FINITE ELEMENT ANALYSIS OF PRIME COMPONENTS OF AN INTERNAL COMBUSTION ENGINE

(Structural, Thermal and Fluid flow Aspects)

A Dissertation submitted to the

Faculty of Technology
of

University of Delhi

TOWARDS THE PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE AWARD OF DEGREE OF MASTER OF ENGINEERING
IN
MECHANICAL ENGINEERING
(THERMAL ENGINEERING BRANCH)

SUBMITTED

BY

MURUGA CHANDRA KUMAR.P
(ROLL No.:3204)

UNDER THE VALUABLE GUIDANCE
OF
PROF. S. MAJI
HOD MECHANICAL

DEPARTMENT OF MECHANICAL ENGINEERING
DELHI COLLEGE OF ENGINEERING
UNIVERSITY OF DELHI

SESSION 2003-2005

DELHI COLLEGE OF ENGINEERING
UNIVERSITY OF DELHI
DELHI-110 042

CERTIFICATE

This is to certify that the dissertation entitled **“THE FINITE ELEMENT ANALYSIS OF PRIME COMPONENTS OF AN INTERNAL COMBUSTION ENGINES”** being submitted by **MURUGA CHANDRA KUMAR.P,** in partial fulfillment for the award of the degree of **Master of Engineering** in **Mechanical Engineering** (**Thermal Engineering Branch**) of **DELHI UNIVERSITY**, is a record of bonafide research work carried out by him at Delhi College of Engineering, Delhi- 110 042 during the year 2004-05.

He has worked under my guidance and supervision and has fulfilled the requirements for the submission of this dissertation, which to my knowledge has reached the required standard. Further, it is also certified that this work has not been submitted for any other degree or diploma in any college to the best of my knowledge and belief.

PROF.S. MAJI

Guide and Head of the Department
Department of Mechanical Engineering
Delhi College of Engineering
Delhi-110 042

CONTENTS

CHAPTER NO. Page Number

LIST OF FIGURES

Figure No. **Name of the figure** **Page Number**

LIST OF TABLES

LIST OF SYMBOLS

v	volume
p	pressure
T	Temperature
V	Velocity
E	Modulus of Elasticity
K	Kelvin
µ	Poisson’s ratio
σ	Stress
KJ	Kilo joules
W	Watt
J	joules
MPa	Mega Pascal
F	Force
N	Revolutions per Minute (RPM)
m	Meter
min	Minute
max	Maximum
mm	Millimeter
DMX	Deflection Maximum
DMn	Deflection Minimum
°C	Degree centigrade
Ø	Diameter
%	Percent

ACKNOWLEDGEMENT

I express my sincere thanks to **Professor S.Maji**, Head of the Department of Mechanical Engineering, Delhi College of Engineering and **Professor C. K. DUTTA**, former Head, Department of Mechanical Engineering, who provided me a great opportunity to do my project in the field of Finite Element Analysis (FEA).

I convey my deepest gratitude to my revered guide **Professor S.Maji**, Head of the Department of Mechanical Engineering, Delhi College of Engineering for his expert guidance, inspiration, suggestion and constant encouragement during entire course of this project work, which enabled me to bring out this report in an eloquent manner. Without his guidance, directions and constructive criticisms, this Finite Element Study would have been impossible. Therefore I deeply thank my guide from my inner heart.

I am deeply indebted to Prof Ashok Dev, Shri Sethi, and Mrs.Meenakshi, Controllers, Computer centre, Delhi College of Engineering, who allowed me to carry out my research in computer center. Without their support and co operations, completion of this work would have been in doubt.

I convey my gratitude to Professor R.K Shukla, Head Librarian, Shri Bharadwaj, Assistant Librarian, and Mr.Pushpander, library Trainee, Central library, Delhi College of Engineering, who permitted me to work in library computers. I would like to thank Mr.Vivek Babu, I year M.E (P), who provided me a computer facility to compile my thesis.

P.Muruga Chandra Kumar

ABSTRACT

Internal combustion engines have been, and will remain for the foreseeable future, a vital and active area of engineering education and research. It has been one of the active areas of research for many academic researches since the inception of internal combustion engine. Most of the researches in internal combustion engines are of operating performance and fuel performance oriented. Almost all of the components in an internal combustion engine are subjected to heat loads. Every mechanical component has been designed for a particular structural and heat strength. Piston seizure and cylinder block melting are typical problems when heat and structural loads on the components exceeds the design strengths. In the current study attempt has been made to simulate the physical working conditions of components of an internal combustion engine. The analysis is virtual simulation (because it was carried out with the help of a digital computer and a Software tool-ANSYS). The currant study emphasis on stress, strain, temperature, heat flux, thermal gradient distributions in the component materials. The study was carried out using the Finite Element Methods Approach. CATIA and ANSYS soft wares have been used to carry out this finite element study. CATIA was used for the solid modeling of engine components and ANSYS was used for the analysis. The study was carried out on **prime** components of an internal combustion engine such as Piston, Engine block, Connecting rod, Exhaust manifold and Exhaust Muffler. Piston and Engine clock are subjected to both structural and thermal loads and connecting rod is subjected to compressive gas loads, Exhaust manifold and exhaust muffler are subjected heat and fluid flow loads. The type of study is **peak moment simulation** which means that only the conditions prevailing at the point of combustion are simulated. Peak stress, deformation and temperature profiles are matter of concern when analysis was carried out in piston, engine block, and connecting rod. Velocity, temperature, and pressure distributions are matter of concern when analysis was carried out on exhaust manifold and exhaust muffler. Study was carried out for different loading conditions, different components materials properties. Peak stresses were compared with the strength of the corresponding materials and finally attempts have been made to optimize the component geometry.

CHAPTER-1

INTRODUCTION

The internal combustion engine was conceived and developed in the late 1800s.It has had significant impact on society, and is considered one of the most significant inventions of the last century. The internal combustion engine has been the foundation for the successful development of many commercial technologies. For example, considering the fact that how internal combustion engine transformed the transportation industry, allowing the invention and improvement of automobiles, trucks, airplanes, and trains. Internal combustion engines can deliver power in the range from 0.01 kW to 20×10^3kW depending on their displacement [4]. They compete in the market place with electric motors, gas turbines, and steam turbines. The major applications are in the vehicular (automobiles and truck), railroad, marine, aircraft, home use, and stationary areas. The vast majority of internal combustion engines are produced for vehicular applications, requiring a power output in the order of 10^2 kW [4]. Internal combustion engines have become the dominant prime power technology in several areas.

The first internal combustion engines used the reciprocating piston-cylinder principle in which a piston oscillates back and forth in a cylinder and transmits power to a drive shaft through a connecting rod and crankshaft mechanism. Valves are used to control the flow of gas into and out of the engine. The components of a reciprocating internal combustion engine, block, piston, valves, crankshafts, and connecting rod, have remained unchanged since the late 1800s.The main differences between a modern day engine and one-built 100 years ago are the thermal efficiency and the emissions level. For many years, internal combustion engine research was aimed at improving thermal efficiency, reducing noise and vibration. As a consequence, the thermal efficiency has increased from about 10% to values as high as 50%.Since 1970,with the recognition of the importance of air quality, there has also been a great deal of work devoted to reducing emissions from engines[4]. Currently, emissions control requirement are one of the major factors in the design and operation of internal combustion engines.

Diesel engine is a high mass burning engine. Diesel engine has been the prime mover for all commercial purposes for so long. The compression ratio of the petrol engine is between 6 and 10 while for a diesel engine it is from 16 and 20 [1].The diesel engine is also called as explosion engine as nature of combustion is explosion of diesel fuel. Diesel engine is a dirty and smoky engine, it is very rigid in construction as the burning pressures, and temperatures are typically high. The inlet air in the diesel engine is unthrottled. The power is controlled by the amount of fuel injected into the cylinder. In order to ignite the fuel-air mixture, diesel engines are required to operate at a higher compression ratio, resulting in a higher theoretical thermal efficiency as compared to the spark ignition engine. Diesel engine performance is limited by the formation smoke, which forms if there is inadequate mixing of fuel and air [1]. The power output of the diesel engine depends upon displacement volume, which decides what amount of fuel that can be burned completely. In order to burn the fuel molecules completely, sufficient amount of air must be sucked in, other wise there is no meaning in injecting large quantity of fuel. Diesel engines are being designed in varies sizes for different applications. Size of diesel engine depends on engine design parameters such as peak combustion pressure and temperature. The power out put of the diesel engine depends on displacement volume of the engine. More the displacement volume more is the amount of burning and hence more is the power output. Peak pressure and temperature in a diesel engine cycle depends on amount of burning. Diesel engines are normally of four or six cylinder engines. Engine size is directly proportional motive power needed. Diesel engines, normally, burn large quantity of fuel and air as they are primarily used in commercial utility vehicles such buses, trucks, Lorries, ships and locomotives etc. Diesel engines are normally multi cylinder types because of the fact that power output is the prime aim rather than aimed at emissions norms.

1.1 Engine combustion cycles

Typical diesel engine combustion cycles have been below [1].

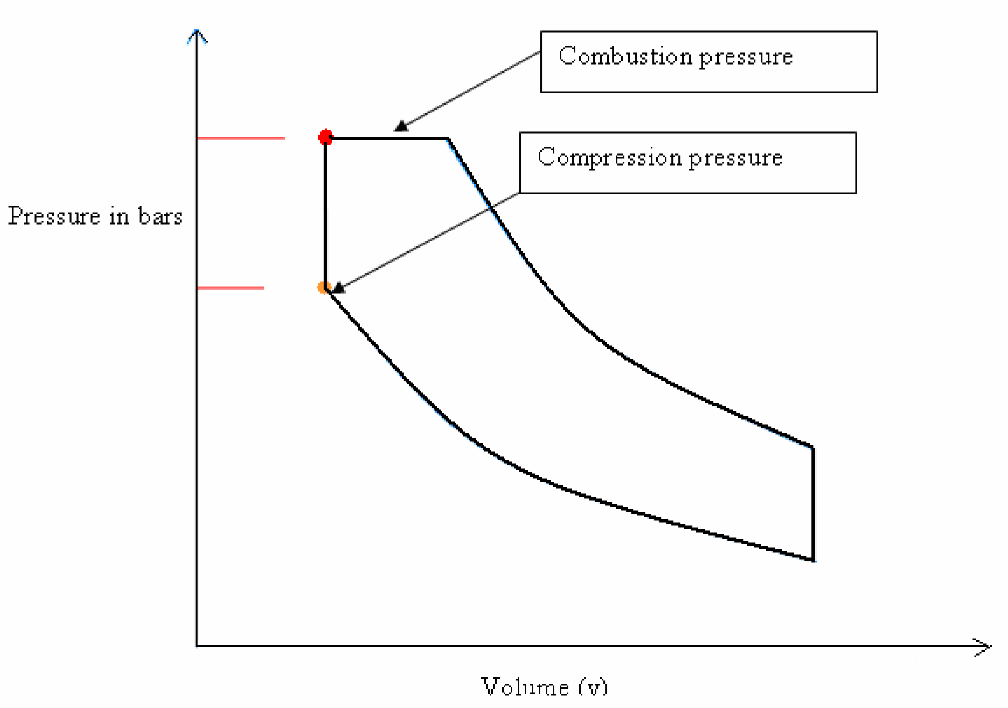


Figure 1.1 Pressure-Volume diagrams [Source: Reference 1]

