

MODELING AND STATIC ANALYSIS OF SINGLE CYLINDER PETROL ENGINE PISTON

A V Hari Babu^{1*}, G Pandu Ranga Swamy², D R Subba Rao³, M Sudhakar Reddy⁴, S Ramesh Kumar Babu⁵,

1. Associate Professor & Head of ME, SVR Engg College, Nandyal.

2, 3, 4, 5 Assistant Professor, SVR Engg College, Nandyal

*Corresponding Author: A V Hari Babu,

When the combustion of fuel takes place in petrol engine cylinder, high temperature and pressure will be developed because the engine will run at high speed and at high loads. Generally piston is made of Al Alloy which is a crucial part in internal combustion engines. This report deals the modeling and analysis of petrol engine piston is done by using cast iron and compositions of aluminum alloy. Initially structural analysis was performed on cast iron, Al Alloy.

The objective of this project report is to model and analyze the petrol engine piston. In this project the geometric model of piston is generated in CATIA V5 software. The geometrical model is cleaned and meshed in to finite elements by using HyperMesh12 software. Finite Element model was solved using ABAQUS software for stress and displacement as per the calculated load.

Keywords: single cylinder petrol engine piston, CATIA, HYPERMESH and ABAQUS.

INTRODUCTION

A piston is a cylindrical engine component that slides back and forth in the cylinder bore by forces produced during the combustion process. The piston acts as a movable end of the combustion chamber. The stationary end of the combustion chamber is the cylinder head. Pistons are commonly made of a cast aluminum alloy for excellent and light weight thermal conductivity is the ability of a material to conduct and transfer heat. Aluminum expands when heated and proper clearance must be provided to maintain free piston movements in the cylinder bore. Insufficient clearance can cause the piston to seize in the cylinder. Excessive clearance can cause a loss of compression and an increase in piston noise. A complete illustration of that piston and its parts are shown below.

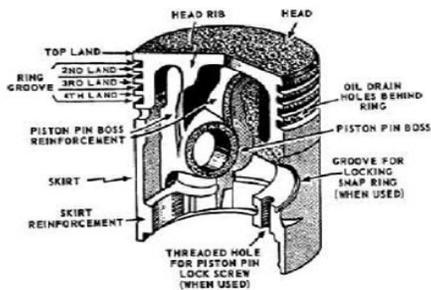


Fig 1.1 Piston feature

Piston features include the piston head, piston pin bore, skirt ring grooves, ring lands, and piston rings. The piston head is the top surface (close to the cylinder head) of the piston which is subjected to tremendous forces and heat during normal engine operation.

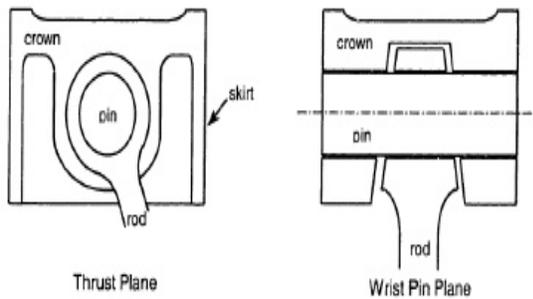
1.1 Type of pistons

There are essentially two types of piston used in today's automotive engine.

1.1.1 Mono piston

By far the most common, and the one used in all passenger car engines, is the mono piston. As the name suggests, the piston comprises of a single component, which is usually made from aluminium. The upper part of the piston that supports the combustion force and holds the piston rings is called the **Crown**. The lower part of the piston that supports the lateral forces against the inner walls is called **Skirt** the piston is linked to the connecting rod via the **wrist pin** on which both components are hinged. An example of a mono piston assembly can be seen in figure in below

Fig 1.2 mono piston assembly



1.1.2 Articulated piston

In some heavy duty diesel engines, due to extremely high combustion temperatures and pressures, it is desirable to have a stainless steel crown section. An aluminum skirt is still preferable due to its low weight and elasticity. Two components of different materials cannot be rigidly joined together in the position, as their differing coefficients of expansion would lead to failure. Instead, articulated pistons comprise a stainless steel crown and an aluminum skirt which are separate components and are hinged separately on the wrist pin. An example of an articulated piston assembly can be shown in figure

