# STATIC AND THERMAL ANALYSIS OF I.C ENGINE PISTON FOR VARIOUS PISTON CROWNS

*A Project report submitted*

*in partial fulfilment of the requirements for the award of the degree of*

**BACHELOR OF TECHNOLOGY**

***In***

**MECHANICAL ENGINEERING**

***by***


| | |
|---|---|
| ***D. DEVI*** | ***(315126520044)*** |
| ***P. BHANU VENKATESH PRASAD*** | ***(315126520020)*** |
| ***B. RAJEEV VARMA*** | ***(315126520025)*** |
| ***G. PRUDHVEESH*** | ***(315126520065)*** |

***Under the esteemed guidance of***

**Dr. B. NAGARAJU,**

B.E., M.Tech., M.E., Ph.D., MISTE, MIE, MSAE.

HOD & Professor

**DEPARTMENT OF MECHANICAL ENGINEERING**

PRAGNANAM BRAHMA

**ANITS**

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES**

Autonomous status accorded by UGC and Andhra University

(Approved by AICTE, Permanently Affiliated to Andhra University, Accredited and reaccredited by NBA, accredited by NAAC- 'A' Grade)

**Sangivalasa, Bheemunipatnam Mandal**

**Visakhapatnam (District) – 531162**


**(2015-2019)**

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES**

Autonomous status accorded by UGC and Andhra University
(Approved by AICTE, Permanently Affiliated to Andhra University, Accredited and reaccredited by NBA, accredited by NAAC- 'A' Grade)
**Sangivalasa, Bheemunipatnam Mandal**
**Visakhapatnam (District) – 531162**

PRAGNANAM BRAHMA
**ANITS**

# CERTIFICATE

This is to certify that the Project Report entitled "**STATIC AND THERMAL ANALYSIS OF I.C ENGINE PISTON FOR VARIOUS PISTON CROWNS** " being submitted by **D.Devi (315126520044) P.BHANU VENKATESH PRASAD (315126520020), B.RAJEEV VARMA (315126520025), G.PRUDHVEESH (315126520065)**, in partial fulfilments for the award of Degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING**. It is the work of bonafide, carried out under the guidance and supervision of **Dr. B. NAGARAJU**, HOD & Professor of Department of Mechanical Engineering, during the academic year of 2015-2019.

| PROJECT GUIDE | HEAD OF THE DEPARTMENT |
|---|---|
| 11.4.19 | 11.4.19 |
| **(Dr. B.NagaRaju)** | **(Dr. B.NagaRaju)** |
| Head of the Department | Head of the Department |
| Mechanical Department | Mechanical Department |
| ANITS, Visakhapatnam. | ANITS, Visakhapatnam. |
| PROFESSOR & HEAD<br>Department of Mechanical Engineering<br>ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE<br>Sangivalasa-531 162 VISAKHAPATNAM Dist. A.P. | PROFESSOR & HEAD<br>Department of Mechanical Engineering<br>ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE<br>Sangivalasa-531 162 VISAKHAPATNAM Dist. A.P. |

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER: 13-4-19

PROFESSOR & HEAD
Department of Mechanical Engineering
ANK NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa 531 162 VISAKHAPATNAM Dist A P

EXTERNAL EXAMINER: 13/4/19

# ACKNOWLEDGEMENTS

The satisfaction that accompanies the successful completion of any task would be incomplete without mentioning of the people where ceaseless cooperation made it possible, whose constant guidance and encouragement crown all the efforts with success.

We sincerely thank to our project guide **Dr. B. Naga Raju**, HOD & Professor, Anil Neerukonda Institute of technology and Sciences for his wholehearted co-operation, unfailing inspiration and valuable guidance. Throughout the project work, his useful suggestions, constant encouragement has gone a long way in his valuable time for his patience and understanding he showed.

We are very thankful to **Prof. T. Subramanyam,** Principal, and **Prof. T.V. Hanumantha Rao,** Dean-Academics, Anil Neerukonda Institute of Technology & Sciences for their valuable suggestions.

We also sincerely thankful to **Dr. M. Raja Roy**, Associate Professor, and **Sri S. Phani Kumar**, Asst. Professor, **Sri J. V. Bhanu Tej,** Asst. Professor. Anil Neerukonda Institute of technology and Sciences for providing us their valuable guidance, suggestions, encouragement & ideas during whole project work externally to complete our project.

We express our sincere thanks to the staff members of Mechanical Engineering for their kind co-operation and support to carry on work.

We also thank all those who contributed directly or indirectly in successfully carrying out this work.

**D. DEVI**

**P. BHANU VENKATESH PRASAD**

**B. RAJEEV VARMA**

**G. PRUDHVEESH**

# ABSTRACT

In I.C. Engine, piston is the most complex and important part. Therefore, for smooth running of vehicle, piston should be in proper working condition. Pistons fail mainly due to mechanical stresses and thermal stresses.

Combustion in an I.C. Engine is generally considered the space between the crown of the piston and cylinder head. In the present work, three pistons which are similar in size and shape except for the shape of the crown have been selected for structural analysis.

In this study work, the analysis of the piston consists of mainly design procedure for piston of 4 stroke petrol engine for KTM Duke 200 and its analysis. The model of the piston is designed for given design specifications and the modelling was carried out using solid works for various piston crowns. Consequently, the analysis was carried out using ANSYS software for the given constraints based on the working conditions of the piston. The structural and thermal analysis was carried out on two different materials of piston i.e. Al Alloy 6061 and Al Alloy 2618, for three different piston crown shapes such as Flat, Dished and Hemi-spherical. This analysis was completed for different known input parameters to obtain out parameters such as stress, deformation, temperature and total heat flux.

Based on the analysis, the piston crown shape for one of the materials which have high factor of safety proves to be the best, as Factor of Safety plays the major role which gives more life time for all conditions and best suited for present generation for 100% complete combustion of fuel in combustion to increase the efficiency. it is analysed that piston made of Al 2618 having hemi spherical crown have less stress induced and reduced heat flux and is best suggested

# Table of contents

# List of Figures

# List of tables

# CHAPTER 1

# 1. INTRODUCTION

Engine pistons are one of the most complex components among all automotive and other industry field components. The engine can be called the heart of a vehicle and the piston may be considered the most important part of an engine. There are lots of research works proposing, for engine pistons, new geometries, materials and manufacturing techniques, and this evolution has undergone with a continuous improvement over the last decades and required thorough examination of the smallest details. Notwithstanding all these studies, there are a huge number of damaged pistons. Damage mechanisms have different origins and are mainly wear, temperature, and fatigue related. The fatigue related piston damages play a dominant role mainly due to thermal and mechanical fatigue, either at room or at high temperature.

The main requirement of piston design is to measure the prediction of temperature distribution on the surface of piston which enables us to optimize the thermal aspects for design of piston at lower cost. Most of the pistons are made of an aluminium alloy which has thermal expansion coefficient, 80% higher than the cylinder bore material made of cast iron. This leads to some differences between running and the design clearances. Therefore, analysis of the piston thermal behaviour is extremely crucial in designing more efficient compressor. Good sealing of the piston with the cylinder is the basic criteria to design of the piston. Also, to improve the mechanical efficiency and reduce the inertia force in high speed machines the weight of the piston also plays major role. To allow for thermal expansion, the diameter of the piston must be smaller than that of the cylinder. The necessary clearance is calculated by estimating the temperature difference between piston and cylinder and considering the coefficient of thermal expansion of piston.

## 1.1 Internal Combustion Engine

An internal combustion engine is an engine that operates by burning its fuel inside the engine. In contrast a steam engine burns its fuel outside the engine. The most common internal combustion engine type is gasoline powered. Others include those fueled by diesel, hydrogen, methane, propane, etc. Engines typically can only run on one type of fuel and require adaptations to adjust the air/fuel ratio or mix to use other fuels. In a gasoline engine, a mixture of gasoline and air is sprayed into a cylinder. This is compressed by a piston and at optimal point in the compression stroke, a spark plug creates an electrical spark that ignites the fuel. The combustion of the fuel results in the generation of heat, and the hot gases that are in the cylinder are then at a higher pressure than the fuel-air mixture and so drive the piston back down. These combustion gases are vented and the fuel-air mixture reintroduced to run a second stroke. The outward linear motion of the piston is ordinarily harnessed by a crankshaft to produce circular motion. Valves control the intake of air-fuel mixture and allow exhaust gasses to exit at the appropriate times.

### 1.1.1 Two-stroke

The two-stroke type of internal combustion engine is typically used in utility or recreational applications which require relatively small, inexpensive, and mechanically simple motors (chainsaws, jet skis, small motorcycles, etc.). The two-stroke engine is simple in construction, but complex dynamics are employed in its operation. There are several features unique to a two-stroke engine. First, there is a reed valve between the air-fuel intake and the crankcase.

Air-fuel mixture enters the crankcase and is trapped there by the one-way reed valve. Next, the cylinder has no valves as in a conventional four stroke engine. Intake and exhaust are accomplished by means of ports - special holes cut into the cylinder wall which allows fuel-air mixture to enter from the crankcase, and exhaust to exit the engine. These ports are uncovered when the piston is in the down position.