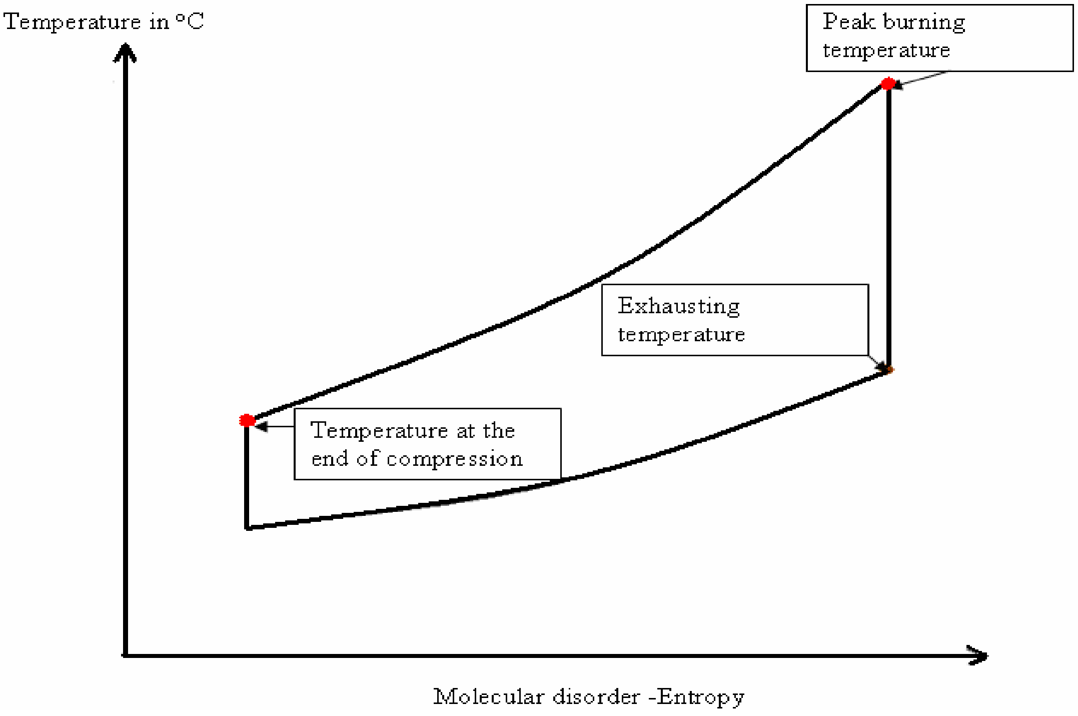


Figure 1.2 Temperature-entropy diagrams

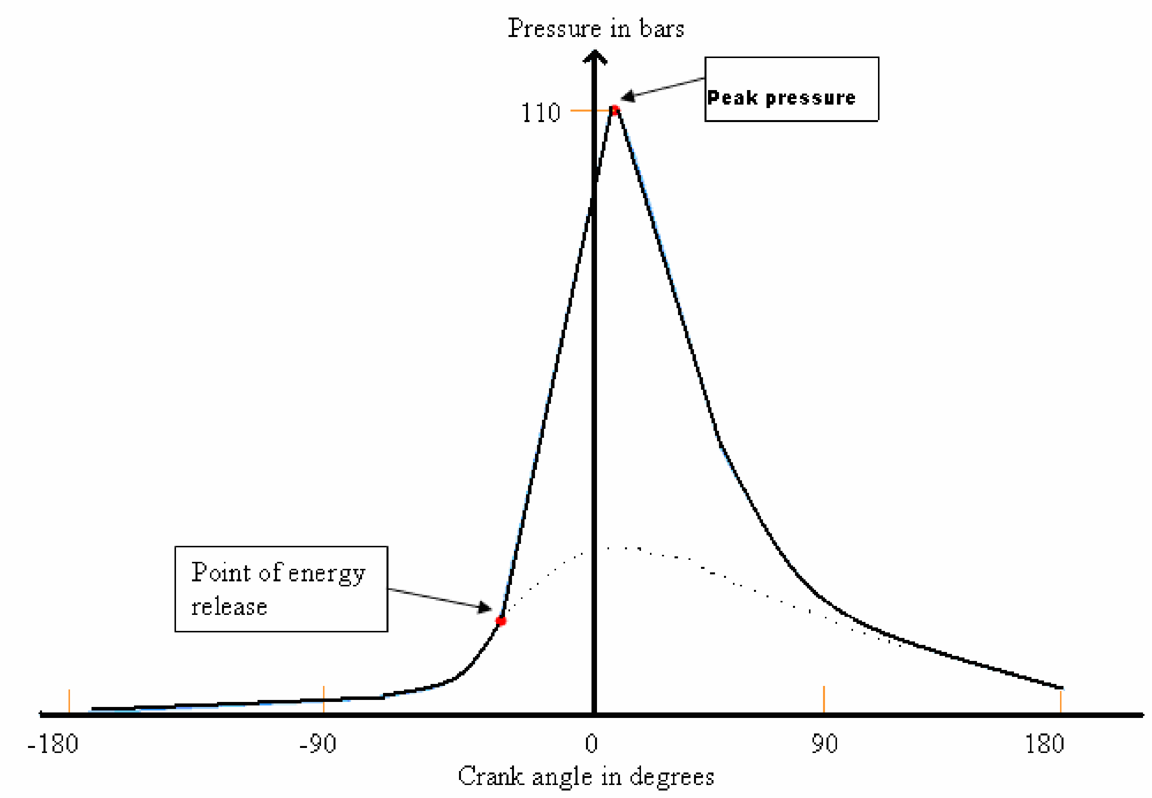


Figure 1.3 Pressure-Crank angle plots

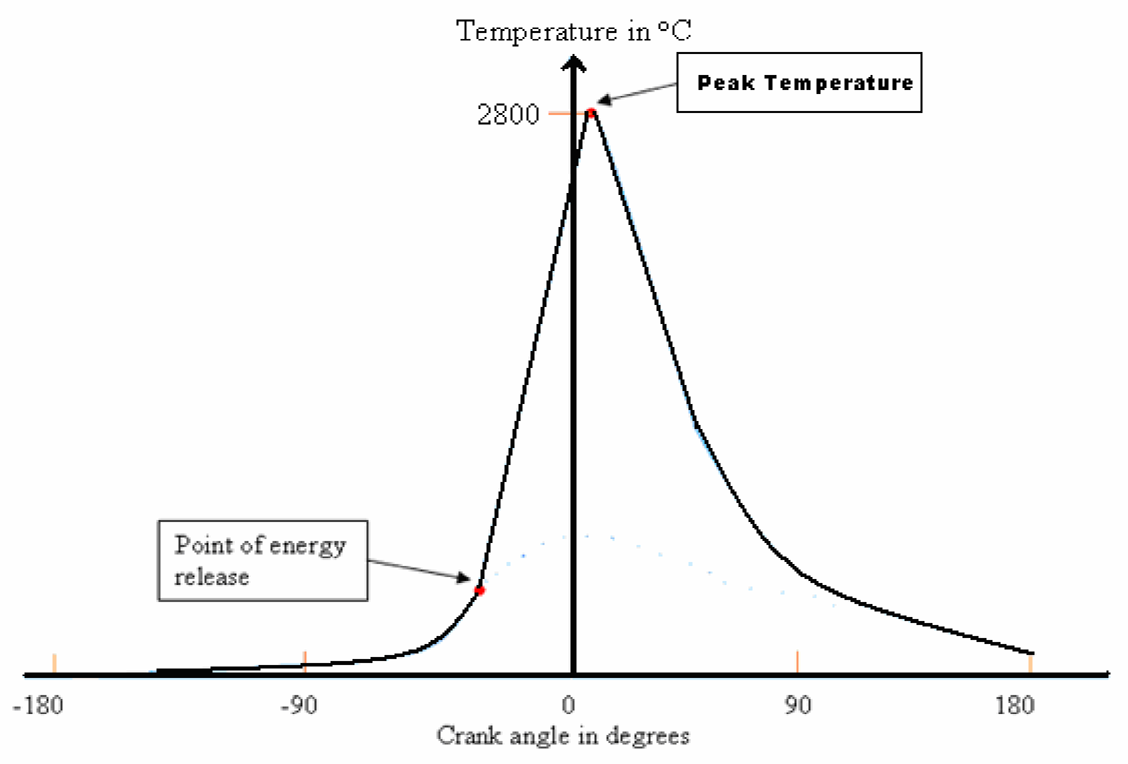


Figure 1.4 Temperature-Crank angle plots [Source: Reference 1]

Peak combustion pressures in typical diesel engines are 120, 110, 100, 90, and 80 bars depending upon the motive power needed and the field of application. Typical diesel engine peak combustion temperatures are 3000 ^{o}C, 2800 ^{o}C, 2500 ^{o}C, 2000 ^{o}C, and 1800 ^{o}C depending upon the field of applications. The typical values are higher for supercharged engines. Higher the peak combustion pressure, greater the amount of power output. Higher the peak combustion temperature, better the energy conversion process. Therefore, there is no limitation, thermodynamically speaking, on the peak pressure and temperature that an engine cycle can deal with. But the limitations are due to material strength. The physical effect of combustion pressure is to strain the engine components and physical effect of temperature is to heat the engine components and sometimes even to the extent of melting. But each material element that we found in this universe has some melting point and strength. Internal combustion engine components are designed for a particular amount of peak load and temeparture.When operating pressures and temperatures exceeds the design limits, the engine designer is not responsible for failure of the component. There fore, the peak combustion pressure and temperature are design parameters. Higher the combustion pressure, greater the amount of stresses on the engine components. Piston is subjected to both structural and thermal loads. Engine block is subjected to both structural and thermal loads. Engine head is also subjected both structural and thermal loads. Connecting rod is subjected to fatigue loads as it takes compression load during expansion and tensile force during suction strokes. Crank shaft is subjected to torsional and bending loads etc. Exhaust and intake manifold are subjected to heat and fluid flow loads.

Internal combustion engines at best can transform about 25 to 35 percent of the chemical energy in the fuel into mechanical energy [1, 3, 6, 4, 7]. About 35 percent of the heat generated is lost to the cooling medium, remainder being dissipated through exhaust and lubricating oil. During the process of combustion, the cylinder gas temperature often reaches quite a high value. A considerable amount of heat is transferred to the walls of the combustion chamber. Therefore, it is necessary to provide proper cooling especially to the walls of the combustion chamber. Due to prevailing high temperatures, chemical and physical changes in the lubricating oil may occur. This causes wear and sticking of the piston rings, scoring of walls or seizure of the piston. Excessive cylinder wall temperatures therefore will raise the operating temperature of the piston head. This in turn will affect the strength of piston seriously. In addition overheated cylinder head may lead to overheated spark plug electrodes causing pre ignition. The exhaust valve may become overheated hot enough to cause pre ignition or may fail structurally. Moreover, pre ignition can increase the cylinder head temperature to the extent of engine failure. As the last part of the charge to burn is in contact with the walls of the combustion space during the burning period, a high cylinder wall or cylinder head temperature will lead to auto ignition in SI engines. In view of the above, the inside surface temperature of the cylinder walls should be kept in a range which will ensure correct clearance between parts, promote vaporization of fuel, keep the oil at its best viscosity. Therefore, heat that is transferred to the walls of the combustion chamber is continuously removed by employing cooling medium. Almost 30 to 35 per cent of the total heat supplied by the fuel is removed by the cooling medium [1, 3, 4, 6,]. Heat carried away by the lubricating oil and heat lost by radiation amount to 5 percent of the total heat supplied. Unless engine is adequately cooled, engine seizure will result. Piston crown is exposed to very high temperatures. The peak cylinder gas temperature may be 2800 ^{o}C while the temperature of the cylinder inner surface may be only 450 ^{o}C due to cooling.

1.2 What current study is all about

In the current study attempt has been made to analyze the working conditions of the **prime components** (piston, engine block, connecting rod, exhaust manifold and exhaust muffler) of an internal combustion engine using Finite Element Analysis concepts. Static stress analysis and steady-state heat transfer analysis were performed on piston. Static stress analysis and steady-state heat transfer analysis were performed on engine block .Static compressive stress analysis was carried out on connecting rod. Computational fluid flow analysis was performed on exhaust manifold and exhaust muffler. All the above analyses were carried out with the help of using **CATIA V5** and **ANSYS**. **CATIA V5** is a solid modeling tool. Since "**Finite Element Method**" procedure needs solid of the component to be analyzed, solid modeling of prime components was done in CATIA V5.**ANSYS** is basically an analyzing system, with which any physical working conditions can be simulated virtually [74]. The components that were modeled in CATIA V5 were brought in to the ANSYS analyzing environment using CATIA-ANSYS interface. The very basic intention is to simulate the working conditions of the engine components **virtually** for different loading conditions,for different component geometries, and for different component materials. Analysis is concerned with diesel engines and their design aspects.