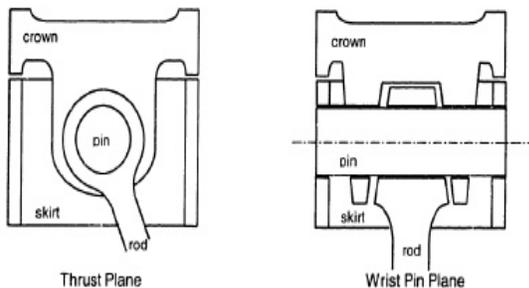


Fig 1.3- articulated piston assembly

The piston motion equations are traditional Newton's law of vertical and lateral motion and rotation applied to the piston in standard manner the forces acting on a piston are exemplified in figure these forces and movements come from interactions of the piston with the liner, rings, wristpin, cylinder pressure and inertia.

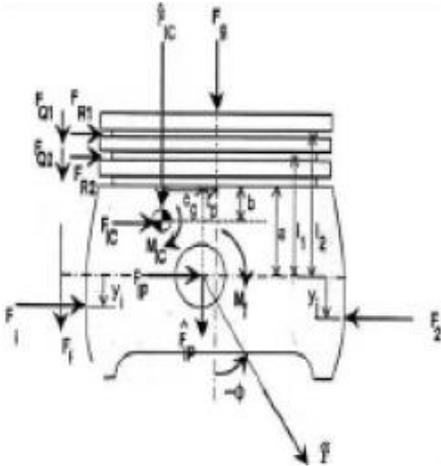


Fig 1.4-forces & movements acting on the piston

2. DESIGN OF PISTION

Design considerations for a piston

Piston should have enormous strength to withstand high gas pressure and inertia forces in addition to disperse the heat of combustion quickly to the cylinder walls. It should provide sufficient bearing area to prevent undue wear in addition to having sufficient support for the piston pin. And also it should have sufficient rigid construction to withstand thermal and mechanical distortion in addition to it acts as effective sealing to the gases not to escaping from combustion chamber.

Engine specification

The Engine selected for this work is a single cylinder four stroke air cooled 100cc proto type petrol engine. The main parameter shows in the table below **table 1**

Engine	Type	4-stroke,air cooled, single cylinder
	Number of cylinders	Single cylinder
	Bore	41 mm
	Stroke	75 mm
	Length of connecting rod	150 mm
	Compression ratio	8.4
	Fuel consumption	87 kmpl (under standard condition)
Performance	Maximum power	6.03kw@7500 rpm
	Maximum torque	8.05n-m@5500 rpm
Transmission	Clutch	Wet multi disc type
	Transmission	4-speed,constant mesh type
	Gear shift pattern	All down foot shift
Dimensions	Overall length	1925mm
	Overall height	1030mm
	Wheel base	1215mm
	Ground clearance	150mm
	Kerb weight with full tank	109kg
Capacity	Fuel tank	9.3 liters including 2.2liters reserve

Table 1: Engine specification

s/no	Description	Nomenclature	Value in mm
1	Thickness of piston head	t_H	7
2	Radial width of piston ring	t_1	2
3	Axial thickness of piston ring	t_2	1
4	Width of top land	b_1	4
5	Width of ring land	b_2	2
6	Thickness of piston barrel at the top end	t_3	5
7	Length of skirt	L_s	18
8	Length of piston pin in the connecting rod bushing	l_1	20
9	Piston pin diameter	d_0	10
10	Radial width of oil ring	t_4	3
11	Axial thickness of piston	t_5	2.5

Table 2 Piston specification

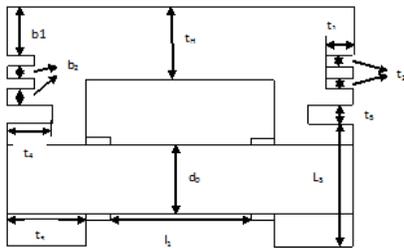


Fig: Piston cut section

CATIA V5 Workbenches

CATIA V5 serves the basic design tasks by providing different workbenches. A workbench is defined as a specified environment consisting of a set of tools that allows the user to perform specific design task. The basic workbenches in CATIA -V5 are

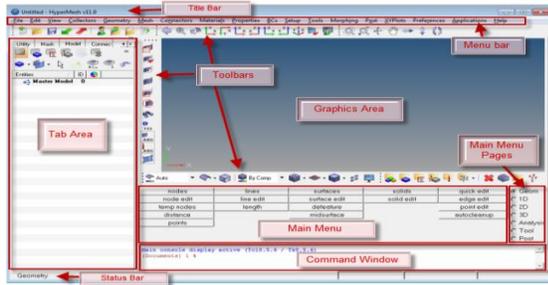
1. Part design
2. Wire frame
3. Surface design
4. Assembly design
5. Drafting
6. Generative sheet metal design
7. DMU Kinematics

Getting Started with HyperMesh

In this tutorial, you will explore the basic concepts of the user interface of Hyper Mesh.