Air-fuel mixture is drawn into the crankcase from the carburettor or fuel injection system through the reed valve. When the piston is forced down, the exhaust port is uncovered first, and hot exhaust gases begin to leave the cylinder. As the piston is now in the down position, the crankcase becomes pressurized, and when the intake port into the cylinder is uncovered, pressurized air-fuel mixture enters the chamber. Both the intake and exhaust ports are open at the same time, which means the timing and air flow dynamics are critical to proper operation. As the piston begins to move up, the ports are closed off, and the air-fuel mixture compresses and is ignited; the hot gases increase in pressure, pushing the piston down with great force and creating work for the engine.

The major components of two-stroke engines are tuned so that optimum airflow results. Intake and exhaust tubes are tuned so that resonances in airflow give better flow than a straight tube. The cylinder ports and piston top are shaped so that the intake and exhaust flows do not mix.

### 1.1.2 Four-stroke

The four-stroke internal combustion engine is the type most commonly used for automotive and industrial purposes today (cars and trucks, generators, etc.). On the first (downward) stroke of the piston, fuel/air is drawn into the cylinder. The following (upward) stroke compresses the fuel-air mixture, which is then ignited - expanding exhaust gases then force the piston downward for the third stroke, and the fourth and final (upward) stroke evacuates the spent exhaust gasses from the cylinder.

The four-stroke cycle is more efficient than the two-stroke cycle, but requires considerably more moving parts and manufacturing expertise. In CI engines fuel is injected into the combustion chamber at about 15° before T.D.C. during the compression stroke. For the best efficiency the combustion must complete within 15° to 20° of crank rotation after T.D.C. in the working stroke. Thus, it is clear that injection and combustion both must complete in the short time.

In S.I engine mixing takes place in carburettor however in C.I engines this has to be done in the combustion chamber. To achieve this requirement in a short period is an extremely difficult job particularly in high speed C.I engines. In order to achieve this, an organized air movement called swirl is provided to produce high relative velocity between the fuel droplets and the air.

## 1.2 Piston:

A piston is a component of reciprocating engines, reciprocating pumps, gas compressors and pneumatic cylinders, among other similar mechanisms. It is the moving component that is contained by a cylinder and is made gas-tight by piston rings. In an engine, its purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod and/or connecting rod. In a pump, the function is reversed and force is transferred from the crankshaft to the piston for the purpose of compressing or ejecting the fluid in the cylinder. In some engines, the piston also acts as a valve by covering and uncovering ports in the cylinder wall.

The main functions of the piston are as follows:

(i) It transmits the force due to gas pressure inside the cylinder to the crankshaft through the connecting rod.

(ii) It compresses the gas during the compression stroke.

(iii) It seals the inside portion of the cylinder from the crankcase by means of piston rings.

(iv) It takes the side thrust resulting from obliquity of the connecting rod.

(v) It dissipates large amount of heat from the combustion chamber to the cylinder wall.

## 1.3 Major Parts of The Piston

### *(i)* Piston Head or Crown

It is the top portion of the piston which withstands the gas pressure inside the cylinder. It has flat, concave or convex shape depending upon the construction of combustion chamber. Shown in Fig 1.1

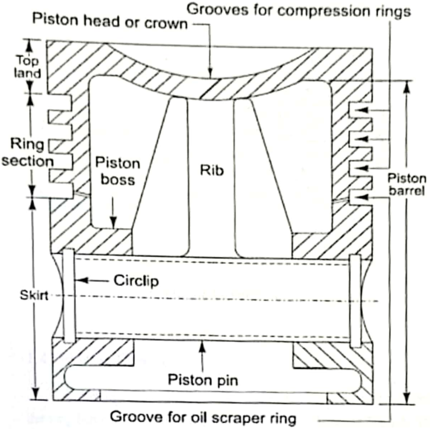


**Fig.1.1 Major parts of the piston**

### *(ii)* Piston Rings

They act as seal and prevent the leakage of gas past the piston. Piston rings are also called '*compression*' rings.

### *(iii)* Oil Scraper Ring

It prevents the leakage of lubricating oil past the piston into the combustion chamber.

### *(iv)* Piston Skirt

It is the lower part of the piston below the piston rings which acts as bearing surface for the side thrust exerted by the connecting rod.

### *(v)* Piston Pin

It connects the piston to the connecting rod. It is also called '*gudgeon*' pin or '*wrist*' pin.

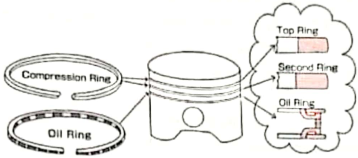


**Fig. 1.2 Piston Rings**

## 1.4 Oil Ring Groove

A groove cut into the piston around its circumference, at the bottom of the ring belt or at the lower end of piston skirt as shown in Fig 1.2. Oil ring grooves are usually wider than compression ring grooves and generally have holes or slots through the bottom of the groove for oil drainage to the interior of the piston as

## 1.5 Compression Ring Groove

A groove cut into the piston around its circumference, in the upper part of the ring belt as shown in Fig 1.2. The depth of groove varies depending on piston size and types of rings used.

## 1.6 The Design Requirements for the Piston:

(i) It should have sufficient strength to withstand the force due to combustion of fuel and also the inertia forces due to reciprocating parts.

(ii) It should have sufficient rigidity to withstand thermal and mechanical distortions.

(iii) It should have adequate capacity to dissipate the heat from the crown to the cylinder wall through the piston rings and the skirt.

(iv) It should have minimum weight to reduce the inertia force due to reciprocating motion.

(v) It should form an efficient seal to prevent leakage of flue gases from combustion chamber to the crankcase past the piston. It should also prevent leakage of lubricating oil into the combustion chamber past the piston.

(vi) It should have sufficient bearing area to take the side thrust and prevent undue wear.

(vii) It should result in noiseless operation.

(viii) It should provide adequate support for the piston pin, which connects the small end of the connecting rod.

Piston crowns have multiple functions

1. Convert the pressure developed in ignition stroke to a downward force-carry that force to the piston rod.
2. Sustain pressure waves generated by occasional knocking.
3. Act as a thermal barrier between cylinder and crank area.
4. Introduce swirl in the fuel-controls knocking and helps more uniform combustion.
5. Proper design would reduce the piston weight and cost.
6. The piston crown is machine to provide channels and cavities for the favourable reception and redirection of jets of flame issuing from the pre combustion chamber.

## 1.7 Materials Used for Study

### 1.7.1 Al 6061-T6 alloy

6061 is a Aluminium alloy containing Magnesium and Silicon as its major alloying elements. Originally called as "Alloy 615". It has good mechanical properties.

Features of Al 6061 alloy:

- Relatively high Strength.
- Good Workability.

- High Resistance to corrosion.
- Widely available.

**Table 1.1 Composition of Al 6061 Alloy**

| Element | Weight Percentage |
|---|---|
| Aluminium (Al) | 95.8-98.6 |
| Manganese (Mn) | Max 0.15 |
| Silicon (Si) | 0.4-0.8 |
| Magnesium (Mg) | 0.8-1.2 |
| Copper (Cu) | 0.15-0.4 |
| Zinc (Zn) | Max 0.25 |
| Others | Max 0.15 |

**Table 1.2 Material properties of Al 6061 Alloy**

| Properties | Metric |
|---|---|
| Density | 2.7g/cc |
| Thermal conductivity | 167 W/mK |
| Ultimate Tensile strength | 310MPa |
| Modulus of elasticity | 68.9 GPa |
| Poisson's ratio | 0.33 |

### 1.7.2 Al 2618 alloy

Aluminium alloys have high ductility and corrosion resistance. At sub-zero temperatures, their strength can be increased. However, their strength can be reduced at high temperatures of about 200-250°C. Aluminium 2618 alloy is an age hardenable alloy containing magnesium and copper. Its major alloying elements are Copper and Magnesium.

Features of Al 2618 alloy

- It has good machinability, formability
- Aluminium 2618 alloy can be heat treated at 530°C (985°F) for a sufficient period of time followed by quenching in water.
- Has high strength.
- Can be welded using resistance welding techniques.

**Table 1.3 Composition of Al 2618 Alloy.**

| Element | Content (%) |
|---|---|
| Aluminium, 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 1.4 Material properties of Al 2618 Alloy**

| Properties | Metric |
|---|---|
| Density | 2.767 g/cm³ |
| Melting point | 510°C |
| Tensile strength | 480 MPa |
| Yield strength | 370 MPa |
| Shear strength | 260 MPa |
| Fatigue strength | 125 MPa |
| Elastic modulus | 73.7 GPa |
| Poisson's ratio | 0.33 |
| Elongation | 10% |
| Thermal conductivity with T61 treatment | 146 W/mK |

# CHAPTER 2

# 2. INTRODUCTION TO MODELLING & FEM

## 2.1 The SolidWorks Software

The SolidWorks CAD software is a mechanical design automation application that lets designers quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

## 2.2 SolidWorks Fundamentals

### 2.2.1 Concepts

Parts are the basic building blocks in the SolidWorks software. Assemblies contain parts or other assemblies, called sub-assemblies. A SolidWorks model consists of 3D geometry that defines its edges, faces, and surfaces. The SolidWorks software lets you design models quickly and precisely. SolidWorks models are

- Defined by 3D design
- Based on components

#### ➢ 3D Design

SolidWorks uses a 3D design approach. As you design a part, from the initial sketch to the final result, you create a 3D model. From this model, you can create 2D drawings or mate components consisting of parts or sub-assemblies to create 3D assemblies. You can also create 2D drawings of 3D assemblies. When designing a model using SolidWorks, you can visualize it in three dimensions, the way the model exists once it is manufactured.