1.3 Objectives of this finite element study

- To find out peak stress in the **piston** for different loading conditions
- To find out peak surface temperature in the **piston** when there is no cooling mediums such water and oil
- To find out peak surface temperature in the piston when cooling mechanism is simulated (convection water and oil cooling)
- To find out peak stress in the **engine block** for different combustion pressures
- To find out peak surface temperature in the **engine block** when there is no cooling mediums such water and oil
- To find out peak temperature in the **engine block** when cooling mechanisms are simulated
- To find out peak compressive stresses in the connecting rod, induced due to gas forces
- To visualize how the flow of exhaust products take place through exhaust **manifold** for different exhausting velocities.
- To visualize how the flow of exhaust products take place in exhaust **muffler** for different exhausting velocities.

CHAPTER-2

FUNDAMENTALS OF FINITE ELEMENT METHODS (FEM)

Finite element method is a numerical procedure for analyzing structures and continuum. Continuum is nothing but physical domain to be analyzed. Usually problem addressed is too complicated to be solved by classical methods [12]. The problem may concern stress analysis, heat conduction, or any of several other areas. The finite element method procedure produces many simultaneous algebraic equations, which are generated and solved on a digital computer. Finite element analysis calculations are performed on personal computers, mainframes, and all sizes in between. Results are rarely exact. The finite element method originated as a method of stress analysis. Today finite elements analysis is used to analyze problems of heat transfer, fluid flow, lubrication, electric and magnetic fields, and many others. Finite element procedures are used in the design of buildings, electric motors, heat engines, ships, air frames, and spacecraft [12]. Manufacturing companies and large design offices typically have one or more large finite element programs in-house.

2.1 Fundamental idea of FEM

It can be imagined that displacement of the right end of the bar shown below is desired.

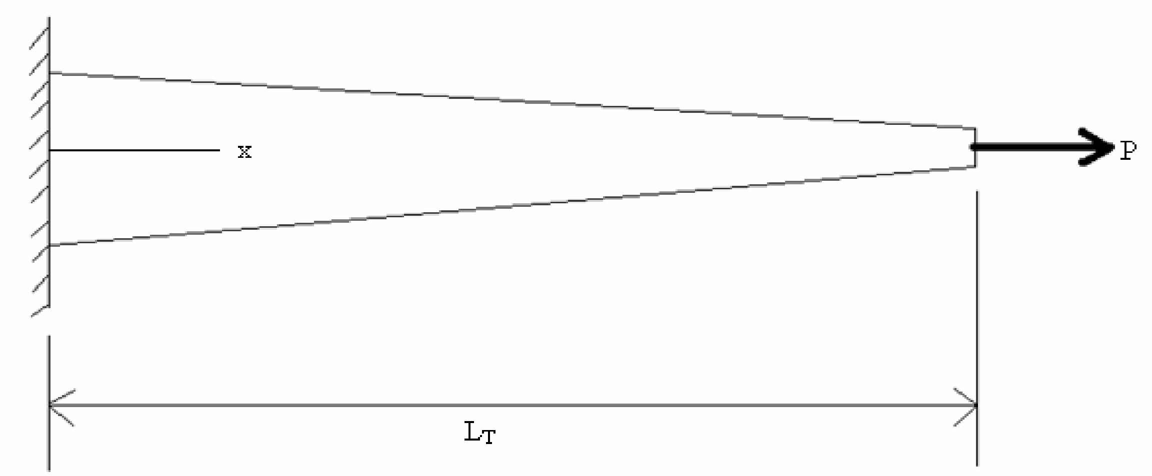


Figure 2.1 A tapered bar under end load P

The classical approach is to write the differential equation of the continuously tapered bar, solve this equation for axial displacement u as a function of x, and finally substitute x = L_T to find the required displacement [12]. The finite element approach to this problem does not begin with differential equation. Instead, the bar is divided by modeling it as a series of *finite* elements, each uniform but of a different cross sectional area A. In each element, u varies linearly with x; therefore **u** is a piece wise smooth function of x.

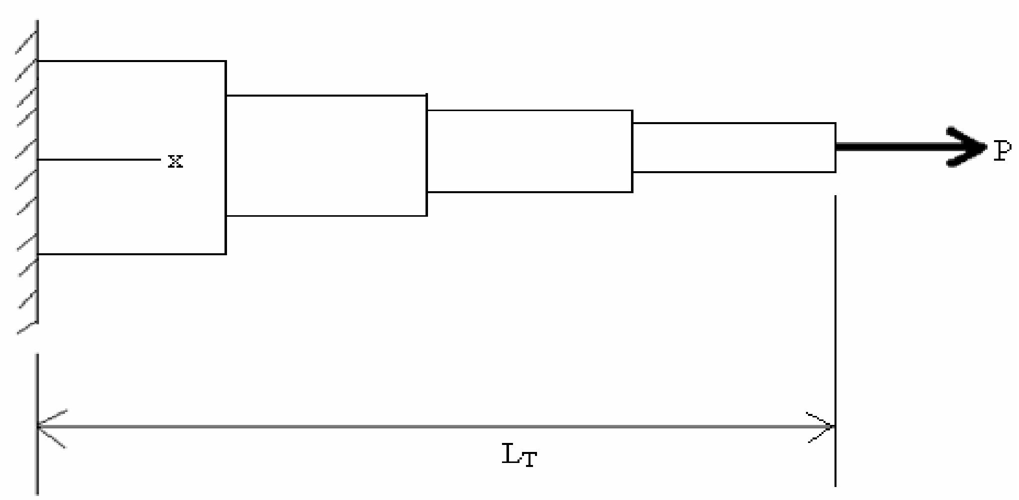


Figure 2.2 a model built of four uniform elements of equal length.

The elongation of each element can be determined from the elementary formula

Element deformation = **PL/AE**

Where P = load applied in N, L = Length of a small element in m.

A= cross-sectional area of an element in m^2

E = Modulus of elasticity (stiffness of the material) in N/ m^2

The end displacement, at x = L_T is sum of the element elongations. Accuracy improves as more elements are used. In the foregoing discussion finite element method models a structure as an assemblage of small parts (elements) [12].Each element is a simple geometry and therefore is much easier to analyze than the actual structure. In essence, we approximate a complicated solution by a model that consists of piecewise-continuous simple solutions. In a heat transfer context, Fig.2.1 might represent a bar with insulated lateral prescribed temperature at the left end, and prescribed heat flow at the right end. One might ask for temperature in the bar as a function of x. The *finite element method* is a method of piecewise approximation in which approximating function is formed by connecting simple functions, each defined over a small region. A *finite element* is a region in space in which a function is interpolated from nodal values of function on the boundary of the region in such a way that inter element continuity is maintained.

2.2 Typical finite element method procedure [12]

1. Divide the structure or continuum into finite elements. Mesh generation programs, called preprocessor, help the user in doing this work.
2. Formulate the properties of each element
3. Assemble the elements to obtain the finite element model of the structure.
4. Apply known loads: apply known loads and/or moments in stress analysis, nodal heat fluxes, convection loads in heat transfer.
5. In stress analysis, specify how the structure is supported. This step involves setting several nodal displacements to known values (which often are zero).In heat transfer analyses, where typically certain temperatures are known, impose all known values of nodal temperature.
6. Solve simultaneous linear algebraic equations to determine nodal degree of freedom (nodal displacements in stress analysis, nodal temperatures in heat transfer)
7. In stress analysis, calculate element strains from the nodal displacements and finally calculate stresses from strains. In heat transfer analysis, calculate element heat fluxes from the nodal temperatures. Output interpretation programs, called post processors, help the user sort the output and display in graphical form.

The power of the finite element method resides principally in its versatility. The method can be applied to various physical problems. **The structure can have arbitrary shape, loads, and support conditions**. Mesh can mix elements of different types, shapes, and physical properties. This great versatility is contained within a single computer program. User input data controls the selected problem type, geometry, boundary conditions, element selection, and so on. Another attractive feature of finite elements is the close physical resemblance between the actual structure and finite element method. The finite element method also has disadvantages [12]. A specific numerical result is found for a specific problem: A finite element analysis provides no closed form solution that permits analytical study of the effects of changing various parameters. A computer, a reliable program, and intelligent use are essential. Experience and good engineering judgments are needed in order to carry out real analysis [12].

2.3 What is FEA?

- *Finite Element Analysis* is a way to simulate physical loading conditions on a design and determine the design's response to those conditions [12, 75].
- The design is modeled using discrete building blocks called *elements*.

Each element has exact equations that describe how it responds to a certain load.

The "sum" of the response of all elements in the model gives the total response of the design.

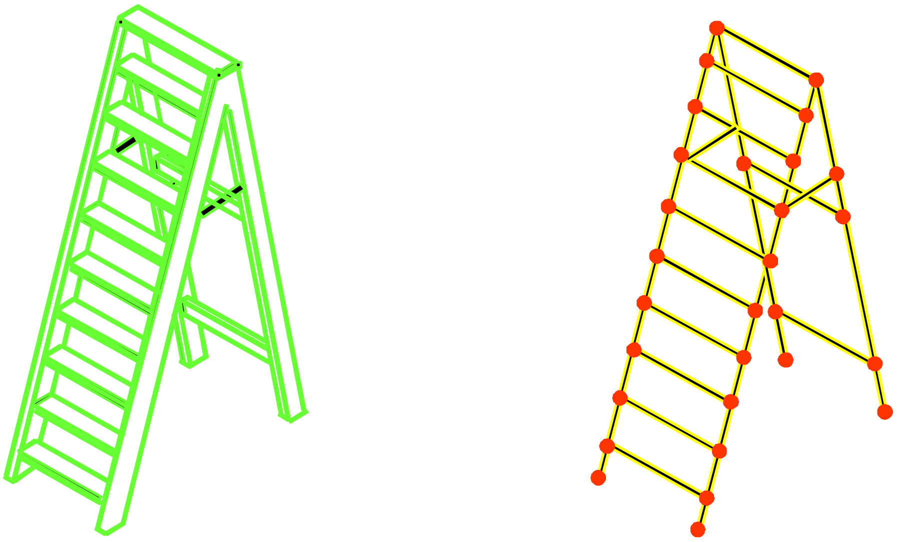
The elements have a finite number of unknowns, hence the name *finite elements*.

Historical Note

- **The finite element method of structural analysis was created by academic and industrial researchers during the 1950s and 1960s.**
- **The underlying theory is over 100 years old, and was the basis for pen-and-paper calculations in the evaluation of suspension bridges and steam boilers.**

- The finite element model, which has a *finite* number of unknowns, can only *approximate* the response of the physical system, which has *infinite* unknowns.
  - So the question arises: *How good is the approximation?*
  - Unfortunately, there is no easy answer to this question. It depends entirely on what is being simulated and the tools that are being used for the simulation.

The following figure shows the physical structure to be analyzed. The physical structure to be analyzed is ladder which is subjected to weight of the applicant. The physical system to be analyzed, then, divided into finite elements and they are connected at the common joints so called “**nodal points**”. An element is the building block of a finite element model which exactly represents the behavior of the physical structure to be analyzed. Each element has exact equations that describe how it responds to a certain load. The “sum” of the response of all elements in the model gives the total response of the design. The elements have a finite number of unknowns, hence the name *finite elements*. More the number of elements, greater is the accuracy of the result.


Physical system

Finite element model

Figure 2.3 Fundamental idea of FEM

2.4 Conversion of Solid model into Finite element model

Finite element method requires finite element model that has element, nodes and it does work on solid model. Therefore solid structure to be analyzed should be converted into finite element model. Finite element model can be obtained by dividing the structure into finite elements and then connecting them together as solid structure is physically continuous. There are variety of element types depending on the type of analysis and physical problem. The element selection has dramatic influence on the result obtained. The type of element and its capability should be very well known to the analyst before carrying out any analysis. The following figure shows how finite element model is obtained.

Meshing is the process used to “fill” the solid model with nodes and elements, i.e., to create the FEA model [74]. An element is a building block of a finite element model, which exactly represents the behavior of the physical structure to be analsyed.There are different types of elements conceptualized as per the type of analysis geometry of the structure such as1-D, 2-Dimensional analysis or 3-Dimensional analysis. Solid stress analysis requires 3-D element.