Tools

The Hyper Mesh interface contains several areas. Each is described below.



Hyper Mesh Graphical Interface

Starting Hyper Mesh

To start Hyper Mesh on a PC, go to Start > Programs > Altair Hyper Works > Altair Hyper Mesh

To start HyperMesh on UNIX, perform the following steps:

- Go to your operating system prompt.
- Enter the full path of the Hyper Mesh script (e.g., <altair_home>\altair\scripts\hm) and press the ENTER key.

Or

- Type in a pre-defined alias that you or a systems administrator has created in the user .alias or .cshrc file in the user home directory.

Start Directory

By default, Hyper Mesh uses a "start directory" for files. Hyper Mesh reads and writes a number of files from the start

Directory:

- At start up, Hyper Mesh reads configuration files (hm.mac, hmmenu.set, etc.).
- Upon closing, Hyper Mesh writes out a command history file (command.cmf) and a menu settings file (hmmenu.set).
- By default, Hyper Mesh will read from/write to this directory for any open, save, save as, import, or export functionality.
- Image files (.jpg) created using the F6 key are saved to the start directory.

To determine the start directory on Windows, perform the following steps:

1. Right-click the Hyper Mesh icon.
2. Go to **Properties**.
3. On the **Shortcut** tab, view the path in the **Start In** field.

On UNIX, the start directory is determined by the following:

- Location in which you typed the command to run Hyper Mesh
- Your "home" directory if configuration files are not found in the start directory

HyperMesh Help

To obtain help for a particular feature, go to the **Help** menu and select **Hyper Mesh and Batch Mesher**. The help is organized by product and contains the following types of information:

- How to use individual functions
- Notes on interfacing Hyper Mesh with external data types
- Tutorials
- Programming guides

Model Files

All files referenced in the Hyper Mesh tutorials are located in the <install_directory>\tutorials\hm\ directory unless otherwise noted.

Opening and Saving Files

In this tutorial, you will learn how to:

- Open a Hyper Mesh file
- Import a file into a current Hyper Mesh session
- Save the Hyper Mesh session as a Hyper Mesh model file
- Export all the geometry to an IGES file
- Export all the mesh data to an OptiStruct input file
- Delete all data from the current Hyper Mesh session
- Import an IGES file
- Import an OptiStruct file to the current Hyper Mesh session

Starting ABAQUS

When you create a model and analyze it, ABAQUS/CAE generates a set of files containing the definition of your model, the analysis input, and the results of the analysis. In addition, ABAQUS/CAE and ABAQUS/Viewer generate replay files that reflect all your interactions with the application. Consequently, before you run either product, you should move to a directory where you have permission to create files.

FINITE ELEMENTS ANALYSIS

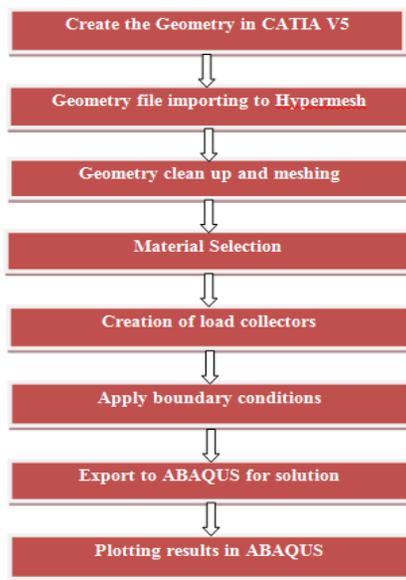
Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze elastic deformation or permanently bent out of shape plastic deformation. The computer is required because of the astronomical number of calculations needed to analyze a large structure. The power and low cost of modern computers has made finite element analysis available to many disciplines and companies.