#### ➢ Component Based

One of the most powerful features in the SolidWorks application is that any change you make to a part is reflected in all associated drawings or assemblies.

### 2.2.2 Terminology

- **Origin**

Appears as two blue arrows and represents the (0, 0, 0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0, 0, 0) coordinate of the sketch.

- **Plane**

You can use planes for adding a 2D sketch, section view of a model, or a neutral plane in a draft feature.

- **Axis**

Straight line used to create model geometry, features, or patterns. You can create an axis in different ways, including intersecting two planes. The Solid Works application creates temporary axes implicitly for every conical or cylindrical face in a model.

- **Face**

Boundaries that help define the shape of a model or a surface. A face is a Selectable area (planar or non-planar) of a model or surface.

- **Edge**

Location where two or more faces intersect and are joined together.

- **Vertex**

Point at which two or more lines or edges intersect. You can select vertices for sketching and dimensioning.

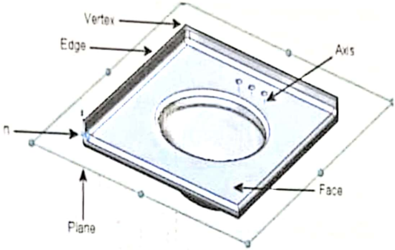


**Fig.2.1 Terminology in Solid Work**

### 2.2.3 User Interface

The SolidWorks application includes user interface tools and capabilities to help you create and edit models efficiently, including

- **Windows Functions**

The SolidWorks application includes familiar Windows functions, such as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste are also part of the SolidWorks application.

- **SolidWorks Document Windows**

SolidWorks document windows have two panels. The left panel, or Manager Panel, contains

- **Feature Manager Design tree**

Displays the structure of the part, assembly, or drawing.

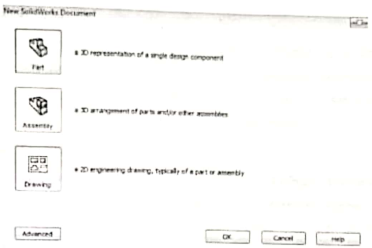


**Fig.2.2 Feature Manager Design Tree**

- **Property Manager**

Provides settings for many functions such as sketches, fillet features, and assembly mates.

- **Configuration Manager**

Let 's you create, select, and view multiple configurations of parts and assemblies in a document

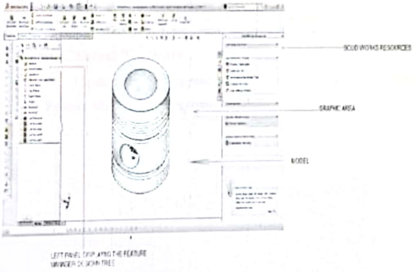


**Fig.2.3 Configuration Manager**

## ➢ Menus

You can access all SolidWorks commands using menus. SolidWorks menus use Windows conventions, including submenus and checkmarks to indicate that an item is active. You can also use context-sensitive shortcut menus by clicking the right mouse button.

## ➢ Toolbars

You can access SolidWorks functions using toolbars. Toolbars are organized by function, for example, the Sketch or Assembly toolbar. Each toolbar comprises individual icons for specific tools, such as Rotate View, Circular Pattern, and Circle.

## ➢ Command Manager

The Command Manager is a context-sensitive toolbar that dynamically updates based on the active document type.

## ➢ Shortcut Bars

Customizable shortcut bars let you create your own sets of commands for part, assembly, drawing, and sketch mode.

## ➢ Context Toolbars

Context toolbars appear when you select items in the graphics area or Feature Manager Design tree.

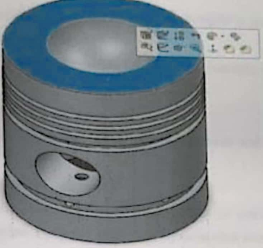

**Fig.2.4 Context Toolbars**

➢ **Previews**

With most features, the graphics area displays a preview of the feature you want to create.

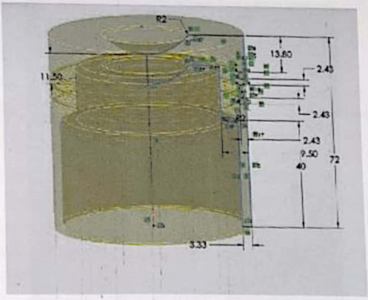


**Fig.2.5 Preview of Piston**

Before you actually design the model, it is helpful to plan out a method of how to create the model.

After you identify needs and isolate the appropriate concepts, you can develop the model:

- **Sketches**

Create the sketches and decide how to dimension and where to apply relations.

- **Features**

Select the appropriate features, such as extrudes and fillets, determine the best features to apply, and decide in what order to apply those features.

- **Assemblies**

Select the components to mate and the types of mates to apply.

- **Dimensions**

You can specify dimensions between entities such as lengths and radii. When you change dimensions, the size and shape of the part changes.

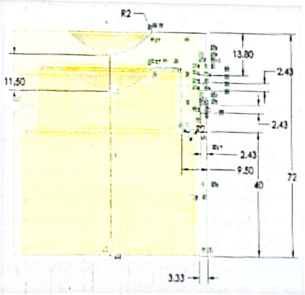


**Fig.2.6 Sketch Dimensions**

### 2.2.4 Assemblies

An assembly design consists of two or more components assembled together at their respective work positions using parametric relations. In Solid Works, these relations are called mates.

- **Assembling Components Using the Mate Property Manager**

In Solid Works, mates can be applied using the Mate Property Manager. Choose the Mate button in the Assemble Command Manager or choose Insert > Mate from the Menu Bar menus; the Mate Property Manager will be invoked, as shown.

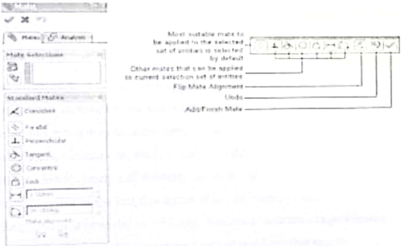


**Fig.2.7 Assembling Components Using the Mate Property Manager**

## 1.3. Finite Element Analysis

### 2.3.1 Introduction to finite element method.

Finite element method is a numerical analysis technique for obtaining the approximate solutions to a wide variety of engineering problems. Although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industry. The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Advances in computer technology and CAD systems, has led to increased use of FEM in research as well as industry as complex problems can be modelled and released with relative ease.

In more and more engineering situations today, we find that it is necessary to obtain approximate numerical solutions closed-form solutions. For example, we may want to find the load capacity of a plate that has several stiffeners and odd-shaped holes, the concentration of pollutants during non-uniform atmospheric conditions, or the rate of fluid flow through a passage of arbitrary shape. Without too much effort, we can write down the governing equations and boundary conditions for these problems, but we see immediately that no several approximate numerical analysis methods have evolved over the years; a commonly used method is the finite difference scheme. The familiar finite difference model of a problem gives a point wise approximation to the governing equations. This model (formed by writing difference equations for an array of grid points) is improved as more points are used. With finite difference techniques we can treat some fairly difficult problems but, for example, when we encounter irregular geometries or an unusual specification of boundary conditions, we find that finite difference techniques become hard to use. Simple analytical solution can be found. The difficulty in these three examples lies in the fact that either of the geometry or some other feature of the problem is irregular or arbitrary. Analytical solutions to problems of this type seldom exist; yet these are the kinds of problems that engineers are called upon to solve.

Unlike the finite difference method, which envisions the solution region as an array of grid points, the finite element method envisions the solution region as built up of many small, interconnected sub regions or elements. A finite element model of a problem gives a piecewise approximation to the governing equations. The basic premise of the finite element method is that a solution region can be analytically modelled or approximated by replacing it with an assemblage of discrete elements. Since these elements can be put together in a variety of ways, they can be used to represent exceedingly complex shapes. As an example of how a finite difference model and a finite element model might be used to represent a complex geometrical shape, consider the hemispherical piston cross section in Figure 4.1. For this device we may want to find the distribution of displacements and stresses for a given force loading or the distribution of temperature for a given thermal loading. The interior coolant passage of the piston, along with its exterior shape, gives it a no simple geometry. A uniform finite difference mesh would reasonably cover the piston. It is 10 nodded tetrahedral elements and the number of countable nodes is 67121 and the number of elements is 53806.

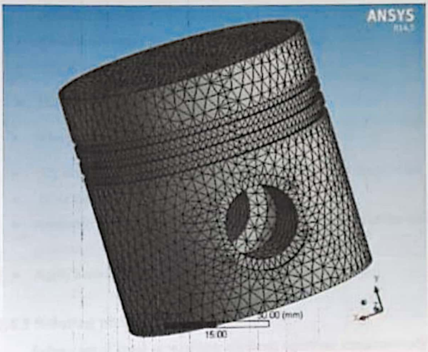


**Fig.2.8 Finite Mesh of Piston**

On the other hand, the finite element model (using the simplest two-dimensional element—the triangle) gives a better approximation to the region. Also, a better approximation to the boundary shape results because the curved boundary is represented by straight lines of any inclination

## 2.4 Basic Steps in The Finite Element Analysis

The basic steps involved in finite element analysis consist of the following

### 2.4.1 Pre-processing phase

Create and discretize the solution domain into finite elements i.e. subdivide the problem into nodes and elements.