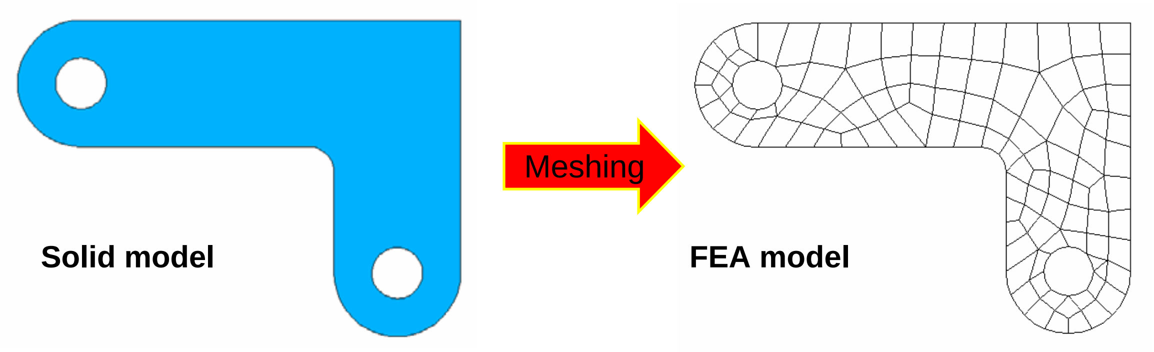


Figure 2.4 Conversion of solid model into FEA model.

2.5 Why is FEA needed?

- To reduce the amount of prototype testing
  - Computer simulation allows multiple “what-if” scenarios to be tested quickly and effectively [74, 75, 95, 12].
- To simulate designs that are not suitable for prototype testing
  - Example: Surgical implants, such as an artificial knee
- The bottom line:
  - Cost savings
  - Time savings… reduce time to market!
  - Create more reliable, better-quality designs

2.6 Types of FEA analysis

There are so many types of analysis that can be performed by finite element concepts in context to mechanical engineering [74, 75, 95, 12]. They are

- Structural analysis
- Thermal analysis
- Computational fluid dynamics analysis
- Coupled physics (coupling of thermal-structural, coupling of fluid –thermal)

2.6.1 Structural Analysis:

Structural analysis is used to determine deformations, strains, stresses, and reaction forces.

- Static analysis
  - Used for static loading conditions.
  - Nonlinear behavior such as large deflections, large strain, contact, plasticity, hyper elasticity, and creep can be simulated.

2.6.2 Thermal Analysis:

- Thermal analysis is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux.
- All three primary heat transfer modes can be simulated: conduction, convection, radiation.
  - Steady-State
    - Time-dependent effects are ignored.
  - Transient
    - To determine temperatures, etc. as a function of time.
    - Allows phase change (melting or freezing) to be simulated.

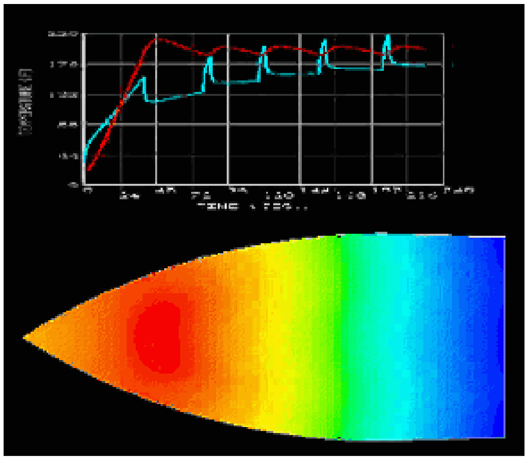


Figure 2.5 Transient Temperature of a Warming Clothes Iron

2.6.3 Computational Fluid Dynamics (CFD)

- To determine the flow distributions and temperatures in a fluid.
- FEA can simulate laminar and turbulent flow, compressible and incompressible flow, and multiple species [74, 75, 95].
- Applications: aerospace, electronic packaging, automotive design
- Typical quantities of interest are velocities, pressures, temperatures, and film coefficients.
- Main objective is to find out pressure, velocity, and temperature of flowing fluid at each and every point in the fluid flow volume.

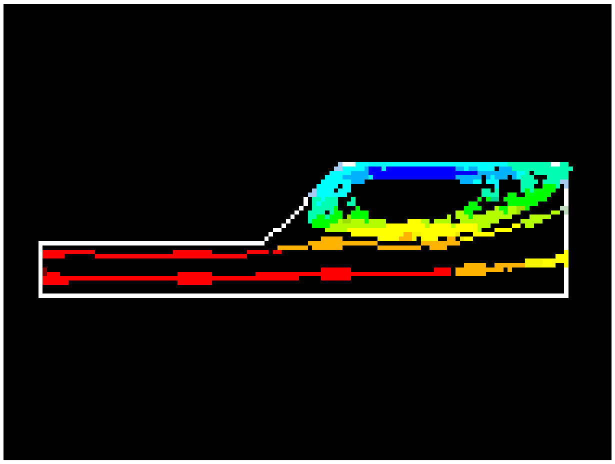


Figure 2.6 Flow of fluid through a Duct

2.7 Summary

- Finite element method is a numerical procedure for analyzing structures and continuum. Continuum is nothing but physical domain to be analyzed
- *Finite Element Analysis* is a way to simulate loading conditions on a design and determine the design's response to those conditions.
- The finite procedure produces many simultaneous algebraic equations, which are generated and solved on a digital computer
- Structural analysis is used to determine deformations, strains, stresses, and reaction forces
- Thermal analysis is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux
- CFD is used to determine the flow distributions and temperatures in a fluid

CHAPTER – 3

Finite Element analysis of Piston

(Structural and thermal aspects)

Piston is a cylindrical component fitted into the cylinder forming the moving boundary of the combustion system. It fits perfectly into the cylinder providing gas tight space with the piston rings and the lubricant. It forms the first link in transmitting the gas forces to the output shaft. Piston is a simple machine element which forms the combustion chamber as well as it is the one which receives the combustion thrust which is to be transferred to the crankshaft via connecting rod. Piston transfers the gas load from cylinder to the connecting rod which in turn transfers the load to the crankshaft in order to obtain mechanical energy [4, 5, 6]. It will be able to transfer the load to the connecting rod only when it is able to sustain the load that is applied on it. Apart from the gas load transfer, it has to conduct the necessary amount of heat to the cooling mediums in order to avoid material melting. At the same time it has to retain enough amount of heat to sustain combustion and also to provide good thermal efficiency. Therefore piston should be capable to sustain both thermal and structural distortions that are induced due to gas force and combustion energy.

Figure 3.1 Typical diesel engine pistons

The following are the important functions of the piston [1, 4, 5, 6]

- It forms the combustion chamber in which burning is taking place.
- It is the member which receives the combustion thrust
- It provides means by which gas force can move through certain fixed distance in order to do work
- It takes the structural stresses induced due to the gas pressure
- It takes the thermal stresses induced due to the heat load

Pistons are normally made up of alloy steels or cast iron or aluminum because piston material should be a good heat transporter and light in weight, which are the prime requirements of a typical internal combustion piston. Aluminum (alloy 1100) is used as a piston material because it is a good heat transporter (high thermal conductivity k =222 W/m K) and one of the less dense material. But it is structurally weak when compared to other materials.

3.1 What is the need to conduct FEA analysis of piston?

These are the possible reasons that necessitate conducting FEA analysis on piston.

1. To check whether or not piston takes the structural stress induced due to gas load
2. To check whether or not piston material takes the heat load (fuel energy released at the point of combustion)
3. Is thermal stress severe?
4. Is piston geometry optimized enough to take the loads?

The above stated questions can be answered only when FEA analysis is carried out on the piston.

3.2 Physical problem- structural load

Diesel engine is a smoky and high mass burning engine. The peak combustion pressure at the point of combustion of a typical diesel engine is in the range of 70 – 120 bars depending upon the application. This pressure is a design parameter. This peak combustion pressure induces the compression stress in the piston. The physical problem is shown below [1, 4, 5,6]. The physical problem is that combustion pressure is acting on the crown of the piston while is supported by the connecting rod with the help of piston pin at the piston pin bosses. Though piston is not rigidly fixed, connecting rod provides necessary support to the piston take the gas load.

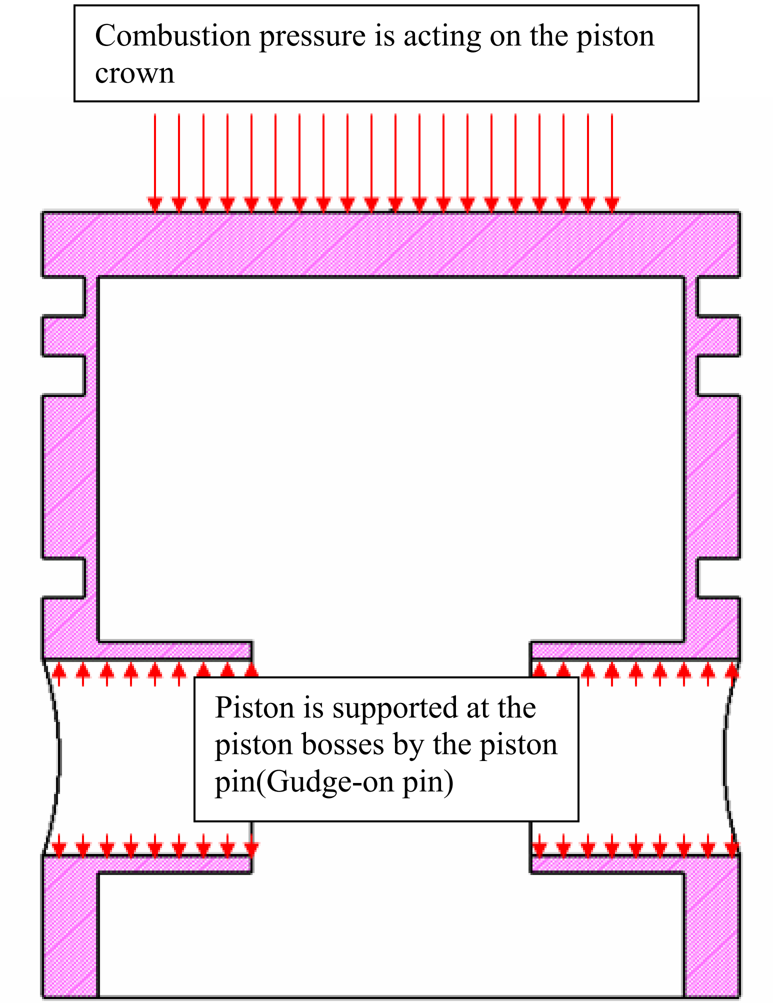
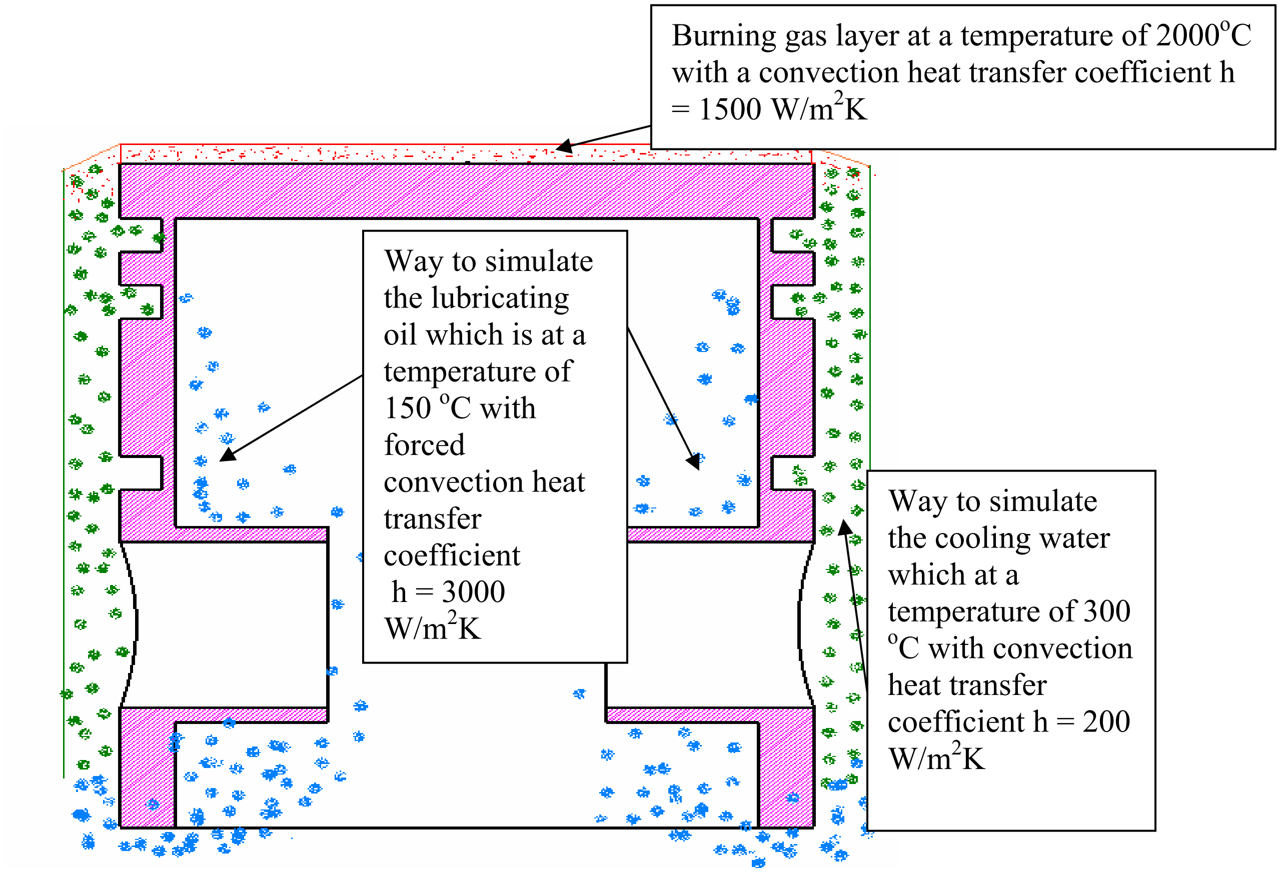