Failure Modes

1. Static loads lead to stresses exceeding yield over a significant region.
2. Loads on bolts, rivets, spot-welds, stitch-welds, fillet-welds, bevel-welds, full penetration-welds, adhesives, nails, tie-rods, links or other connection devices are too high.
3. Strains reach fracture levels in brittle materials.
4. Surface strains cause damage to protective coatings.
5. Buckling of components leads to local damage or progressive collapse.
6. Combined bending and compression leads to excessive stress and failure.
7. Fatigue or sudden fracture is reached.
8. Vibration frequencies are located where applied loading causes damage

PROCESS METHODOLOGY

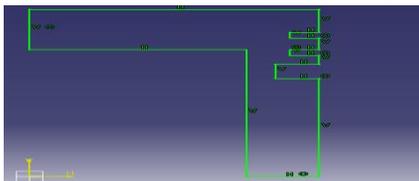
The geometrical model has been created in CATIA V6. 3D model of piston is imported into the HyperMesh for preprocessing. Preprocessing of model consist of meshing, selection of material properties, creation of load collectors and apply boundary conditions on model. Then model is exported to ABAQUS for solving problem. Results of solution plotted in ABAQUS which is well known postprocessor of Hyper Works software



Methodology flow chart

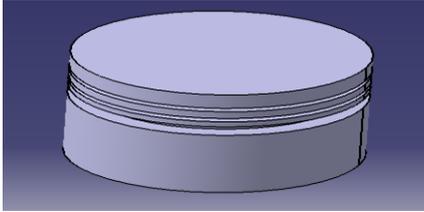
Creating geometry in CATIA

1. Open CATIA V5 Software.
2. Go to start, select Mechanical Design and select Part Design.
3. Select one plane from specification tree and select sketcher.
4. Draw the cross sectional area of the piston as show in the following figure.



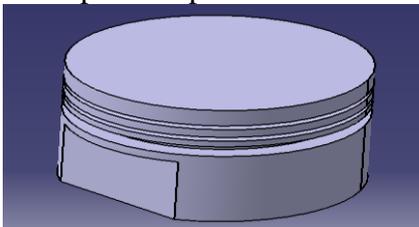
Piston cross section view

5. Give the constraints (dimensions) as per the piston specification table and exit from the sketching workbench.
6. Select shaft tool for to revolve the sketch in 360° about its vertical axis.



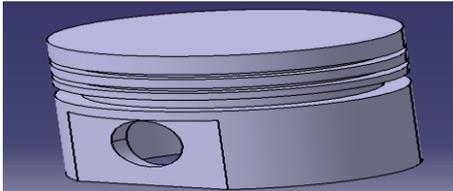
Revolved piston isotropic view

7. Create a plane by selection offset from plane option
8. Draw a rectangle by selecting newly created plane and exit from sketcher
9. Select pocket option and select that rectangle box with a depth of 3 mm.



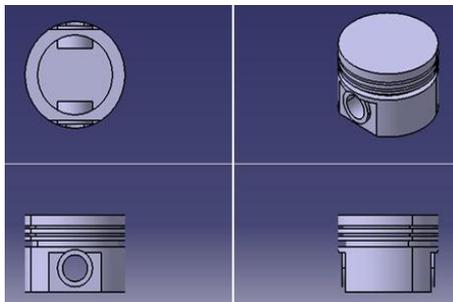
Rectangle piston isotropic view

10. Create one hole with diameter of 10 mm on the piston as shown in the following figure



Piston pin hole isotropic view

11. Draw two circles with diameter 14 mm and 10 mm and pad it.
12. Select pocket-1, pocket-2 and pad-1 from specification tree then click on mirror tool by selecting mid plane as a reference for to mirror the object.
13. Save the model with name Piston



Piston views

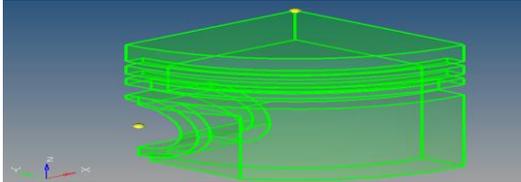
Geometry file importing to Hypermesh

1. Open HyperMesh software select ABAQUS profile

2. Go to import option select geometry, browse piston geometry file which is created in CATIA
3. Click on solid shape option and topo
4. Save the file with name piston

