EXPLORING ALLOYING ELEMENTS IN AUTOMOTIVE COMPONENTS AND ANALYSIS OF CRANKSHAFT ASSEMBLY USING ANSYS

Hagos Abay[1]*, Prabhakar. S[2]

[1*] Lecturer, Mechanical department, Automotive Engineering Stream, Kombolcha Institute of Technology, WOLLO University, Kombolcha-208, Ethiopia. Email: hagosabay122@gmail.com

2 Associate professor, Mechanical department, Automotive Engineering Stream, Kombolcha Institute of Technology, WOLLO University, Kombolcha-208, Ethiopia. Email: praba.rockson@gmail.com

Abstract

The seminar paper deals with study of an assembly of the Piston, Connecting rod and Crank shaft. It is proposed to replace with two new sets of materials for the components of the assembly and check the parameters by performing the static, dynamic analysis.The assembly of piston connecting rod and crank shaft are done by solidworks. The piston material is aluminum alloy because it has higher thermal conductivity and lighter in weight. Connecting rod is subjected to compressive loads due to the gas force of the piston and the force of crankshaft, therefore we need materials which have higher strength and capable of withstanding this compressive force and mostly titanium is used. In this project, the main parts of the assembly i.e. engine piston, connecting rod and crankshaft are modeleand assembled as per the given design. Structural Analysis is done in Ansys with sizing meshing.

Keywords: Piston; Connecting rod; Crank shaft; static body analysis; meshing; dynamic analysis

Introduction

The present analysis for the piston, connecting rod and crank shaft assembly is done to replace the existing materials with two different sets of materials. This is done as a part of decreasing the weight of the components and to increase the speed of the engine. The components of the assembly are to be rigid and at the same time, the assembly is to perform as a mechanism. This can be done as a rigid body analysis for the components and flexible body analysis for the assembly acting as mechanism.The models are developed from the dimensions provided in the 2D drawing. A detailed Finite Element (FE) model is to be created for this purpose. The FE model consists of the main parts of the Mechanism assembly, and it includes a description of the mechanical loads and the contact interaction between them. The meshing is done using sizing meshing. Static, analysis is done.

1.1 piston

Piston is the important part in an engine which works and produces the result. Piston forms a guide and bearing for the small end of connecting rod. It transmits the force of explosion in the cylinder, to the crank shaft through the connecting rod. Piston transmits the driving force of combustion to the crankshaft and makes the crankshaft to rotate. They also act as a moveable gas-tight plug that keeps the combustion in the cylinder. It has to get rid of the heat from combustion.

1.1.1 Materials for the Piston

Cast Iron, Aluminum Alloy and Cast Steel etc. are the common materials used for piston of an Internal Combustion Engine. Cast Iron pistons are not suitable for high speed engines due its more weight. The Aluminum Alloy Piston is lighter in weight and enables much lower running temperatures due to its higher thermal conductivity. The coefficient of expansion of this type of piston is about 20% less than that of pure aluminum piston but higher than that of cast iron piston. It is easily machinable, light in weight and has good thermal conductivity (can transfer heat quickly).

1.2 Connecting rod

The connecting rod links the piston to the crankshaft. The upper end has a hole in it for the piston wrist pin and the lower end (big end) attaches to the crankshaft. It is under tremendous stress from the reciprocating load of the piston. With every rotation, it is stretched and compressed and the load increases to the third power with increasing engine speed.

1.2.1Materials used for Connecting Rod

Steel is normally used for construction of automobile connecting rods because of its strength, durability, and lower cost. But, they have high mass density and make the crank shaft heavy. This limits the speeds of the engine.

1.3 Crankshaft

The crankshaft converts the up and down (reciprocating) motion of the pistons into a turning (rotary) motion. It provides the turning motion for the wheels. The crankshaft is connected to the pistons by the connecting-rods.

1.3.1 Materials used for Crankshaft

The crankshaft is made usually either with alloy steel or cast iron.

2. Theoretical frame work

2.1. Analysis procedure methodology

Before starting analysis it should follow some procedures starting from the study of the material.

