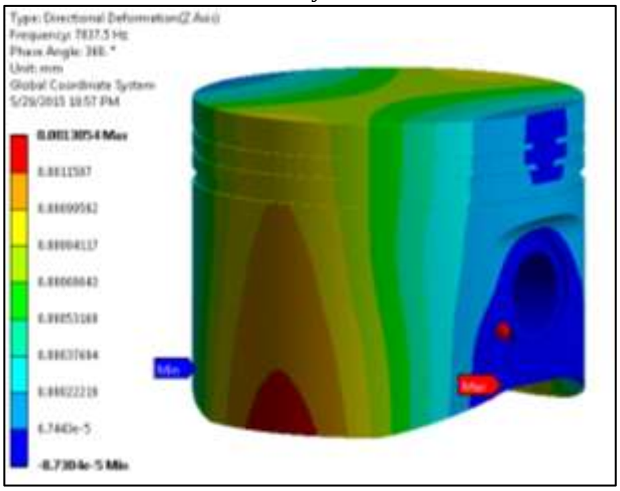


Fig. 15: Directional deformation

By the above deformation plot fig.15 we can observe that, maximum deformation is at the skirt region. This predicts that the piston slap is occurring at the skirt region.

G. Fatigue Analysis

The fig.16 shows the fatigue life of piston obtained by performing fatigue analysis in ANSYS. The fatigue life of piston obtained is $1e^8$ cycles.

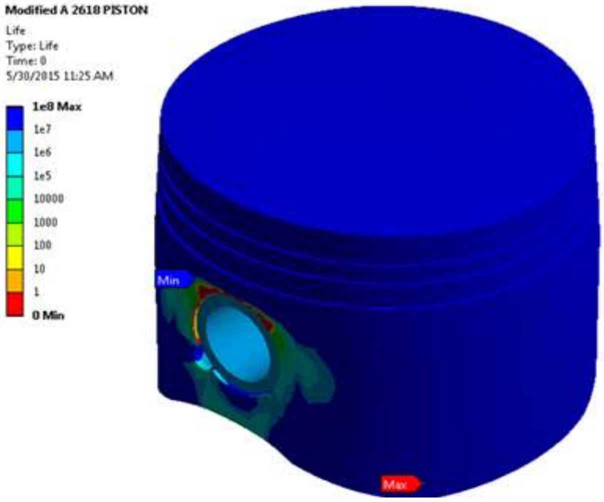


Fig. 16: Fatigue life

H. Multi Body Dynamic Analysis

Finite element model for multi body dynamic analysis of piston is as shown in the fig.17 below. Here the modeling is done in ANSYS.

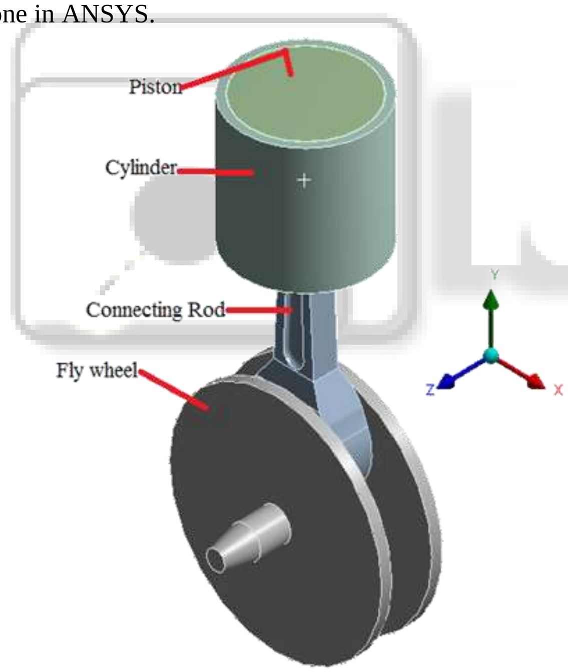


Fig.17: FEM for MBD analysis

Table 7 gives the details about different joints that are defined in MBD analysis.