- Assume a shape function to represent the physical behaviour of an element; that is an approximate continuous function is assumed to represent the solution of an element.
- Develop equations for all the elements in the mesh.
- These generally take form
- [K]{U} = {F}
- Where [K] is a square matrix, known as stiffness matrix
- {U} is the vector of (unknown) nodal displacements or temperature
- {F} is the vector of applied nodal forces
- Assemble the elemental equations to obtain the equations of the whole problem. Construct the global stiffness matrix.
- Apply boundary conditions, initial conditions, and loading.

### 2.4.2 Solution phase

Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results of primary degrees of freedom or unknowns, such as displacement values at different nodes in structural problem or temperature values at different nodes in heat transfer problem.

### 2.4.3 Post processing phase

- Computation of any secondary unknowns or variables e.g. the gradient of the solution.
- Interpretation of the results to check whether the solution makes sense.
- Tabular and/or graphical presentation of the results.

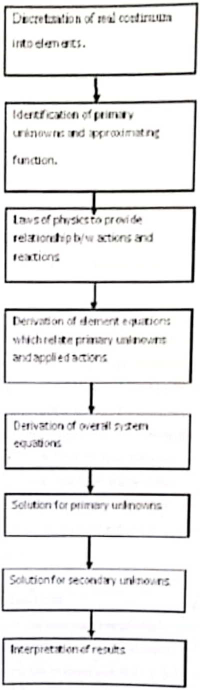


**Fig.2.9 Tabular presentation of results.**

## 2.5 PROCEDURE OF FINITE ELEMENT ANALYSIS

The approximating functions (sometimes called interpolation functions) are defined in terms of the values of the field variables at specified points called nodes or nodal points. Nodes usually lay on the element boundaries where adjacent elements are connected. In addition to boundary nodes, an element may also have a few interior nodes. The nodal values of the field variable and the interpolation functions for the elements completely define the behaviour of the field variable within the elements.

Another advantage of the finite element method is the variety of ways in which one can formulate the properties of individual elements. There are basically three different approaches. The first approach to obtaining element properties is called the direct approach because its origin is traceable to the direct stiffness method of structural analysis. The direct approach suggests the need for matrix algebra in dealing with the finite element equations. Element properties obtained by the direct approach can also be determined by the variation approach. The variation approach relies on the calculus of variations and involves extremizing a functional. For problems in solid mechanics the functional turns out to be the potential energy, the complementary energy, or some variant of these, such as the Reissner variation principle.

Regardless of the approach used to find the element properties, the solution of a continuum problem by the finite element method always follows an orderly step-by-step process. To summarize in general terms how the finite element method works we will succinctly list these steps now; they will be developed in detail later.

### 2.5.1 Discretize the continuum

The first step is to divide the continuum or solution region into elements. The turbine blade has been divided into triangular elements that might be used to find the temperature distribution or stress distribution in the blade. A variety of element shapes may be used, and different element shapes may be employed in the same solution region. Indeed, when analysing an elastic structure that has different types of components such as plates and beams, it is not only desirable but also necessary to use different elements in the same solution.

Although the number and the type of elements in a given problem are matters of engineering judgment, the analyst can rely on the experience of others for guidelines.

### 2.5.2 Select interpolation functions

The next step is to assign nodes to each element and then choose the interpolation function to represent the variation of the field variable over the element. The field variable may be a scalar, a vector, or a higher-order tensor. Often, polynomials are selected as interpolation functions for the field variable because they are easy to integrate and differentiate. The degree of the polynomial chosen depends on the number of nodes assigned to the element, the nature and number of unknowns at each node, and certain continuity requirements imposed at the nodes and along the element boundaries. The magnitude of the field variable as well as the magnitude of its derivatives may be the unknowns at the nodes.

### 2.5.3 Find the element properties

Once the finite element model has been established, the element properties are found out.

### 2.5.4 Assemble the element properties to obtain the system equations

To find the properties of the overall system modelled by the network of elements we must assemble all the element properties. In other words, we combine the matrix equations expressing the behaviour of the elements and form the matrix equations expressing the behaviour of the entire system. The matrix equations for the system have the same form as the equations for an individual element except that they contain many more terms because they include all nodes. The basis for the assembly procedure stems from the fact that at a node, where elements are interconnected, and the value of the field variable is the same for each element sharing that node. A unique feature of the finite element method is that the system equations are generated by assembly of the individual element equations. In contrast, in the finite difference method the system equations are generated by writing nodal equations.

### 2.5.5 Impose the boundary conditions

Before the system equations are ready for solution, they must be modified to account for the boundary conditions of the problem. At this stage we impose known nodal values of the dependent variables or nodal loads.

### 2.5.6 Solve the system equations

The assembly process gives a set of simultaneous equations that we solve to obtain the unknown nodal values of the problem. If the problem describes steady or equilibrium behaviour, then we must solve a set of linear or nonlinear algebraic equations. If the problem is unsteady, the nodal unknowns are a function of time, and we must solve a set of linear or nonlinear ordinary differential equations.

### 2.5.7 Make additional computations

Many times we use the solution of the system equations to calculate other important parameters. For example, in a structural problem the nodal unknowns are displacement components. From these displacements we calculate element strains and stresses. Similarly, in a heat-conduction problem the nodal unknowns are temperatures, and from these we calculate element heat fluxes.

## 2.6 Range of Applications

Applications of the finite element method divide into three categories, depending on the nature of the problem to be solved. In the first category are the problems known as equilibrium problems or time-independent problems. The majority of applications of the finite element method fall into this category. For the solution of equilibrium problems in the solid mechanics area, we need to find the displacement distribution and the stress distribution for a given mechanical or thermal loading. Similarly, for the solution of equilibrium problems in fluid mechanics, we need to find pressure, velocity, temperature, and density distributions under steady-state conditions. In the second category are the so-called Eigen value problems of solid and fluid mechanics. These are steady-state problems whose solution often requires the determination of natural frequencies and modes of vibration of solids and fluids. Examples of Eigen value problems involving problems involving both solid and fluid mechanics appear in civil engineering when the interaction of lakes and dams is considered and in the aerospace engineering when the sloshing of liquid fuels in the flexible tanks is involved.

## 2.7 Commercial Finite Element Software

The first commercial finite element software made its appearance in 1964. The Control Data Corporation sold it in a time-sharing environment. No pre-processors (mesh generators) were available, so engineers had to prepare data element by element and node by node. The introduction of personal computers (PCs) powerful enough to run finite element software provides extremely cost-effective problem solving.

Today we have hundreds of commercial software packages to choose from. A small number of these dominate the market. It is difficult to make compared on finite element basis

**Table 2.1 Leading Commercial Finite Element Software Companies**

| Company Name | Product Name | Web Site |
|---|---|---|
| Hibbitt, Karlsson & Sorensen | ABAQUS | www.hks.com |
| Ansys, Incorporated | ANSYS | www.ansys.com |
| Structural Data Research Corp. | SDRC-Ideas | www.sdrc.com |
| Parametric Technology, Inc. | RASNA | www.ptc.com |
| MSC Software Corp. | MSC/NASTRAN | www.mscsoftware.com |

In contrast to the early days, we can now use computer-aided design (CAD) software or solid modellers to generate complex geometries, at either the component or assembly level. We can (with some restrictions) automatically generate elements and nodes, by merely indicating the desired nodal density. Software is available that works in conjunction with finite elements to generate structures of optimum topology, shape, or size. Nonlinear analyses including contact, large deflection, and nonlinear material behaviour are routinely addressed.

## 2.8 Introduction to Ansys

ANSYS is a general-purpose finite element modelling package for numerically solving a wide variety of mechanical problems. ANSYS simulation software enables organisations to confidently predict how their products will operate in the real world. It expands the use of physics. It gains access to any form of engineering field someone may account in. The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis.

A typical ANSYS analysis has three distinct steps

- Build the model
- Apply loads and obtain the solution
- Review the results

- **Building a Model**

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a job name and analysis title. Then, you use the PREP7 pre-processor to define the element types, element real constants, material properties, and the model geometry.

- **Apply loads and obtain the solution**

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 pre-processor.

- **Review the results**

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results. Two postprocessors are available-POST1 and POST26.

### 2.8.1 Loading overview

The main goal of a finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. The loads can be applied on the model in a variety of ways in the ANSYS program. Also, with the help of load step options, one can control how the loads are actually used during solution.

### 2.8.2 Solution

In the solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The results of the solution are

- nodal degree-of-freedom values, which form the primary solution.

- derived values, which form the element solution.

The element solution is usually calculated at the elements' integration points. The ANSYS program writes the results to the database as well as to the results file (*Jobname*.RST, RTH, RMG, or. RFL).

Several methods of solving the simultaneous equations are available in the ANSYS program: frontal solution, sparse direct solution, Jacobi Conjugate Gradient (JCG) solution, Incomplete Cholesky Conjugate Gradient (ICCG) solution, Preconditioned Conjugate Gradient (PCG) solution, and an automatic iterative solver option (ITER).

### 2.8.3 Model generation

The ultimate purpose of a finite element analysis is to re-create mathematically the behaviour of an actual engineering system. In other words, the analysis must be an accurate mathematical *model* of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system.