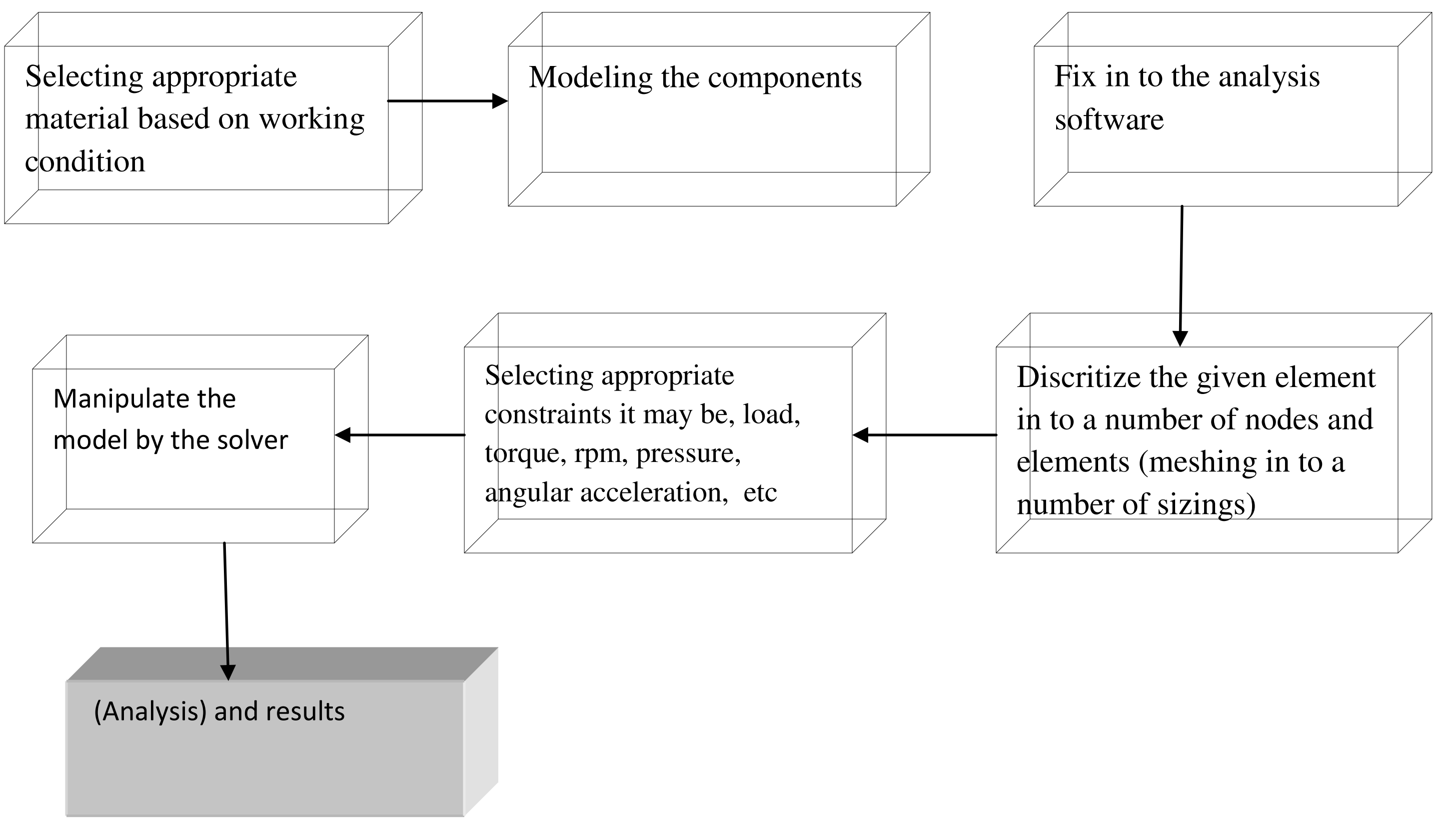


Figure 1: analysis procedure

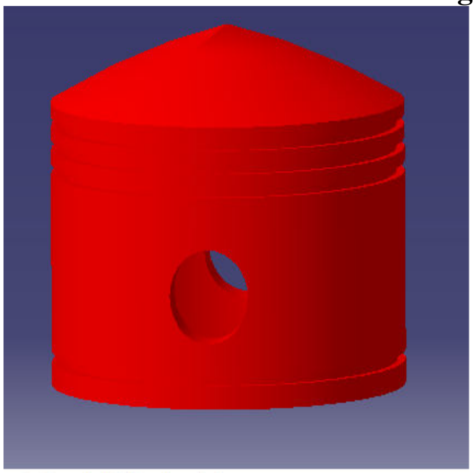


fig 1: catia 3D drawing of piston

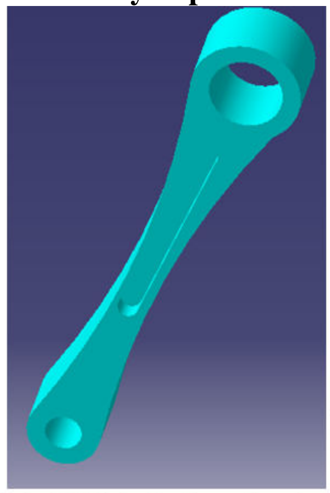


fig 2 : Catia 3D drawing of connecting rod

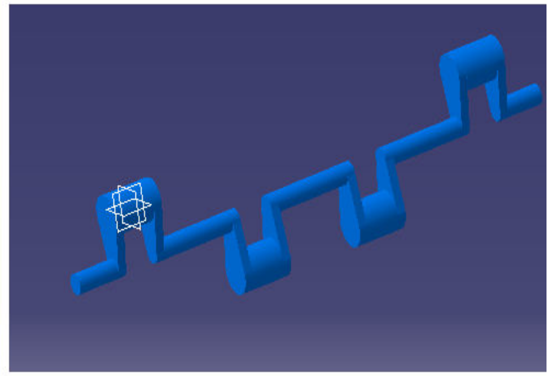


fig 3 : Catia 3D drawing of crankshaft

Figure 2: part drawing of piston, connecting rod and crankshaft

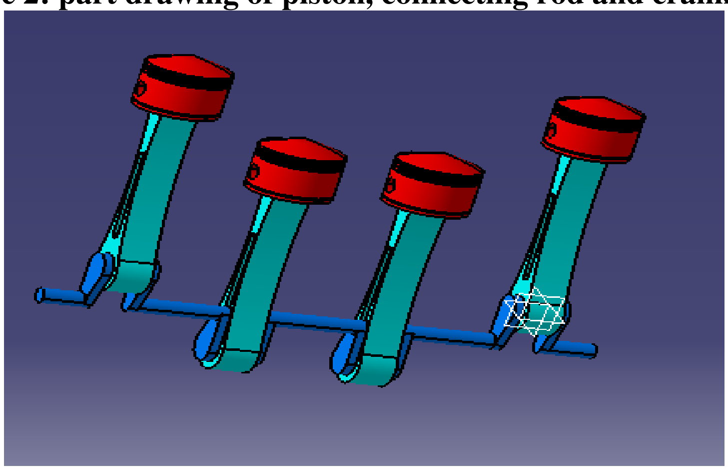


Figure 3: assembly drawing of piston, connecting rod and crankshaft

Modeling of the component is done from machine element book with their dimension [1].

2.2 Introduction to FEA

FEA is a computer model, of a material or design either new design or existing product refinement, that is stressed and analyzed for specific results. It is used in to represent the numerous algorithms (functions) in linear and non linear systems. Linear systems do not take into account plastic deformation and Non-linear systems consider the plastic deformation and are capable of testing a material up to fracture. Many FEA programs also are equipped with the capability to use multiple materials within the structure [8]. [5] discussed a project, Proton Exchange Membrane (PEM) energy unit are progressively being refered to by governments as a conceivable pathway to the decrease of ozone depleting substance outflow. It is one of the forthcoming force hotspots for car applications, prepare machines, stationary cogeneration frameworks, and portable electronic gadgets. Be that as it may, the dryness of the film of a PEM power device diminishes the ionic conductivity, bringing about execution decrease. In this work, a two-dimensional model is utilized to examine the fundamental and collaboration impacts of five outline factors, at three levels in a proton trade layer (PEM) energy unit. Investigation is directed for working possibilities of 0.7 and 0.6V and a scope of current densities. An engine that picks up its energy from a hydrogen tank and a power device Stored in a tank. The substance vitality from the hydrogen will be changed over into electrical vitality by the power device to push the prepare at up to most extreme speed of 80km/hr. Prepare apparatuses like Fans, lighting may likewise keep running on PEM energy unit. This new hydrogen prepare is along these lines ideal for shorter, calmer extends of the system that jolt hasn't yet come to.