Figure 3.2 Statement of the physical problem-structural load

The main objective of conducting FEA analysis on the piston is to assertion whether or not engine piston takes the structural stress induced due to gas pressure. When the stresses induced due to peak combustion pressures are greater than the yield point of the material, it is very obvious that piston will fail structurally [28]. So, the ultimate objective is to find out what is the maximum stress induced in the piston and then to compare with the materials yield strength or design strength.

3.3 Physical problem- thermal load

Combustion is taking place in a space formed by cylinder surfaces, piston top surface (crown), and by the cylinder head [1, 4, 5, 6]. Cylinder wall is the ultimate heat path for transferring heat from the cylinder inner surface to the cooling medium. At the point of combustion, moving gas comes into contact with a wall; there exists a relatively stagnant gas layer which acts as a thermal insulator. Heat transfer from the cylinder gases takes place through cylinder walls to the cooling medium. A large temperature drop is produced in the stagnant gas layer adjacent to the walls and also on the piston top surface. Physical problem to be analyzed is shown below. Combustion gas forms a thin layer of high temperature burning layer on the piston top surface. Very practically speaking piston does have any connection to the cooling water in the water jacket, but virtually speaking piston has a strong connection with the cooling water as the temperature difference between piston and cooling water, physically pulls the heat from the combustion chamber to the water jacket. In a high burning engine such as diesel engine, the peak cylinder gas temperature may be 2800 K while the temperature of the inner wall surface may be only 450 K due to cooling [1, 3, 4, 6, 7]. Heat is transferred from the gases to the piston and cylinder walls when the gas temperature is higher than the cylinder wall temperature.



Burning gas layer at a temperature of 2000^{o}C with a convection heat transfer coefficient h = 1500 W/m^{2}K

Way to simulate the lubricating oil which is at a temperature of 150 ^{o}C with forced convection heat transfer coefficient h = 3000 W/m^{2}K

Way to simulate the cooling water which at a temperature of 300 ^{o}C with convection heat transfer coefficient h = 200 W/m^{2}K

Fig 3.3 Statement of the physical problem- thermal load

The rate and direction of flow of heat varies depending upon the temperature differential. If no cooling is provided, there could be no heat flow, so that whole piston would soon reach an average temperature of the cylinder gases and if this temperature is more than the melting temperature of the piston material, it will have to melt. So, the ultimate objective of conducting FEA analysis on piston is to find out what are the peak material temperature and its distribution due to heat load and also to plot thermal gradient and heat flux [35, 45, 46]. There are three major kinds of analysis normally performed on piston. They are

- Static structural stress analysis
- Dynamic stress analysis
- steady state heat transfer analysis (To obtain temperature distributions, heat flow rates, heat flux and temperature gradient)
- Transient heat transfer analysis (To obtain temperature distributions with respect to time , time heat flow rates, heat flux and temperature gradient)

- Thermal stress analysis (due to temperature gradient and constraining)

In the current study, only static structural stress and steady state heat transfer analysis were performed.

3.4 Approach of the analysis

Although piston needs **coupled structural- thermal analysis** in order to predict the thermal stresses in the engine block, in the current study both the analysis were treated separately because of the fact that engine block is not rigidly fixed and also fact that thermal stresses are very minimum compared to the compressive stress. Therefore following approach was used to carry out the analysis.

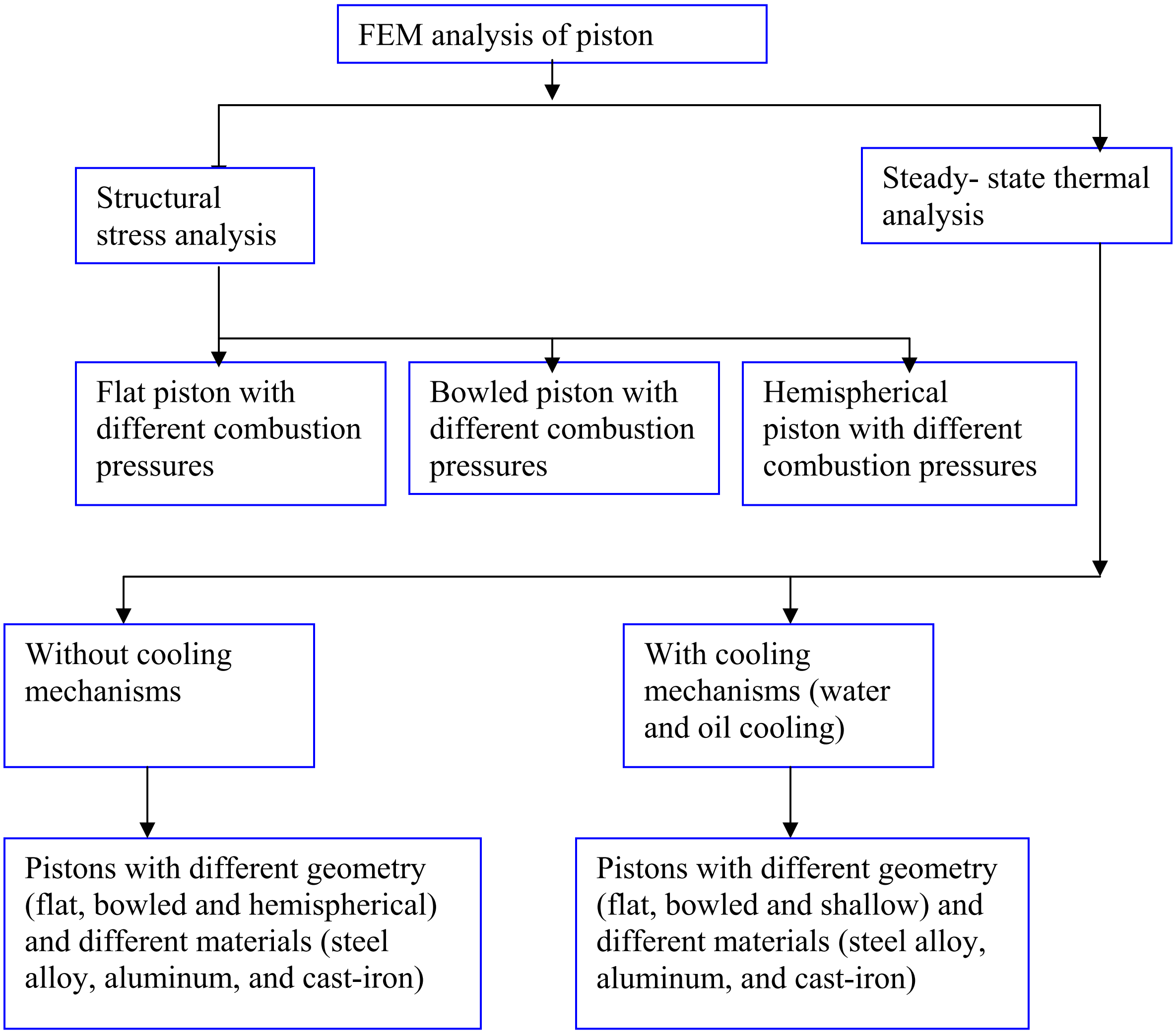


Figure 3.4 Approach of the analysis

3.5 Inputs required to carrying out the above analysis

- Approximate geometry of the piston
- Materials properties of the piston material such as [5]
  - Young’s modulus of alloy steel -1040 (E = 207 GPa)
  - Young’s modulus of aluminium alloy -6061 (E = 69 GPa)
  - Young’s modulus of cast iron Grade 80-55-06 (E = 168 GPa)
  - Poisson’s ratio of steel material (μ=0.3)
  - Thermal conductivity of the alloy steel (k = 52 W/mK)
  - Thermal conductivity of the cast iron (k = 46 W/mK)
  - Thermal conductivity of the aluminum (k = 180 W/mK)
  - Yield strength of the alloy steel (1040) S = 375 N/m^2
  - Yield strength of the aluminium alloy(6061) S = 276 N/m^2
  - Strength of the cast iron(80-55-06) S = 379 N/m^2

3.6 Typical FEM procedure to carry out the this analysis [12]

- **Geometry** creation (solid modeling of piston) [12, 74, 75, 95]
- Dividing the solid model into number of elements and then connecting these elements with each other (**meshing**) and thereby obtaining the finite element model.
- Applying boundary conditions
  1. for **structural** analysis-boundary conditions means defining combustion pressures and fixing constraints
  2. for **thermal** analysis – boundary conditions means defining gas convection load , convection water cooling load, and convection oil cooling load.
- Solving the problem
- Results and their interpretation.

3.7 Geometry creation and meshing

Geometry of the piston is needed to carry out this analysis. Solid model of the engine block was created in solid modeling environment called “**CATIA V5**”.Solid model is the base for carrying out any FEA or CFD analysis. Solid model of the piston that was created in **CATIA V5** solid modeling environment is shown in the figure 3.5. The created solid model was brought in to the ANSYS analysis environment using **ANSYS-CATIA** interface. Solid model used in any FEA analysis can be very simple and at the same time it reflects the physical reality of the piston. Therefore, small small features like rounds, chamfers, holes have not been modeled [74, 75]. Since the analysis to be made on the different geometry, different piston geometries were created. The geometries fundamentally differ by the top portion (piston crown) of the piston. The different piston crowns are flat, bowled, shallow and hemispherical. The very purpose of conducting analysis on different geometries is that different piston crowns would give us different stress profiles from which we can choose the best piston configuration and also that we can optimize the piston geometry. Different piston geometry provides different stresses due to the fact that different area of cross section provides different stress profiles. As shown in the figure below, solid model is very simple but at the same time, it reflects a physical reality. Solid model of the typical piston is complex piece as it has oil holes and ring grooves, piston pin bosses. Piston has so many features like piston pin bosses, and ring grooves. Solid model will have to be turned into to FEM model by dividing the solid model into number of small small elements. This process is called “**discritization**”.Discritization is the process of dividing the solid model into finite number of elements. After dividing the solid model into finite number of elements, they have to be connected to each other as the solid volume is continuous and physically connected at each and every material point.