In ANSYS terminology, the term *model generation* usually takes on the narrower meaning of generating the nodes and elements that represent the spatial volume and connectivity of the actual system. Thus, *model generation* in this discussion will mean the process of *defining the geometric configuration of the model's nodes and elements*. The ANSYS program offers the following approaches to model generation:

- Creating a solid model within ANSYS.
- Using direct generation.
- Importing a model created in a computer-aided design (CAD) system.

### 2.8.4 Meshing

The procedure for generating a mesh of nodes and elements consists of three main steps

- Set the element attributes.
- Set mesh controls (optional). ANSYS offers a large number of mesh controls, which one can choose from to suit their needs.
- Generate the mesh.

The second step, setting mesh controls, is not always necessary because the default mesh controls are appropriate for many models. If no controls are specified, the program will use the default settings on the DESIZE command to produce a free mesh. As an alternative, one can use the Smart Size feature to produce a better-quality free mesh.

Different types of mesh elements are

a) Solid-70

b) Solid-90

c) Solid-87

## 2.9 Introduction to Ansys Workbench

ANSYS Workbench is the framework upon which the industry's broadest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user every step of the way. Even complex multi physics analyses can be performed with drag-and-drop simplicity.

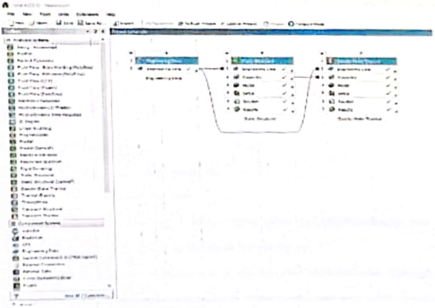

**Fig.2.10 Workbench 14.5 Interface**

The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analyses, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis.

The ANSYS Workbench interface is arranged into two primary areas: The toolbox and the project Schematic. The toolbox contains the system templates that you can use to build a project. The project Schematic is the area if the interface where you will manage your project. The new project schematic view shows an overall view of the entire simulation project. Engineering intent, data relationships and the state of the entire project are visible at a glance, even for complex analyses involving multiple physics. In addition to this, you will see a menu bar and a toolbar with frequently used functions. You can also use context menus, accessible via a right- mouse click, on schematic items, and cells. Context menus provide capabilities to add to and modify projects. The entire process is persistent. Changes can be made to any portion of the analysis and the ANSYS Workbench platform will manage the execution of the required applications to update the project automatically, dramatically reducing the cost of performing design iterations.

### 2.9.1 Ansys workbench features

- Bidirectional, parametric links with all major CAD systems.
- Integrated, analysis-focused geometry modelling, repair, and simplification via ANSYS Design Modeller.
- Highly-automated, physics-aware meshing.
- Automatic contact detection.
- Unequalled depth of capabilities within individual physics disciplines.
- Unparalleled breadth of simulation technologies.
- Complete analysis systems that guide the user start-to-finish through an analysis.
- Comprehensive multi physics simulation with drag-and-drop ease of use.

- Flexible components enable tools to be deployed to best suit engineering intent.
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance.
- Complex project schematics can be saved for re-use.
- Pervasive, project-level parameter management across all physics.
- Automated what-if analyses with integrated design point capability.
- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions.

## 2.10 Structural Analysis

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

### Types

The seven types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements. Structural analyses are available in the ANSYS/Multi physics, ANSYS/Mechanical, ANSYS/Structural, and ANSYS/Linear Plus programs only.

One can perform the following types of structural analyses

- **Static Analysis**

Used to determine displacements, stresses, etc. under static loading conditions. It comprises of both linear and non-linear static analysis.

Nonlinearity can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

- **Modal Analysis**

Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

- **Harmonic Analysis**

Used to determine the response of a structure to harmonically time varying loads.

- **Transient Dynamic Analysis**

Used to determine the response of a structure to arbitrarily time varying loads. All non-linearity mentioned under Static Analysis above are allowed.

- **Spectrum Analysis**

An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

- **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.

- **Explicit Dynamics Analysis**

ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

- In addition to the above analysis types, several special-purpose features are available
- Fracture mechanics
- Composites
- Fatigue



Scanned with CamScanner

- p-Method

## 2.11 Thermal Analysis

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution one performs via ANSYS calculates nodal temperatures and then uses the nodal temperatures to obtain other thermal quantities. The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

**Types**

ANSYS supports two types of thermal analysis

- A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
- A transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

# CHAPTER 3

# 3. LITERATURE REVIEW

The finite element analysis is performed using software to investigate and analyze static structural & steady state thermal stress distribution at the real engine condition during combustion process. Piston skirt may appear deformation usually causes crack on the upper end of the piston head. Due to deformation, stress concentration is caused on the upper end of the piston and the stress distribution on the piston mainly depends on the deformation of piston. Therefore, piston crown should have enough stiffness to reduce the deformation. The preliminary analysis presented in the paper was to compare the behavior of the combustion engine piston made of different type of materials under static load. Finite element analysis is used to analyze stresses in a piston of an internal combustion engine. The stresses due to combustion gas load only are considered so as to reduce the weight and hence to increase the power output of engine.

The distribution of the temperature on the top surface of the piston which predicts the top surface of the piston may be going to damaged or broken during the operating conditions. The materials with high thermal conductivity are considered better than the material type of low thermal conductivity.

### 3.1 Design and Analysis of IC Engine piston using CATIA and ANSYS Software.

**Satish kumar [1]** designed a piston for 4 stroke petrol engine for hero splendor pro bike and its analysis by its comparison with original piston dimensions used in bike. The design procedure we involve determination of various piston dimensions using analytical method under maximum power condition. The following results were drawn by him:

The use of analytical method is nearly equal to the actual dimensions used. Hence it provides a fast procedure to design a piston and takes very less time and from the obtained results of von mosses stress analysis the design is considered to be safe.

**3.2 Design and Analysis of piston of Internal Combustion Engine ON Different Materials using CAE Tool Analysis.**

**G. Siva Prasad [2]** done design and structural analysis of the piston with different materials like Al alloy 4032, Al SI 4340 Alloy steel and Titanium Ti-6Al-4v. The following results were drawn by him:

The maximum stress intensity is on the piston crown in all the materials, but stress intensity is close to the yield strength of the different materials piston it analyzed that the Al alloy 4032 is suitable for I.C Engine piston.

**3.3 Theoretical Analysis of Stress and Design of Piston Head using CATIA & ANSYS.**

**Dilip Kumar sonar [3]** has done project on stress distribution on piston of IC Engine by using FEA. The paper describes the FEA technique to predict the higher stress and critical region on the component. The design specifications provided by KIRLOSKAR ENGINE is used. The following results were drawn by him.

A though thermal stress is not the responsible for different slice of damaged piston, it remains a problem on engine piston and its solution remain a goal for piston manufactures. From the analysis, it is evident that thermal stress was higher than mechanically induced stress. Hence it could be concluded that the piston was failed due to the thermal load rather than the mechanical load and during optimization design there could be put into consideration to ensure that thermal load is reduced

**3.4 Design Analysis and optimization of piston using CAE Tools.**

**Vaibhar[4]** has done projected on stress distribution of two different aluminium alloys piston by using CAE tools. The piston is of 4-stroke single cylinder of Bajaj Pulsar 220cc motorcycle. This paper illustrates the produce for analytical design of two different aluminium alloy pistons. The following results were drawn by him.

Deformation is low in AL-CH4 1250 alloy piston as compared to conventional piston. About 30% mass reduction is possible with ALGH4 1250 alloy. Safety factor increased by 27% at same working condition. Thus AL-GH4 1250 alloy piston is better than convectional alloy piston.

### 3.5 Thermal Analysis of Aluminium Alloy Piston.

**B.A.Devan[5]** done project on thermal distribution on different piston materials. Piston mainly fails due to thermal conditions. So as to search out proper thermal distribution different piston materials are considered. The following results were drawn by him.

For different material on piston is observed that total heat flux reduces in Al SiC composite compared to Al-Si, Al-Mg-Si. Alloy the maximum heat flux reduced by increases comparison of carbides in Al SiC alloy. Results comparison between theoretical and analysis simulated done and found approximately same.

### 3.6 Thermal analysis and optimization of IC engine part using finite element method.

**A.R Bhagat [6]** has described stress distribution of 4 stroke piston engine by using FEA. The paper describes mesh optimization with using FEA technique to predict the higher stress and critical region on the component. The optimization is carried out to reduce the stress concentration on the upper end of the piston i.e. piston crown and piston skirt and sleeve. The following results were drawn by him:

In order to reduce the stress concentration, the piston crown should have enough stiffness to reduce the deformation.

### 3.7 Design, Thermal analysis and Optimization of a piston using ANSYS.

**Shahanwaz Adam Havale [7]** presents a numerical method using thermos-mechanical decoupled FEM to calculate the thermal stress only caused by the uneven temperature distribution. In this work, the main example is placed on the study of thermal behavior of functionally graded materials obtained by means of using a commercial code ANSYS on aluminum alloy piston surfaces. The following conclusions are drawn by him:

It can be seen that the distribution of thermal mechanical coupling is minimum in the type of eutectic alloy. But for spherical eutectic alloy (AlSi 18 Cu Mg Ni) the maximum stress is medium and the distribution of heat flow rate intensity is maximum in eutectic alloy.