The model is made into number of elements and nodes at the elements. The nodes make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Depending upon the stresses, the node density is assigned. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes.

2.3. Objective functions available:

Mass, volume, temperature, Strain energy, stress, strain, Force, displacement, velocity, acceleration, etc

The loading conditions which may be applied to a system are:

- ✓ Point, pressure, thermal, gravity, and centrifugal static loads
- ✓ Thermal loads from solution of heat transfer analysis
- ✓ Heat flux and convection
- ✓ Point, pressure and gravity dynamic loads

Meshing in sizing mesh

The meshing of the assembly is done in hyper mesh as shown in fig 4 *Assigning material properties to the piston assembly [7-9].*

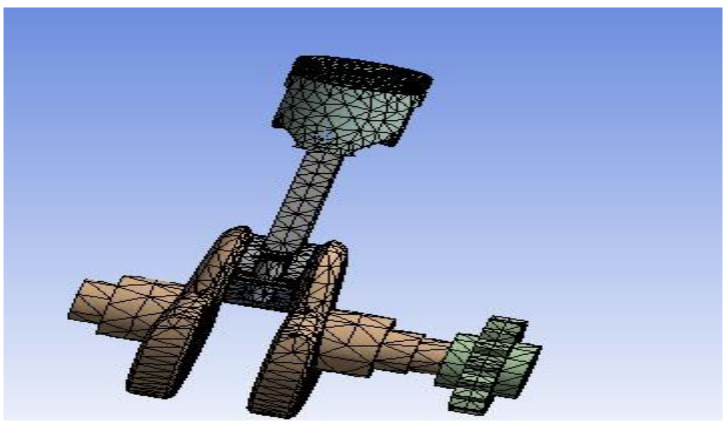


Figure 4: Meshing of the Assembly

2.4. Introduction to ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all.

These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand [8].

2.3.1 Specific capabilities of ANSYS

Structural - It the most common application of the finite element method used for bridges, buildings, machine tools, components, etc.

Static Analysis - Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses.

Transient Dynamic Analysis - Used to determine the response of a structure to arbitrarily time-varying loads.

All nonlinearities mentioned under Static Analysis above are allowed.

Buckling Analysis - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigen value) buckling and nonlinear buckling analyses are possible.

In addition to the above analysis types, several special-purpose features are available such as Fracture mechanics, Composite material analysis, Fatigue, and both p-Method and Beam analyses.

Thermal - ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions.

Fluid Flow - Modeling a vast range of analysis types such as: airfoils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, 3 D flow patterns in a pipe bend.

Coupled Fields - A coupled-field analysis is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of engineering. Examples of coupled-field analysis are thermal-stress analysis, thermal-electric analysis, and fluid-structure analysis.

Modal Analysis - A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed.

Harmonic Analysis - Rotating machinery is subjected to Harmonic analysis to predict the sustained dynamic behavior of structures to consistent cyclic loading. It is done to verify whether or not a machine design will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations [3-8].

3. Analysis and Results

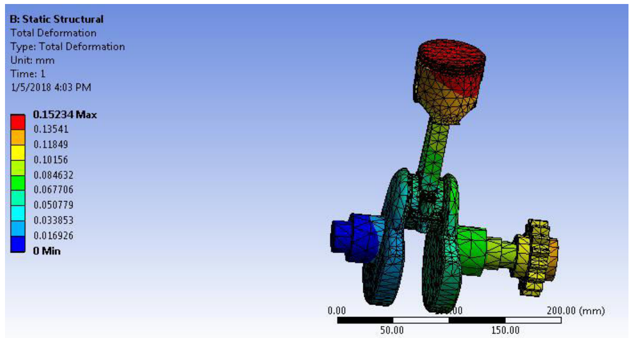
3.1. Static Analysis of piston, connecting rod and crankshaft assembly