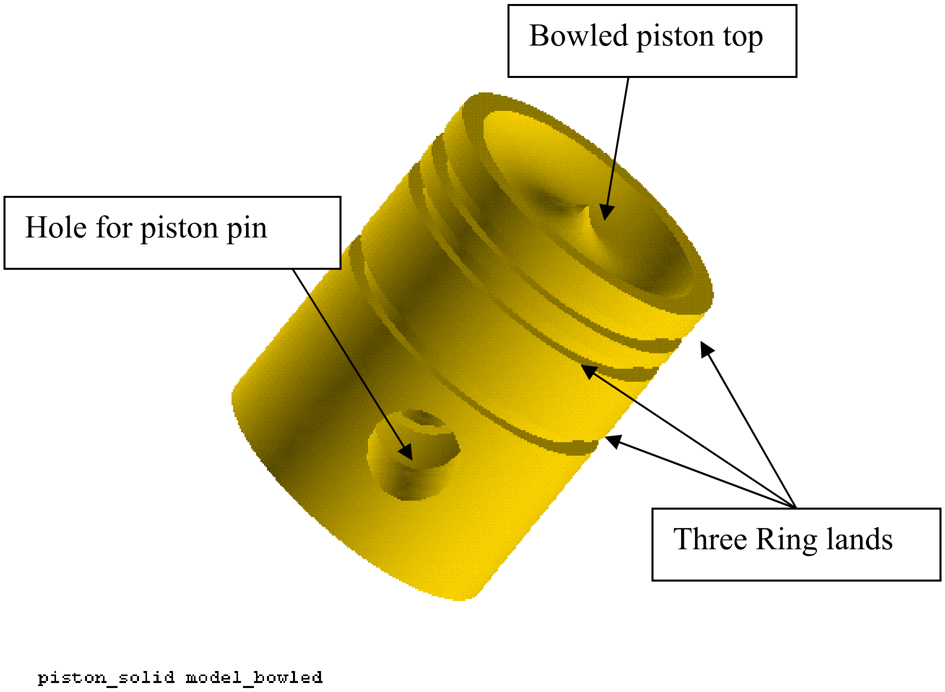


Figure 3.5 Solid model of the piston created in CATIA V5 and then imported into ANSYS analyzing environment using ANSYS-CATIA interface.

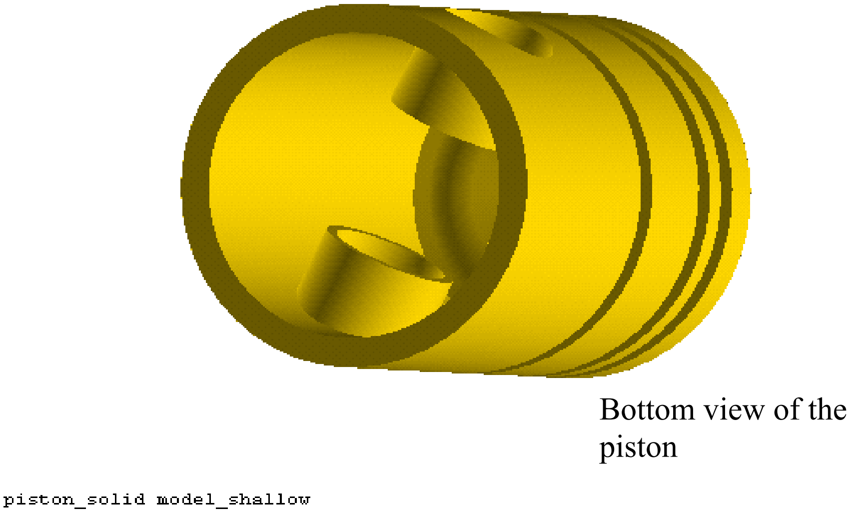
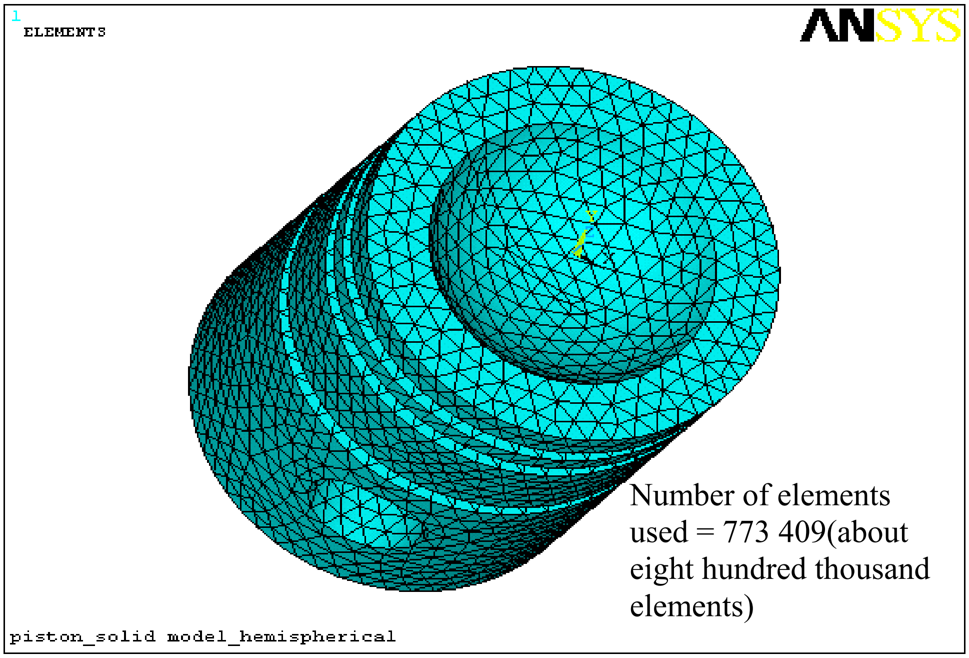


Figure 3.6 Bottom view of the piston created in CATIA V5 and then imported into ANSYS analyzing environment using ANSYS-CATIA interface.

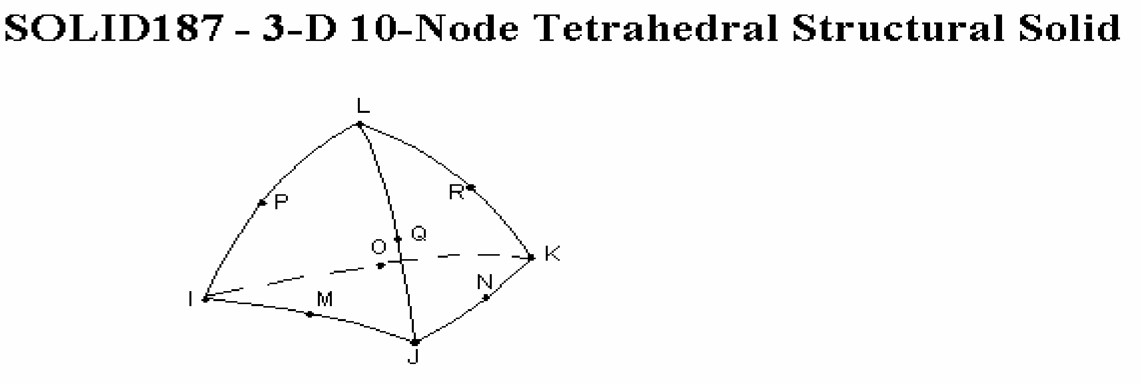
In simple terms meshing means that connecting the elements with each other. An element is the building block of the finite element model. So the type of element (linear element or higher order elements), number of nodes and their capabilities are important parameters for selecting the elements for a particular analysis. Meshed model of the piston is shown below.



Number of elements used = 773 409(about eight hundred thousand elements)

Figure 3.7 Meshed model of the piston

In this analysis, the element used was **Tet 10 node3 SOLID 187** [74] which is a solid element which has all six degrees of freedom (translation in all X, Y, and Z and rotations about all three axes); it also has stress stiffening and buckling capabilities. This element is shown below.



SOLID187 - 3-D 10-Node Tetrahedral Structural Solid

Figure 3.8 Element name – Tet 10 node SOLID 187

Meshing is an important step in FEA analysis. Mesh must be adequate enough to obtain the accurate results while keeping the computational time in mind. More the number of elements, better the accuracy, on the other hand more the number of elements more the computational time. The number of elements used in this analysis was 773 407 (about eight hundred thousands) .In order to obtain accurate information about the temperature gradients and temperature distributions, more number of elements have been placed at the corners, rounds, and at the holes by using ANSYS **smart meshing** capability. Smart meshing is the special feature in ANSYS meshing capability, which places more number of elements at points where stress gradient and strain gradient are more. The ultimate purpose is to obtain more accurate information at the very sensitive places.

3.8 Physical boundary conditions – Structural loading

Structural loading simply means that applying gas pressure loads. The combustion gas pressure was applied on the crown (top surface) of the piston and it is constrained at the piston pin bosses. The gas pressures taken for the study was 120,110, and 100 bars. (Typical peak pressures of a diesel engine).

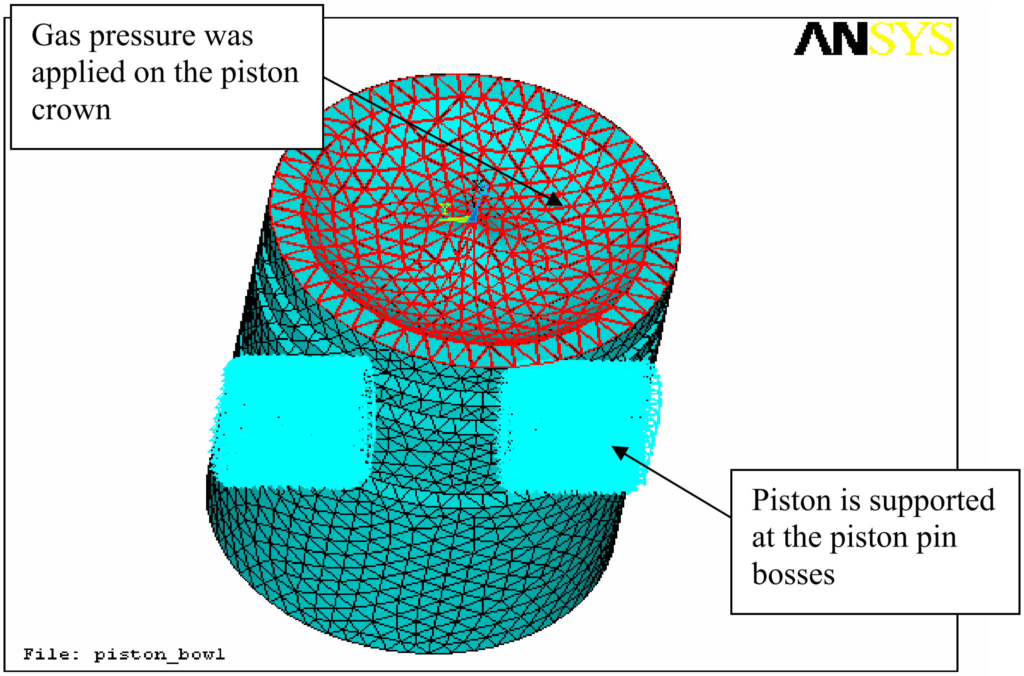
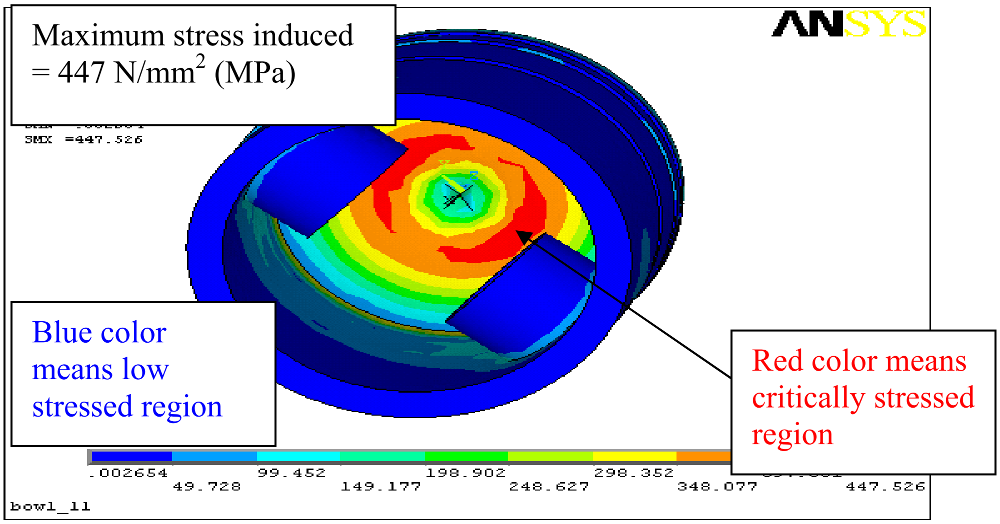


Figure 3.9 Structural boundary condition of the piston when **burning** is taking place.