**3.8 Thermal analysis of piston for the influence of secondary motion.**

**Vinay V.Kuppast [8]** done FEA on the piston deformation due to thermal load.

The simulation results are used to predict the effect of temperature on piston deformation and its secondary motion which are the principal source of engine vibration and noise. The conclusions are drawn by him:

The FEA analysis of piston reveals that the thermal stresses induced in the piston are proportional to the directional deformation. Hence the FEA results are concluded conveniently for the prediction of engine vibration and noise.

## 3.9 The Scope of Work from Literature Survey.

A comprehensive examination of existing literature revealed some of the gaps in works reported on Structural and Thermal analysis of piston with different crowns. The work done particularly in the area of static and thermal analysis of piston with different shapes of crowns had been published only in a limited scope.

From the above literature, it is understood that various researchers considered various parameters to find the effect on structural and thermal stresses in static with change in materials, various crown shapes, and thermal conditions. The challenging task is to get a better factor of safety for best crown shape for an optimal material.

Hence in this work the authors have chosen two different materials with three different crown shapes for a piston. For each material static and thermal analysis was carried for different load conditions to know the effect of stress induced, total heat flux and corresponding factor of safety. This project helps us to understand the importance of crown shapes on stresses induced and heat bearing capacity of the piston.

# CHAPTER 4

# 4. CALCULATIONS AND ANALYSIS

## 4.1 Definition of Problem

The present work deals with determining the stresses induced due to Pressure loads on a piston and heat flux on piston used in I.C. Engines.

Generally, the pistons are in cylindrical in shape & reciprocate within the cylinder due to gas forces which vary continuously throughout the cycle. Generally, the pistons are subjected to high temperature on the top surfaces during combustion of fuel and this high temperature prevails for a short very short duration in the cycle.

### 4.1.1 Engine specifications:

The engine used for this work is a single cylinder four stroke liquid cooled type KTM Duke

CC petrol engine. The engine specifications are given in Table 4.1

**Table 4.1 Engine Specifications**

| Parameters | Values |
|---|---|
| Engine Type | Four stroke, petrol engine |
| Number of cylinders Single cylinder | Single cylinder |
| Bore | 72mm |
| Stroke | 49mm |
| Displacement volume | 200cm$^3$ |
| Compression ratio | 11.5:1 |
| Maximum power | 18.44KW at 10000rpm |
| Maximum Torque | 19.2Nm at 8000rpm |
| Number of revolutions/cycles. | 2 |

## 4.1.2 Analytical design

Nomenclature,

IP = indicated power produced inside the cylinder (W)

$\eta$ = mechanical efficiency = 0.8 (Assumption)

n = number of working stroke per minute = N/2 (for four stroke engine)

N = engine speed (rpm)

L = length of stroke (mm)

A = cross-section area of cylinder (mm2)

V = volume of the piston (mm3)

$t_h$ = thickness of piston head (mm)

D = cylinder bore (mm)

Pmax = maximum gas pressure or explosion pressure (MPa) = 15.03 MPa

$\sigma_t$ = allowable tensile strength (MPa)

$\sigma_{ut}$ = ultimate tensile strength (MPa)

F.O.S = Factor of Safety = 2.25

K = thermal conductivity (W/m K)

## 4.2 Calculation of Dimensions of Piston:

### 4.2.1 Thickness of piston head ($t_h$):

The piston thickness of piston head calculated using the following Grashoff's formula,

$t_h = D \sqrt{((3P)/(16\sigma_t))}$ in mm

P= maximum pressure in N/mm²=15.03 N/mm².

D= cylinder bore/outside diameter of the piston in mm= 72mm.

$\sigma_t$=permissible tensile stress for the material of the piston.

= $\sigma_t$ =276/2.25=110 MPa.

$t_h$ = 11.5mm.

### 4.2.2 Radial thickness of ring ($t_1$):

$t_1 = D\sqrt{3Pw/\sigma_t}$

Where,

D = cylinder bore in mm=72mm.

Pw= pressure of fuel on cylinder wall in N/mm². Its value is limited from 0.025N/mm² to 0.042N/mm². Here Pw value is taken as 0.042N/mm² while $\sigma_t$= 110Mpa for aluminium alloy.

(t1): 2.436mm.

### 4.2.3 Axial thickness of ring ($t_2$)

The thickness of the rings may be taken as

$t_2 = 0.7t_1$ to $t_1$

$= t_1$=2.436mm.

### 4.2.4 Width of the top land ($b_1$)

The width of the top land varies from

$b_1 = t_h$ to $1.2\ t_h$

$=1.2\ T_H = 1.2 \times 11.5 = 13.8$mm.

### 4.2.5 Width of other lands ($b_2$):

Width of other ring lands varies from

$b_2 = 0.75t_2$ to $t_2$

$=t_2= 2.436$mm.

### 4.2.6 Maximum Thickness of Barrel at the top end ($t_3$):

$t_3 = 0.03*D + b + 4.5$ mm

$t_3$=0.03*72+2.436+4.9=9.5mm.

### 4.2.7 Thickness of piston barrel at the open end ($t_4$):

$t_4$= 0.25 $t_1$ to 0.35 $t_1$)

$t_4$=0.35*9.5=3.325mm.

### 4.2.8 Piston pin diameter ($d_o$):

$d_o$=0.03D=21.66mm.

### 4.2.9 length of piston (l)

l= D to 1.5D, l= D= 72mm.

## 4.3 Analysis of the Model:

Here Stress analysis of the piston model has been performed to obtain the value and parameters at which the piston would be damaged. Damages may have different origins: mechanical stresses; thermal stresses; wear mechanisms; temperature degradation, oxidation mechanisms; etc. For this analysis parameters like Pressure, Temperature, Thermal Stress. have been used and to discuss the effects of these parameters on the model.

### 4.3.1 Structural analysis:

When air-fuel mixture is ignited, pressure from the combustion gases is applied to the piston head, forcing the piston towards the crankshaft. Due to the pressure at the piston head, there are mainly two critical areas: piston pin holes and localized areas at the piston head. Subsequently will be presented different engine pistons where the cracks initiated on those areas. The pressurized gases travel through the gap between the cylinder wall and the piston. The upward motion of the piston is against the pressure of the gases. The causes a tremendous effect on the piston head leading to its damage and deformation of the piston head.

In this project, KTM Duke 200 engine is considered and design is made according to the specification of its commercial piston. While analysing, using STATIC STRUCTURAL ANALYSIS, first analysis is performed considering pressure only. Here the pressure under consideration is 15.03MPa. Pressure is applied on the piston head and the pin hole is considered as the fixed support. This process is done for piston of two alloys and for 3 different piston crowns i.e. Flat, Dished, Hemi-Spherical by varying pressure acting i.e. 15MPa, 10MPa, 5MPa. From analysis total deformation, Equivalent strains and stress are obtained.

### 4.3.2 Calculation of maximum gas pressure on piston head

Mechanical efficiency of the engine (η) = 80 %.

$$\eta = \frac{Brake\ power\ (B.P)}{Indicated\ Power\ (I.P)}$$

$$B.P. = \frac{2\pi NT}{60} = \frac{2\times\pi\times19.2\times8000}{60} = 16KW$$

$$I.P = \frac{BP}{\eta} = \frac{16}{0.8} = 20KW$$

$$I.P = P\times A\times L\times\frac{N}{2}$$

$$20 \times 1000 = P_{MEAN} \times \frac{\pi}{4} \times 0.072^2 \times 0.049 \times \frac{8000}{2\times60}$$

So, Indicated mean effective pressure = 1.503 MPa

Maximum Gas Pressure $P_{max} = 10\times P_{mean} = 10 \times 1.503 = 15.03$ MPa or $N/mm^2$.

### 4.3.3 Boundary conditions

The following boundary conditions are applied to model for further analysis of the piston

1. Fixed support is provided at the piston pin hole as shown in Fig 4.1, the force acting on the piston is transferred to connecting rod with the help of piston pin.
2. Frictionless support is provided along the walls of the piston as shown in Fig 4.2. As the piston reciprocates inside the cylinder at very high speed, to reduce heat due to friction between the cylinder inner walls and piston walls smooth finish is provided.
3. Pressure is applied on the piston crown uniformly as shown in Fig 4.3.


**FIG 4.1 Fixed Support at Piston Pin**


**Fig 4.2 Frictionless Support to Sidewalls**

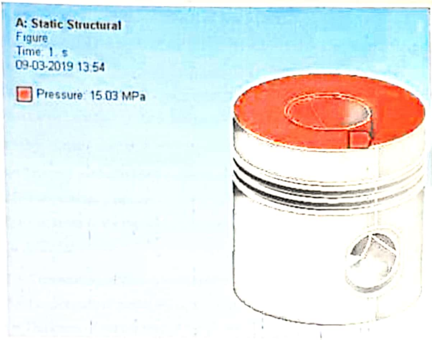


**Fig 4.3 Pressure Applied on Piston Crown.**

### 4.3.4 Thermal analysis:

Thermal stresses are difficult to simulate because there are, in a piston, two kinds of thermal stresses. Thermal stresses due to the vertical distribution of the temperature along the piston high temperatures at the top and lower temperatures at the bottom. There is a homogeneous and regular gradient of temperature on the radial direction along the head of the component. It is observed that the bowl rim area is the area where temperatures are higher. Thermal deformations under the operating bowl rim temperature are constrained by the surrounding material

It involves determination of temperature distribution throughout the piston body. It also involves the determination of heat fluxes to be assigned on the lateral surfaces of the piston. These are the essentially the boundary conditions to be assigned.