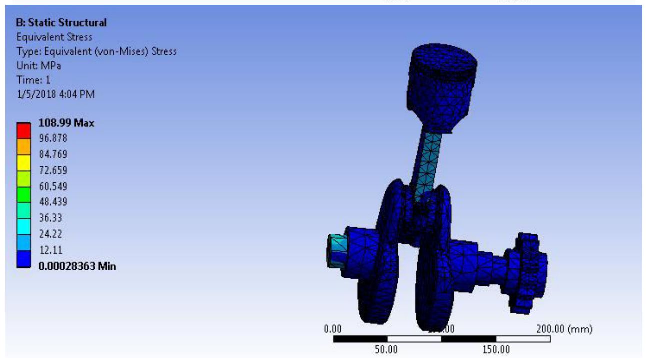
3.1.1 Piston (Aluminum alloy), Connecting rod (Al alloy) & Crankshaft (EN308)

The static analysis of the piston, connecting rod and crankshaft assembly is done by applying the material with properties in table1 and the results are shown in fig 5.

Table 1: Material properties:

Component material	Young’s modulus	Poison’s ratio	Density(kg/mm^3)
Piston – Al - alloy	68900	0.29	0.0000027
Conn.rod- Al- alloy	68900	0.29	0.0000027
Cr.shaft- alloy steel	205000	0.25	0.00000785





Deformation stress

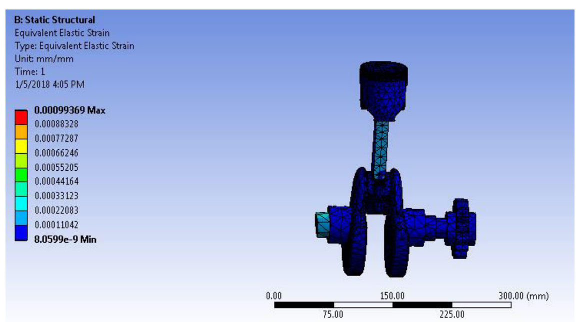


Figure 5: Output of Displacement, Stress and Strain

3.1.2 Static analysis of Piston (Aluminum alloy), Connecting rod (Titanium) and Crankshaft (HighAlloy steel)

The static analysis of the piston, connecting rod and crankshaft assembly is done by applying the other material with properties in table2 and the results are shown below [5-7].

Table 2: Material properties

Component material	Young's modulus	Poison's ratio	Density(kg/mm^3)
Piston – Al - alloy	71000	0.33	0.0000027
Conn.rod- Al- alloy	116000	0.4	0.0000045
Cr.shaft- alloy steel	207000	0.29	0.00000785