3.9 Results and their physical interpretation

After applying the boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing structural stress strain equations for each and every element and these formulated governing equations were solved for the deformations from which all the other quantities such stresses, strains etc can be calculated.

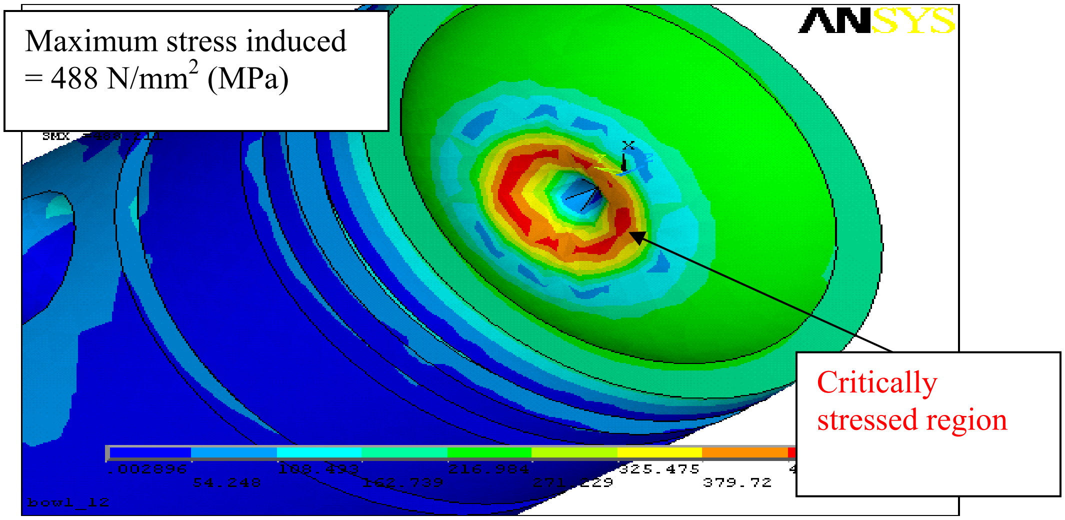


Maximum stress induced = 447 N/mm^2 (MPa)

Blue color means low stressed region

Red color means critically stressed region

Figure 3.10 **Von-mises stress** (equivalent stress) contour of the piston when applied peak combustion pressure = 110 bars on a bowled piston

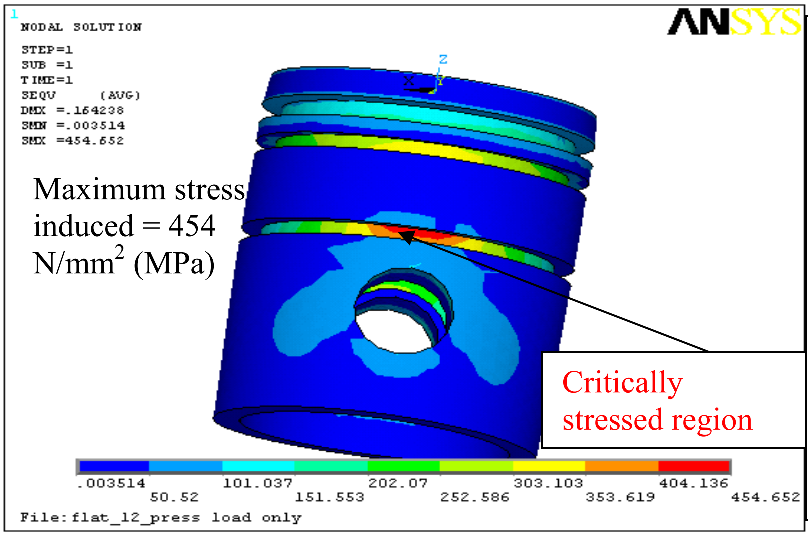


Maximum stress induced = 488 N/mm^2 (MPa)

Critically stressed region

Figure 3.11 **Von-mises stress** (equivalent stress) contour of the piston when applied peak combustion pressure = 120 bars on a bowled piston

As can be seen from the color contours, red color indicates the regions which are severely stressed and blue color indicates the regions of low stresses. It is very clear that tip of the bowled piston subjected to a very high amount stresses. It is because of curvature and stress concentration effects. The curvature effect means that the circumference of the tip is compressed all around by the gas pressure, hence more stress is created is created in that region. In overall view, total top portion is subjected to a very high stresses. As it is physically expected, stress increases when we increase the combustion pressure. The ring grooves in the piston are also subjected to a higher stresses because of the fact area of cross section(small thickness) which has to resist the load is less compared to the other sections of the piston.

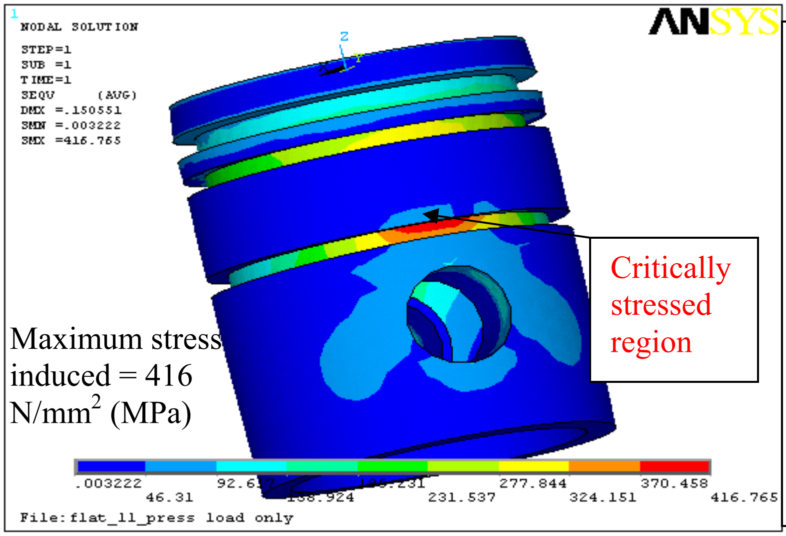


Maximum stress induced = 454 N/mm^2 (MPa)

Critically stressed region

Figure 3.12

Von-mises stress (equivalent stress) contour of the piston when applied peak combustion pressure = 120 bars on a flat piston.



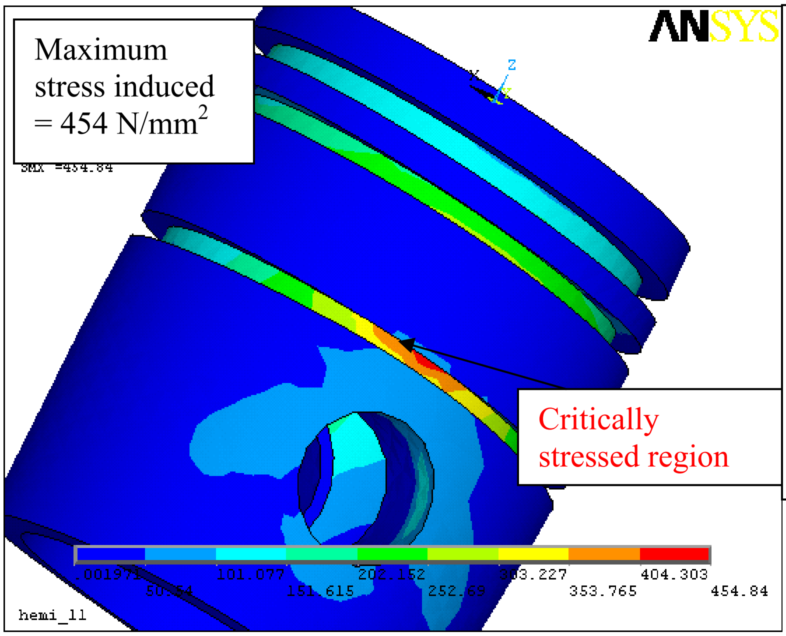
Critically stressed region

Maximum stress induced = 416 N/mm^2 (MPa)

Figure 3.13

Von-mises stress (equivalent stress) contour of the piston when applied peak combustion pressure = 110 bars on a flat piston.

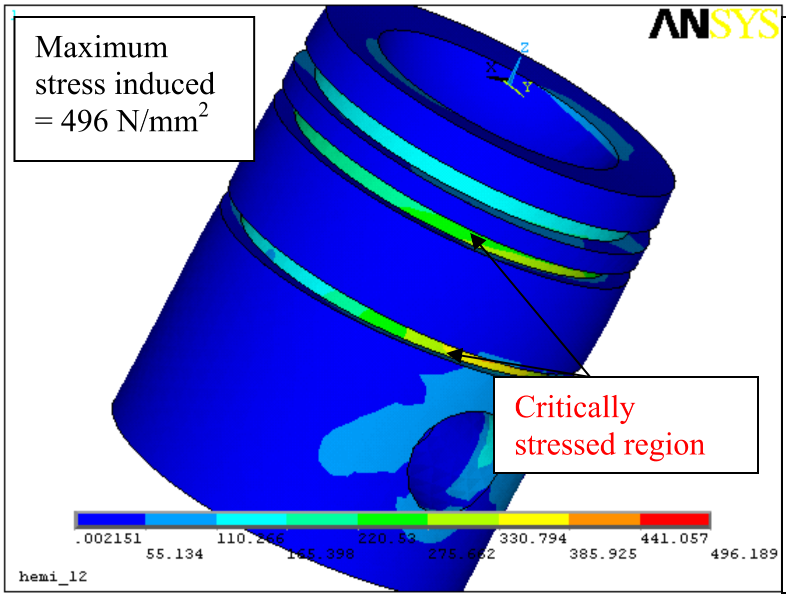
As can be seen from the above plots, since flat piston has more normal area of cross section to resist the load stress in the top portion is less very normal and region of peak stress has been shifted form top to ring grooves. There are no curvature and stress concentration effects as there is tip. In overall view, total ring grooves are severely stressed. As it is physically expected, stress increases when we increase the combustion pressure.



Maximum stress induced = 454 N/mm^2

Critically stressed region

Figure 3.14 **Von-mises stress** (equivalent stress) contour of the piston when applied peak combustion pressure = 110 bars on a hemispherical piston.

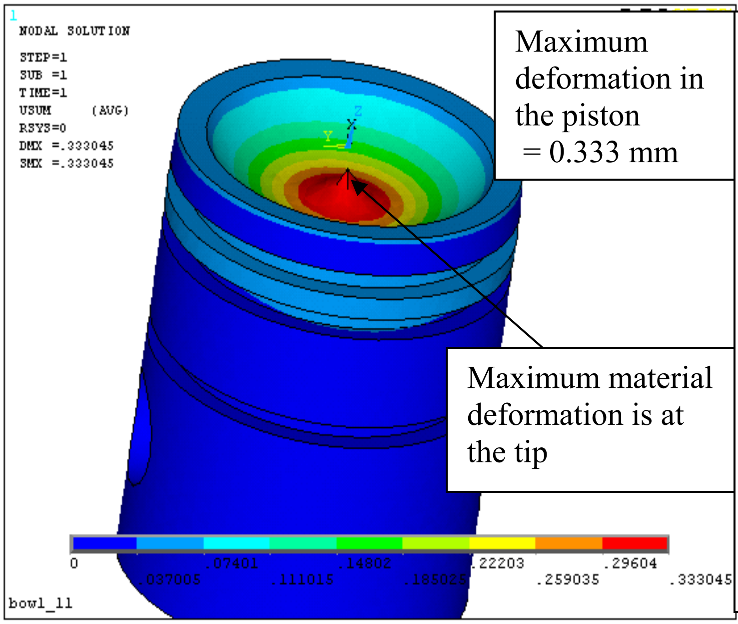


Maximum stress induced = 496 N/mm^2

Critically stressed region

Figure 3.15 **Von-mises stress** (equivalent stress) contour of the piston when applied peak combustion pressure = 120 bars on a hemispherical piston.