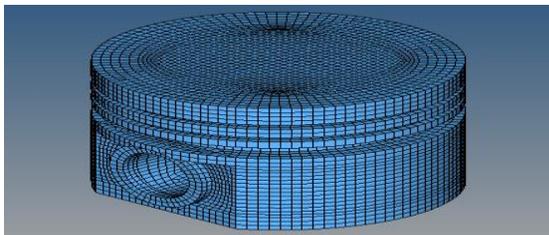
Geometry clean up and meshing

1. Create a centre node with arc option by selecting piston outer arc
2. Cut the piston into four quarter part by selecting trim with plane option
3. Delete three quarter parts and you will get a quarter part of a piston as shown in following figure



Piston quarter cut

4. Trim the geometry by using surface option and by line sweep option so that we can get six face volume blocks
5. Go to 3-d page select solid map, select one volume option give the element size as 1 pick the six face volume click on mesh
6. Repeat step 5 till whole quarter part of piston get meshed
7. A mapped mesh of mixed elements of 8 node hexahedral and 6 node prism elements will appear
8. Go to tool page select reflect option
9. Reflect all elements one time about vertical axis and second time with horizontal axis
10. Select edge option from tool page Give connectivity between all elements by choosing equivalence option
11. A complete meshed model shown in following figure



Complete Mapped mesh model

12. Save the file with name complete mesh

This FEA model consist of mixed elements of 8 node hexahedral, 6 node prism, 4 node quad element and 3 node tria elements

S. no	Name of the element	No.of elements
1	Hexahedral 8 node	22576
2	Prism 6 node (Penta 6	268

	node)	
3	Quad 4 node	1632
4	Tria 3 node	8
	Total	24484

Table: 4 No of Elements in FEA modal

Material selection

In this report three materials are selected for to do a comparison for to get best material selection for the piston of a proto type petrol engine. Those three materials are

1. Aluminium alloy Al-2681
2. Cast iron

1. Aluminium alloy Al-2681

Aluminium / Aluminum alloys have high ductility and corrosion resistance. At subzero temperatures, their strength can be increased. However, their strength can be reduced at high temperatures of about 200-250°C. Aluminium / Aluminum 2618 alloy is an age harden able alloy containing magnesium and copper.

The following table shows the chemical composition of Aluminium / Aluminum 2618 alloy.

Element	Contents (%)
Aluminium / Aluminum, Al	93.7
Copper, Cu	2.30
Magnesium, Mg	1.60
Iron, Fe	1.1
Nickel, Ni	1.0
Silicon, Si	0.18
Titanium, Ti	0.07

Table: 5 Chemical compositions of AL-2681

The physical properties of the Aluminium / Aluminum 2618 alloy are tabulated below.⁷

Properties	Metric
Density	2.768 g/cm ³
Melting point	510 ⁰ C

Table: 6 Physical properties of AL-2681

The mechanical properties of the Aluminium / Aluminum 2618 alloy are outlined in the following table.⁸

Properties	Metric
Ultimate strength	440 Mpa
Yield strength	370 Mpa

Shear strength	260 Mpa
Fatigue strength	125 Mpa
Elastic modulus	70-80 GPa
Poisson's ratio	0.33
Elongation	10%

Table: 7 Mechanical properties of AL-2681

2. Cast iron

An alloy of iron that contains 2 to 4 percent carbon, along with varying amounts of silicon and manganese and traces of impurities such as sulfur and phosphorus. It is made by reducing iron ore in a blast furnace. The liquid iron is cast, or poured and hardened, into crude ingots called pigs, and the pigs are subsequently re-melted along with scrap and alloying elements in cupola furnaces and recast into molds for producing a variety of products.

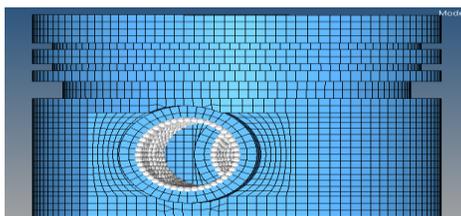
Most cast iron is either so-called gray iron or white iron, the colours shown by fracture. Gray iron contains more silicon and is less hard and more machinable than white iron

Properties	Metric
Density	6.8 gm/cm ³
Young's modulus	110 Gpa
Melting point	1090°C
Ultimate strength	350 Mpa
Yield strength	240 Mpa