Type of pair	Type of joint
Ground - Solid (Cylinder)	Fixed
Piston – Cylinder	Translational
Connecting rod – Crank shaft	Revolute
Piston – Cylinder	Translational
Connecting rod – Gudgeon pin	Revolute
Piston – Gudgeon pin	Revolute
Ground – Crank shaft	Revolute

Table 7: Types of joints

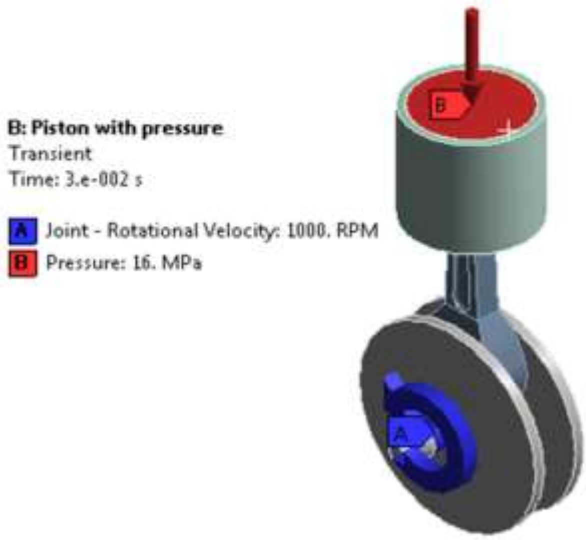
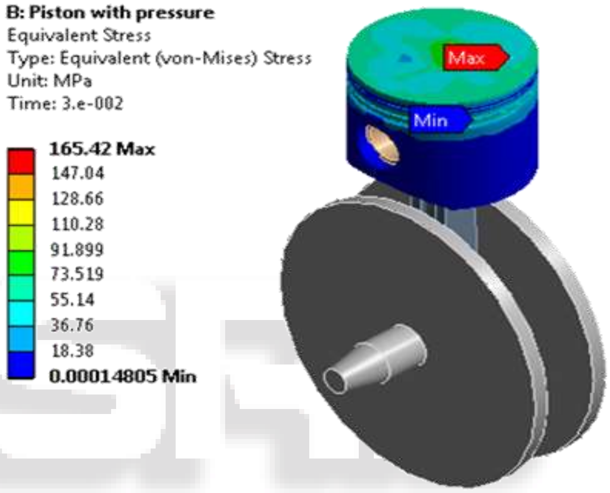


Fig. 18: FEM with boundary conditions



In MBD analysis maximum stress value obtained at piston is 165.42MPa which is less than allowable yield strength of the material. Therefore design of piston is considered to be safe.

V. CONCLUSION

- Static analysis is done on piston and design of baseline model of piston is modified and obtained results well within the permissible values.
- Linear and bilinear analysis is carried out on piston and verified for assumed preliminary design consideration.
- Modal analysis of piston is performed and with the Campbell diagram, design of piston is verified and considered as safe.
- Harmonic analysis is carried away by following modal analysis, using the frequency values for different mode shapes which are obtained in modal analysis and verified that the component is safe at different frequency levels.
- By harmonic analysis prediction of piston slap is done.
- With the prediction of high cycle fatigue life, the life of piston is evaluated and obtained a satisfactory result of 1e8 cycles.

- MBD analysis is carried out and analyzed the results about critical stress regions in the assembly.

VI. Scope of Future Work

- In future, MBD analysis is carried out for higher engine speeds.
- Optimization of piston geometry to reduce noise due to piston slap is done by noise and vibration analysis (NVH).
- In future, different materials are considered for piston design to reduce the weight and to increase the strength.

References

[1] Ch. VenkataRajam et al "Design analysis and optimization of piston using CATIA and ANSYS" volume 1, issue 2, January 2013.
[2] JaumeAngueraLlort"Designing and modeling of piston in combustion engines" 2013-14
[3] BedajangamSagar Kishore and N.P.Jadhav "Optimization of geometrical parameters of four stroke engine piston" volume 4, issue 2, February 2014.
[4] Ajay Raj Singh and Dr. Pushpendra Kumar Sharma "Design, analysis and optimization of three aluminum piston alloys using FEA" volume 4, issue 1, January 2014.
[5] S. N. Kurbet et al "Finite element and MBD analysis of piston to predict the engine noise" volume 2, January 2013.
[6] Conor P McNally "Development of a numerical model of piston secondary motion for internal combustion engines" June 2000.
[7] Jen Fillpczyk and ZbigniewStanik "Piston damages-case studies and possibilities of early detection" volume 19, 2012
[8] Vinay V Kuppast et al "The study on effect of torque on piston lateral motion" volume 2, issue 8, August 2013.
[9] Dr. K. Lingaiah, "Machine Design Data Handbook" volume 1, March 2006.