### 4.3.5 Theoretical calculation of heat flux:

Piston head and the walls of the piston are subjected to high temperatures at expansion stroke.

Let us consider 400°C On the piston head and 150°C on the piston walls. Maximum Heat flux of the piston crown is calculated as

$q = -k\frac{dT}{dx}$ From Fourier's law

K= Thermal conductivity of piston material

dT=Temperature gradient ($T_2$-$T_1$)

dx=Thickness of the top land of the piston(t)

$q = -k\frac{(T2-T1)}{t}$

$T_2$= Temperature of the top surface of the piston crown. 400°C

$T_1$=Temperature of piston walls. 150°C

t = Thickness of the top land of the piston

**Case-I** Maximum Heat flux in Al 2618-T61 Alloy Piston

$q = -k\frac{(T2-T1)}{t}$

$q = -146\times\frac{(400-150)}{11.5\times10-3}$

$=3.173 \times 10^6$ W/m2

**Case-II** Maximum Heat flux in Al 6061-T6 Alloy Piston

$q = -k\frac{(T2-T1)}{t}$

$q = -167\times\frac{(400-150)}{11.5\times10-3}$

$=3.630 \times 10^6$ W/m2

# CHAPTER 5

# 5. RESULTS AND COMPARISON

The results shown below are three types of pistons made of Al 6061 Alloy and Al 2618 alloy. The structural & thermal analysis is carried out for three different types of piston having the same basic dimensions with exception being the shape and size of the bowl on the piston crown.

These three types are as given below

a) Flat

b) Dished

c) Hemispherical

The piston with material Al 6061-T6,

The fig 5.1 shows the Total deformation of the piston made up of Al 6061-T6 with flat crown.

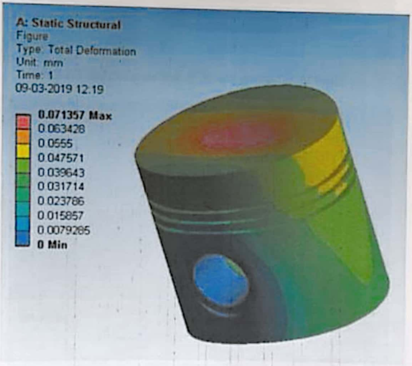


**FIG 5.1 -Total Deformation Of Flat Crown For Al 6061.**

Similarly, the fig 5.2 & 5.3 shows the Total Deformation of pistons with dished crown and hemispherical crown for the piston material Al 6061-T6.


**FIG 5.2-Total deformation of dished crown for Al 6061.**

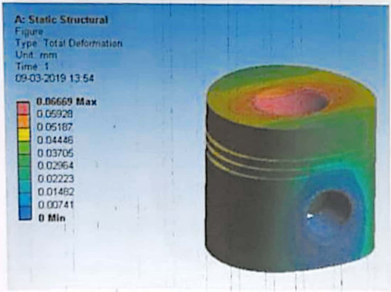


**FIG 5.3-Total deformation of hemi-spherical crown for Al 6061**

The static analysis is carried out on piston with different crowns using Ansys. The Fig 5.4, 5.5 & 5.6 shows the variation of equivalent stress induced in a piston with different piston crowns such as flat, dished & hemispherical for piston material Al 6061-T6

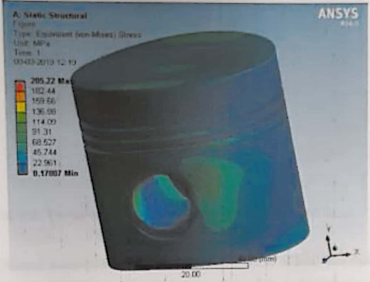


**FIG 5.4- Variation of Equivalent stress for a piston with flat crown for Al 6061.**

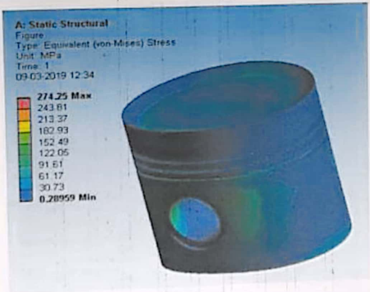


**FIG 5.5- Variation of Equivalent stress for a piston with dished crown for Al 6061.**

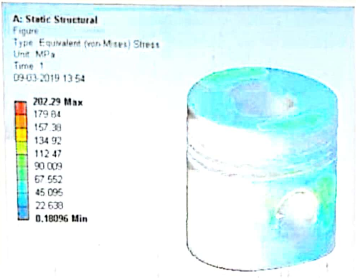


**FIG 5.6- Variation of Equivalent stress for a piston with hemi spherical crown for Al 6061.**

The thermal analysis is carried to know the behaviour of total heat flux for a piston material with three different crowns. The fig 5.7, fig 5.8 & fig 5.9 shows the variation of the heat flux for a piston material Al 6061-T6. With three different piston crowns such as flat, dished & hemispherical.

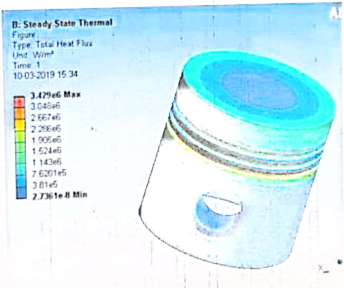


**FIG 5.7- Variation of total heat flux for piston with flat crown for Al 6061.**



Scanned with CamScanner


**FIG 5.8- Variation of total heat flux for piston with dished crown for Al 6061.**

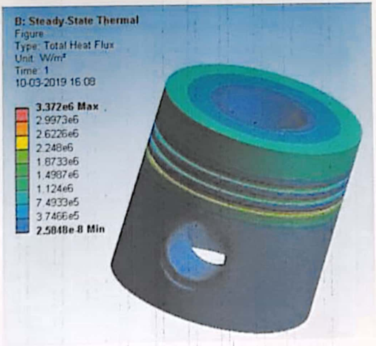


**FIG 5.9- Variation of total heat flux for piston with hemispherical crown for Al 6061.**

The same structural & thermal analysis is carried for another piston material Al 2618 with three different crowns, to know the behaviour of total deformation, Equivalent stress & Total heat flux, which are reported in the fig 5.10 to fig 5.17.