Table: 8 Mechanical properties of Cast iron

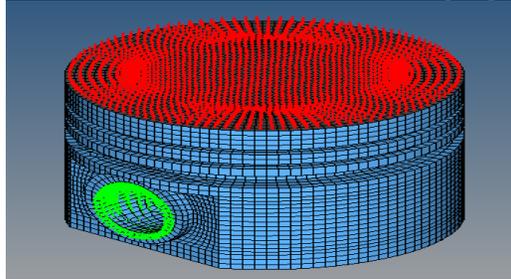
Creating load collectors and applying boundary condition

1. Create load collector by using load collector option from tool bars
2. Give name as fixed, select colour green and image card as initial condition
3. Go to analysis page select constraints select all node at the pin hole of the piston as shown in following figure



4. Select all DOF click create
5. Create load collector by using load collector option from tool bars
6. Give name as pressure, select colour red and image card as history condition

7. Go to analysis page select pressure, select elements top surface at the piston head and give magnitude as 12.5 MPa, shown in following figure



8. Go to analysis page select load steps
9. Give name as static
10. Click on load cols select fixed and pressure
11. Click on create and also on edit
12. Select step parameters, perturbation, analysis processor static, load_op_options, Boundary_op and D load_op select return
13. Save the file

Exporting to ABAQUS

1. Select export option from tool bar
2. Select export solver deck
3. Set path and name to the file with extension of .inp
4. Click export
5. Open ABAQUS and set working directory
6. Import Hyper Mesh export file and delete model-1
7. Right click on job and select create job-1
8. Right click on job-1 and select submit
9. Wait till it complete the solving after that save the model

Plotting results in ABAQUS

1. After completing the solving right click on job-1 select results
2. Select plot contours on deformation option from tool bar
3. Go to field output and select stress, select Misess
4. Go to field output and select stress, select Max-principle.
5. Go to field output and select spatial displacement at nodes, select magnitude
6. Save the results and exit

RESULTS AND DISCUSSIONS:

The following FEA results are got from ABAQUS software, when the piston head experience 12.5 MPa pressure inside the cylinder.

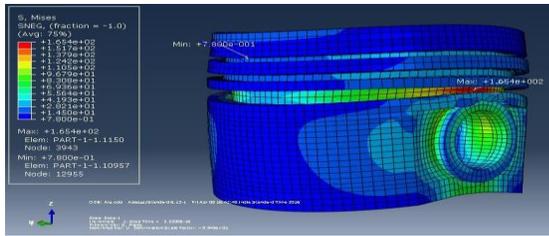


Fig: 1 Von-Mises stress distribution in Al-2618 material piston

The above figure 1 shows Mises stress distribution in an Al-2618 material piston. The maximum stress is 165.4 MPa and minimum stress is 0.78 MPa,

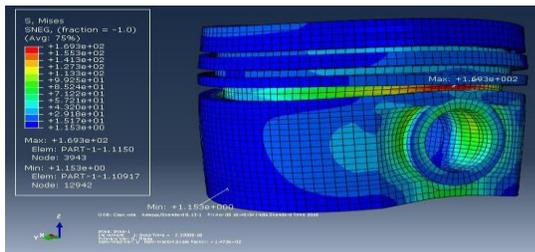


Fig:2 Von-Mises stress distribution in Cast iron material piston

The above figure 2 shows Mises stress distribution in a Cast iron material piston. The maximum stress is 169.3 MPa and minimum stress is 1.153 MPa,

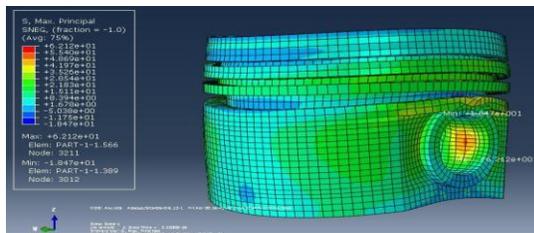


Fig: 3 Maximum principle stress distribution in Al-2618 material piston

The above figure 3 shows Maximum principle stress distribution in an Al-2618 material piston. The maximum stress is 62.12 MPa and minimum stress is -18.47 MPa,.