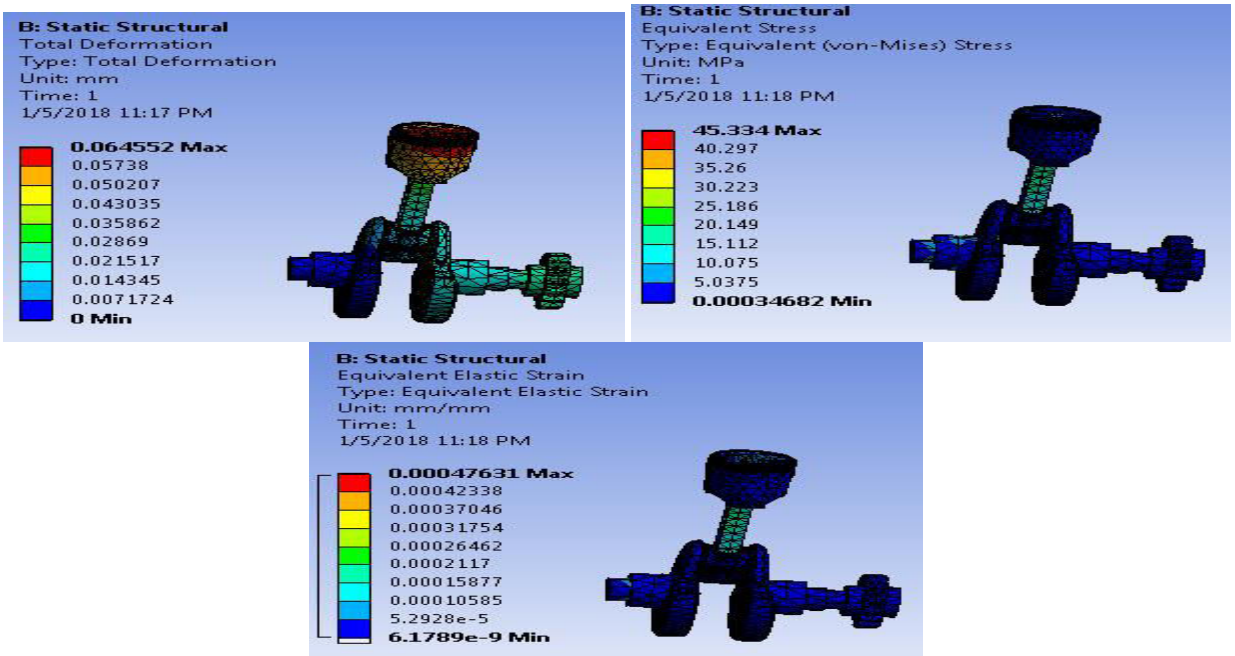


Figure 6: Output of Displacement, Stress and Strain

3.2 Comparison of the results

Table 3: comparison of static analysis

	Displacement(mm)		Von Mises stress(N/mm^2)	strain
Al - alloy	0.1523	Piston –Al alloy Connecting rod- Crankshaft- High alloy steel	108.99	0.00088328
Connecting rod – titanium	0.05738	Piston – Al alloy Crank shaft- high alloy steel	45.334	0.00047631

Static, analysis is done on the assembly of piston, connecting rod and crankshaft.

The materials considered for piston are Aluminum alloy and Aluminum alloy, for connecting rod are Aluminum alloy and Titanium and for crankshaft are EN308 and High Alloy steel

By observing the static analysis results,

- ✓ The stresses are increased in the piston AL alloy , connecting rod Al- alloy and high alloy steel crankshaft Piston, Titanium Connecting rod and High Alloy Steel

4. Conclusions

The assembly of piston connecting rod and crank shaft are done by solidworks. The piston material is aluminum alloy because it has higher thermal conductivity and lighter in weight. Connecting rod is subjected to compressive loads due to the gas force of the piston and the force of crankshaft, therefore we need materials which have higher strength and capable of withstanding this compressive force and mostly titanium is used.

Generally, Ansys have the capability of analyzing different conditions for any element including the fracture of the elements.

In static analysis of piston with aluminum alloy, connecting rod with titanium and crankshaft with high alloy steel have low stresses created during analyzing and it increases fatigue life of the connecting rod lower cap and crankshaft web which is makes contact during dynamic motion in the component interest of connecting rod.

5. References

[1] Goenka, P. K. and Oh, K. P., 1986, “An Optimum Connecting Rod Design Study – A Lubrication Viewpoint,” Journal of Tribology, Transactions of ASME, July 1986, Vol. 108.
[2] A.Prem kumar, DESIGN & ANALYSIS OF CONNECTING ROD BY COMPOSITE MATERIAL, IJRDO, International-Journal-Of- Mechanical-And-Civil-Engineering , VOL 2 ISSUE 7 July 2015 Paper 1.
[3] Montazersadgh, F. H. and Fatemi, A., 2007, “Dynamic Load and Stress Analysis of a Crankshaft,” SAE Technical 2007, Society ofAutomotive Engineers, Warrendale, PA, USA.
[4] Prof. Vivek C. Pathade1 , Dr. Dilip S. Ingole, “Stress Analysis of I.C.Engine Connecting Rod by FEM and Photoelasticity”, IOSR Journal of Mechanical and Civil Engineering, Volume 6, Issue 1 (Mar. - Apr. 2013), PP 117-125.
[5] Christo Ananth, "Analysis and Optimization of PEM fuel cells Design", Smashwords Publishing, Los Gatos, California, United States, ISBN: 978-81-910-749-7-0, September 2017, pp: 1-36.
[6] Text book on Machine Design by R.S. Khurmi.
[7] Bhandari, V. B., 1994, “Design of Machine Elements,” Tata McGraw-Hill.
[8] HAUG, E.J., WEHAGE, R.A., and BARMAN, N.C., 1982, “Dynamic Analysis and Design of Constrained Mechanical Systems”, Journal of Mechanical Design