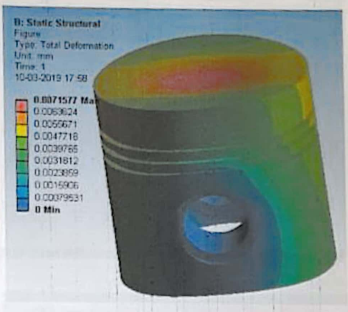


**FIG 5.10 -Total deformation of flat crown for Al 2618**

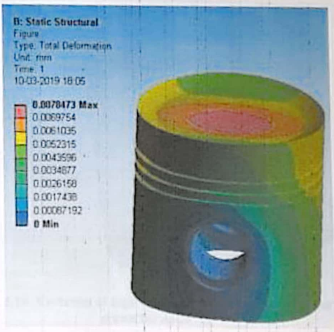


**FIG 5.11-Total deformation of dished crown for Al 2618**



Scanned with CamScanner

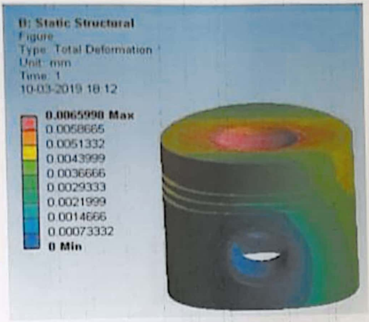


**FIG 5.12-Total deformation of hemi-spherical crown for Al 2618.**

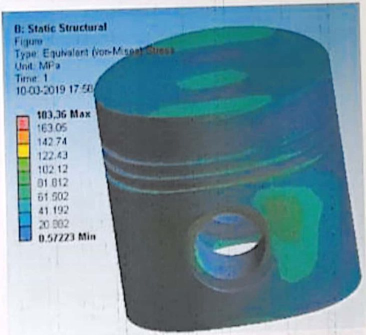


**FIG 5.13- Variation of Equivalent stress for a piston with flat crown for Al 2618.**


**FIG 5.14- Variation of Equivalent stress for a piston with Dished crown for Al 2618.**

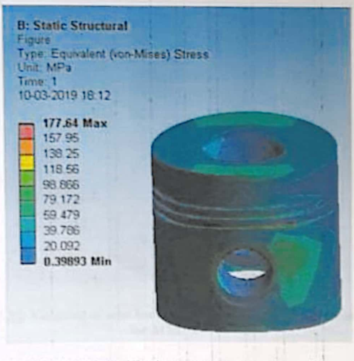


**FIG 5.15- Variation of Equivalent stress for a piston with Hemispherical crown for Al 2618.**

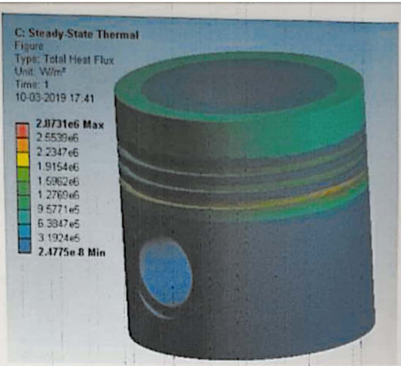


**FIG 5.16- Variation of total heat flux for piston with flat crown for Al 2618.**

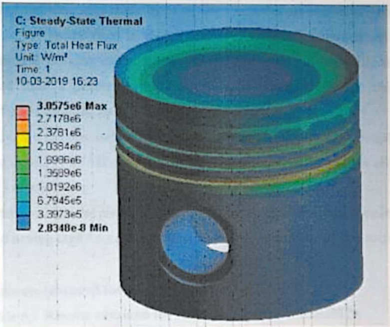


**FIG 5.17- Variation of total heat flux for piston with dished crown for Al 2618.**

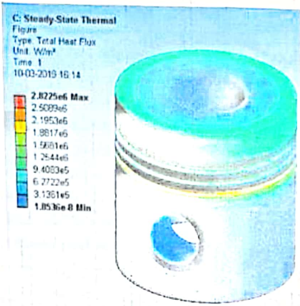


**FIG 5.18- Variation of total heat flux for piston with hemispherical crown for Al 2618.**

The results obtained for Total deformation, Equivalent stress for a piston having three different crowns which are made up of Al 6061 material are shown in Table 5.1. Further the factor of safety (fos) is also determined from the obtained values of equivalent stress and working stress. It may be inferred from the table 5.1, that the stresses induced from a piston having flat crown is noted as 205.22 Mpa and corresponding f.o.s is found to be as 1.51.

Similarly, the stresses induced for a piston having dished and hemi-spherical shape are found to be 274.25 MPa and 202.29Mpa, further the values of f.o.s are noted as 1.13 and 1.53 respectively.

It is understood that the piston with hemi-spherical crown shape may be induced less stresses and having high f.o.s on comparison with the piston having flat and dished crown shapes.

The above results are tabulated for a piston of material Al6061-T6

**Table 5.1 Results obtained for Al 6061-T6 at load 15.03MPa**

| Crown Shape | Total deformation (mm) | Strain | Stress (MPa) | Fos |
|---|---|---|---|---|
| **Flat** | 7.1357e-002 | 3.338e-003 | 205.22 | 1.51 |
| **Dished** | 7.8939e-002 | 4.2311e-003 | 274.25 | 1.13 |
| **Hemi-Spherical** | 6.6690e-002 | 3.5303e-003 | 202.29 | 1.53 |

The results obtained for Total deformation, Equivalent stress for a piston having three different crowns which are made up of Al 2618 material are shown in Table 5.2. Similarly, the same results of equivalent stresses and f.o.s are noted for a piston with a material Al2618-T61 having three different crown shapes. It was inferred that the piston with hemi-spherical shaped may be induced has stress with high f.o.s.

From the results further inferred that the piston made up of the Al2618-T61 with hemi-spherical shape can be withstand for high load bearing capacity with high F.o.s. The above results are tabulated for a piston of material Al 2618-T61

**Table 5.2 Results obtained for Al 2618-T61 at load 15.03MPa**

| Crown Shape | Total deformation(mm) | Strain | Stress (MPa) | Fos |
|---|---|---|---|---|
| Flat | 7.1577e-003 | 2.6495e-004 | 183.36 | 2.39 |
| Dished | 7.8473e-003 | 3.7639e-004 | 260.6 | 1.68 |
| Hemi-Spherical | 6.5998e-003 | 2.5676e-004 | 177.64 | 2.47 |

The same analysis is done for different load conditions i.e. for 5 MPa, 10Mpa, from the obtained results, A graph is plotted for the stresses produced in two material at different loads for three different crown shapes i.e. Flat, dished & Hemispherical.

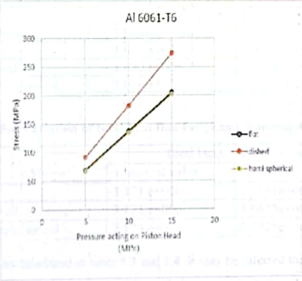


**Fig 5.19- Stress Produced for Variation of Pressure for Al 6061-T6.**

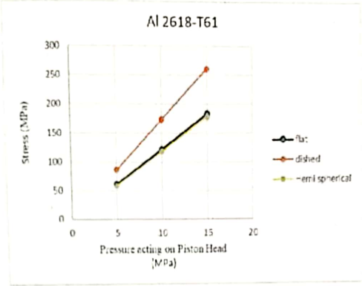


**Fig 5.20 - Stress Produced for Variation of Pressure for Al 2618-T61.**

The thermal analysis is carried out to the known the behaviour of total heat flux as the piston is subjected to high temperatures in operation. Hence thermal analysis is carried out for a piston with two different materials with three different crown shapes having boundary conditions as T2= Temperature of the top surface of the piston crown. i.e. 400°C. T1=Temperature of the piston walls. i.e.150°C

**Table 5.3 Variation of total heat flux for piston material Al 6061**

| Shape | Total Heat Flux (W/m²) | |
|---|---|---|
| | Theorical value | Analytical value |
| Flat | 3.630 e+006 | 3.429e+006 |
| Dished | 3.630 e+006 | 3.660e+006 |
| Hemispherical | 3.630 e+006 | 3.372e+006 |

**Table 5.4 Variation of total heat flux for piston material Al 2618**

| Shape | Total Heat Flux (W/m²) | |
|---|---|---|
| | Theorical value | Analytical value |
| Flat | 3.173 e+006 | 2.8731e+006 |
| Dished | 3.173 e+006 | 3.0575e+006 |
| Hemispherical | 3.173 e+006 | 2.8225e+006 |

The results are tabulated in table 5.3 and 5.4. It may be inferred that the piston with material Al6061 with hemi-spherical shape may be reduced heat flux in comparison with other two crown shapes.

# CONCLUSIONS

# 6. CONCLUSIONS

The present work deals with static and thermal analysis of piston made of two different materials such as Al6061 & Al 2618 alloys for three different crown shapes such as flat, dished & hemi-spherical for different loads. The following conclusions were drawn from this work:

1. It is observed from static analysis that Aluminium alloy Al 2618 has almost 40% more factor of safety in comparison with Al 6061 for the same design models.
2. A Piston with Hemi spherical crown exhibits lower induced stresses and minimum deformation in comparison with other piston crowns.
3. Dished crown has the least factor of safety, in which high stresses are induced and is least suggestable.
4. It is observed from thermal analysis that heat flux reduces in alloy Al 2618 compared to Al 6061. Out of which by considering Hemispherical crown, has the least heat flux value for both materials.
5. The heat flux values obtained analytically are good in agreement with theoretical results.
6. Both flat and hemispherical crown pistons have nearby values structurally and thermally, but by using hemispherical crown better results are obtained.
7. It is observed that out of two materials used for work, Al 2618 alloy proves to be best and also Hemi-spherical crown shape is best of the three shapes considered in terms of structural and thermal.

# 7.REFERENCES

[1] Satish kumar of SKP college Thiruvanamalai. " Design and Analysis of IC Engine piston using CATIA and ANSYS Software" Research Gate, issue May-2015.

[2] G.Siva Prasad, K.Dinesh Achari, E.Dileep Kumar Goud, M.Nagaraju, K.Srikanth "Design and Analysis of piston of Internal Combustion Engine ON Different Materials using CAE Tool Analysis" International Journal of Engineering and Techniques - Volume 2 Issue 3, May - June 2016.

[3] Dilip Kumar Sonar,Madhura Chattopadhyay "Theoretical Analysis of Stress and Design of Piston Head using CATIA & ANSYS" International Journal of Engineering Science Invention Volume 4 Issue 6|| June 2015

[4] Vaibhav V. Mukka war, Abhishek D. Bangale, Nithin D. Bhushale, Ganesh M. Surve" Design Analysis and optimization of piston using CAE Tools" International Journal of Advances in Mechanical and Civil Engineering, ISSN: 2394-2827 Volume-2, Issue-2, April-2015.

[5] B.A.Devan, G. Ravindra Reddy "Thermal Analysis of Aluminium Alloy Piston" International Journal of Emerging Trends in Engineering Research (IJETER), Vol. 3 No.6, 2015.

[6] A. R. Bhagat, Y. M. Jibhakate "Thermal analysis and optimization of IC engine part using finite element method" International Journal of Modern Engineering Research (IJMER) www.ijmer.com Vol.2, Issue.4, July-Aug 2012.

[7] Shahanwaz Adam Havale1, Prof. Santosh Wankhade "Design, Thermal analysis and Optimization of a piston using ANSYS" International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 04 Issue: 12 | Dec-2017

[8] Vinay V. Kuppast, Dr.S.N. Kurbet\, H.D. Umeshkumar, Adarsh B.C, Karnataka "Thermal analysis of piston for the influence of secondary motion" International Journal of Engineering Research and Applications (IJERA) Vol. 3, Issue 3, May-Jun 2013 .

[9] V. B. Bhandari, "Design of Machine Elements", 3rd Edition, McGraw Hill.

[10] I. C. Engines by Ganeshan

## 8.WEB REFERENCES

[1]http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=ma6061t6

[2] https://www.azom.com/amp/article.aspx?ArticleID=6617

[3] www.rajaroy.co.in