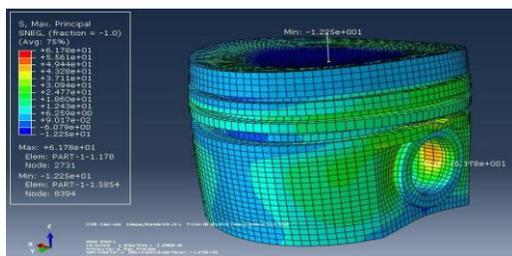


Fig: 4 Maximum principle stress distribution in Cast iron material piston

The above figure 4 shows Maximum principle stress distribution in a Cast iron material piston. The maximum stress is 61.78 MPa and minimum stress is -12.25 Mpa. As comparing the

AL-2618 material the maximum principle stress value is decreased in cast iron material by 0.35 MPa

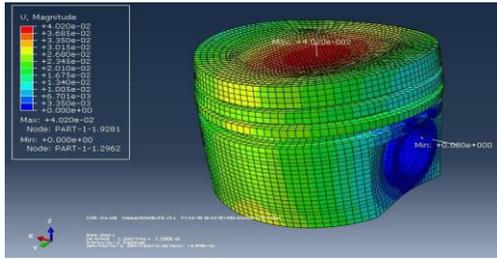


Fig: 5 Deformation in Al-2618 material piston

The above figure 5 shows deformation in an Al-2618 material piston. The maximum deformation is 0.0402 mm,

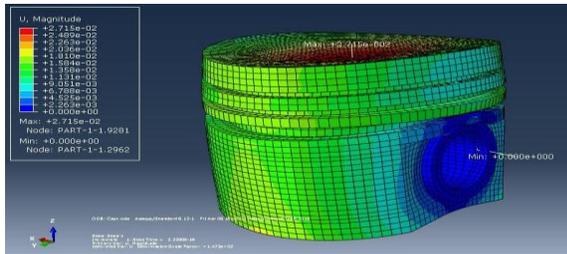


Fig: 6 Deformation in Cast iron material piston

The above figure 6 shows deformation in a Cast iron material piston. The maximum deformation is 0.02715 mm. As comparing the AL-2618 material the maximum deformation value is decreased in cast iron material by 0.01305 mm.

The following table shows the result comparison between two materials

s.no	Material name	Young's modulus (MPa)	Poisson's ratio	Density (g/mm ³)	Mass (Gms)	Von Mises Stress (MPa)	Max-Principle stress (MPa)	Deformation(mm)
1	Al-2618	73.7x10 ³	0.33	0.00276	51.9	165.4	62.12	0.04020
2	Cast iron	110 x10 ³	0.22	0.00680	127.5	169.3	61.78	0.02715

Table: 9 FEA result comparison between two materials

CONCLUSION

Compared results based on AL Alloy and cast iron (Al-2618, Cast Iron). AL alloy gives best results. . Because the stresses induced in the piston reduced. Hence the prepared piston with AL-Alloy specimen show better results than the normal Cast Iron.

REFERENCE

- P.carvalheira,p.Goncalves FEA of two engine pistons made of al cast alloy A390 and Ductile iron 65-45-12 under service conditions.
- V. Ucar, A. Ozel, “Use of the finite element technique to analyze the influence of coating materials, material phase state and the purity on the level of the developed thermal stresses in plasma coating systems under thermal loading conditions”, Surface and Coatings Technology, Vol. 142 144, 2001, pp. 950-953
- R. Ravi Raja Malarvannan¹, P. Vignesh² “Experimental Investigation And Analysis Of Piston By Using Composite Materials,” Department Of Manufacturing Engineering, College Of Engineering Guindy. Vol 04, Article K100; November 2013.
- Hongyuan Zhang, Xingguo Ma, Shuangliang Ge “Modeling And Transient Dynamics Analysis To a new Type Of Engine Piston” 2013.
- Ghodake A. P. Patil K.N. “Piston Design And Analysis By Cae Tools” Department Of Mechanical Engineering, Snd Coe & Rc Yeola, Nashik, India Iosr Journal Of Engineering (Iosrjen) Issn: 2250-3021 Isbn: 2878-8719 Pp 33-36 National Symposium On Engineering And Research
- Ch.Venkata Rajam, P.V.K.Murthy, M.V.S.Murali Krishna, G.M.Prasada Rao “Design Analysis And Optimization Of Piston Using Catia And Ansys” International Journal Of Innovative Research In Engineering & Science Issn 2319-5665(January2013, Issue 2 Volume1)