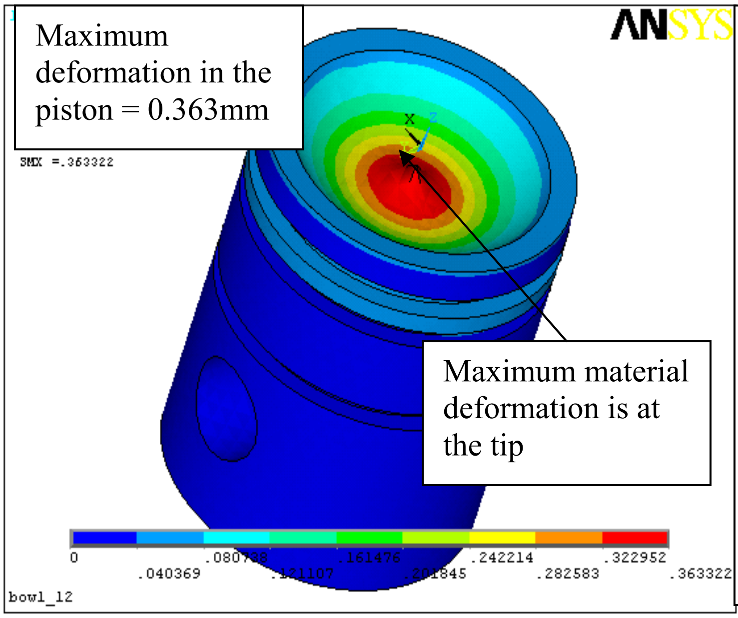
As can be seen from the above plots, since hemispherical piston does not have any protruding tip, there was no high stresses in the top of the piston. Since hemispherical piston sufficient area of cross section to resist the load, region of peak stress has been shifted form top to ring grooves. There are no curvature and stress concentration effects as there is tip. In overall view, total ring grooves are severely stressed. As it is physically expected, stress increases when we increase the combustion pressure. It can be noted that areas where piston is supported is also stressed to a moderate levels.



Maximum deformation in the piston = 0.333 mm

Maximum material deformation is at the tip

Figure 3.16 deformation profile of the piston when applied peak combustion pressure = 110 bars on a bowled piston



Maximum deformation in the piston = 0.363mm

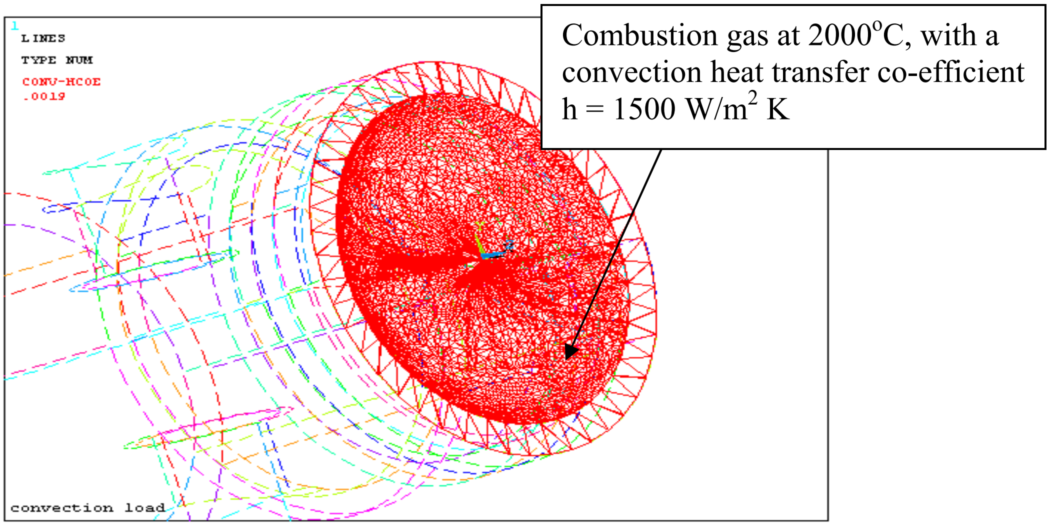
Maximum material deformation is at the tip

Figure 3.17 deformation profile of the piston when applied peak combustion pressure = 120 bars on a bowled piston

Since tip portion of the bowled piston is severely stressed, deformation is also more on that region. The red color indicates the critically deformed regions. The maximum deformation in the material is 0.333, 0.362 for the corresponding pressures of 110,120 bars respectively. As we increase the pressure, deformation is also increases.

3.10 Physical boundary conditions – Thermal loading without cooling

Piston has to retain sufficient amount of heat in order to provide have good thermal efficiency and effective energy conversion. Virtually speaking there is no limit on the surface temperature of the piston but the physically there is a restriction on the maximum temperature that piston surface can have. This restriction is mainly because of the fact each and every engineering material has to melt when the temperature is above the melting point of the material. Thermal boundary conditions are heat energy (gas at very high temperature with high heat transfer co-efficient) released when fuel is burned and convection cooling loads as shown below [36, 4, 3]. The first stage in the thermal study was to simulate the piston without any cooling mediums in order to see the surface temperature of the piston when there is no cooling. It is worthwhile to conduct this analysis because engine designer should fore, see **how his engine** would behave if there is no cooling mediums for the engine that he designed and also that this analysis is the base from which engine designer can calculate how much cooling is necessary to keep the engine operating at its best performance state. Typical internal combustion engine leaves about 30% of the combustion energy to the cooling water. The boundary conditions are combustion gas is at a temperature of 2000 ^{o}C [36] with a convection heat transfer coefficient of 1500W/m^{2}K and Since bottom of the piston is exposed to crankcase, average crankcase temperature is specified at the bottom of the piston.

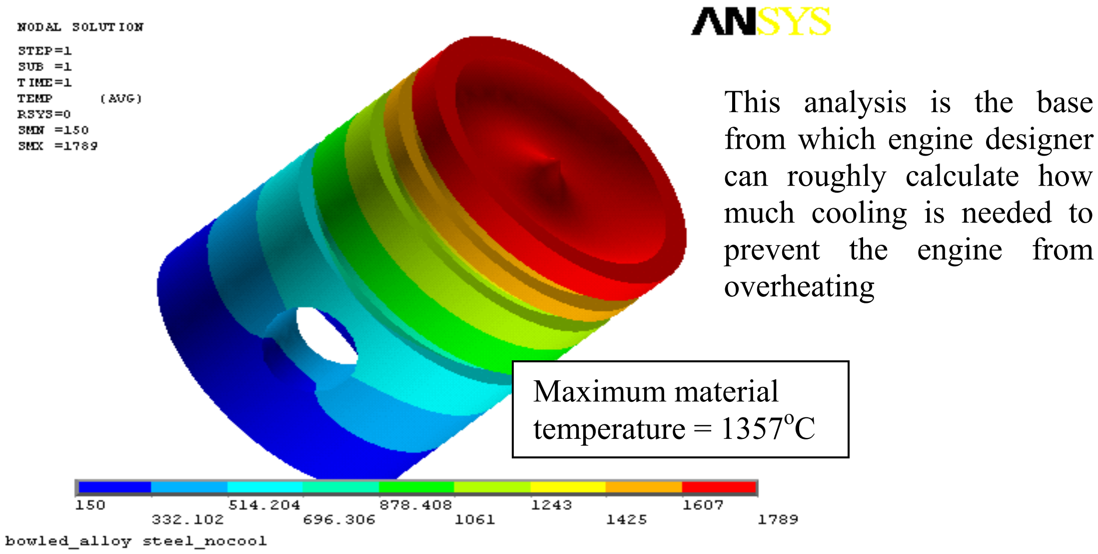


Combustion gas at 2000°C, with a convection heat transfer co-efficient h = 1500 W/m^2 K

Figure 3.18 Thermal boundary condition on the piston **with no cooling** mechanisms [41, 36].

3.11 Results and their physical interpretation (without any cooling)

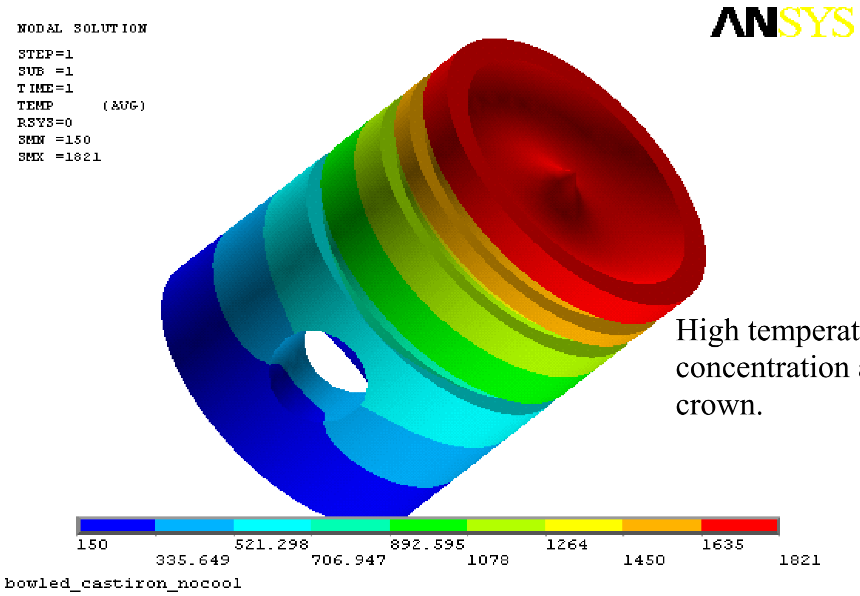
After applying the thermal boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing Fourier heat transfer equations for each and every element and these formulated governing equations were solved for the temperatures from which all the other quantities such temperature gradient, and thermal flux can be calculated. Temperature distribution is shown below.



This analysis is the base from which engine designer can roughly calculate how much cooling is needed to prevent the engine from overheating

Maximum material temperature = 1357°C

Figure 3.19 Temperature distributions of the piston made up of alloy steel **with no cooling** mechanisms in **bowled** piston geometry



High temperature concentration at the piston crown.

Figure 3.20 Temperature distributions of the piston made up of cast iron **with no cooling** mechanisms in a **bowled** geometry

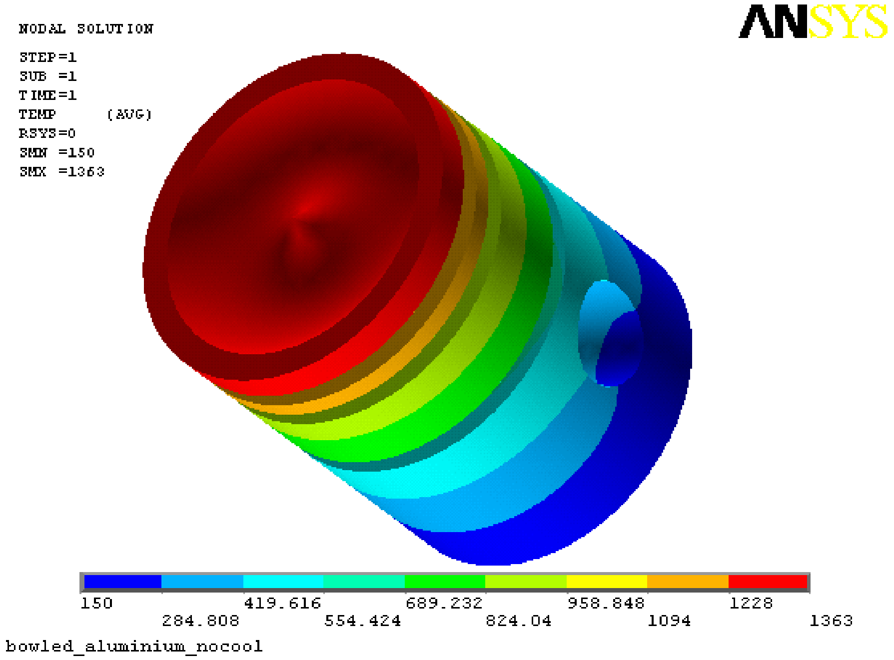


Figure 3.21 Temperature distributions of the piston made up of aluminium steel **with no cooling** mechanisms in **bowled** piston geometry

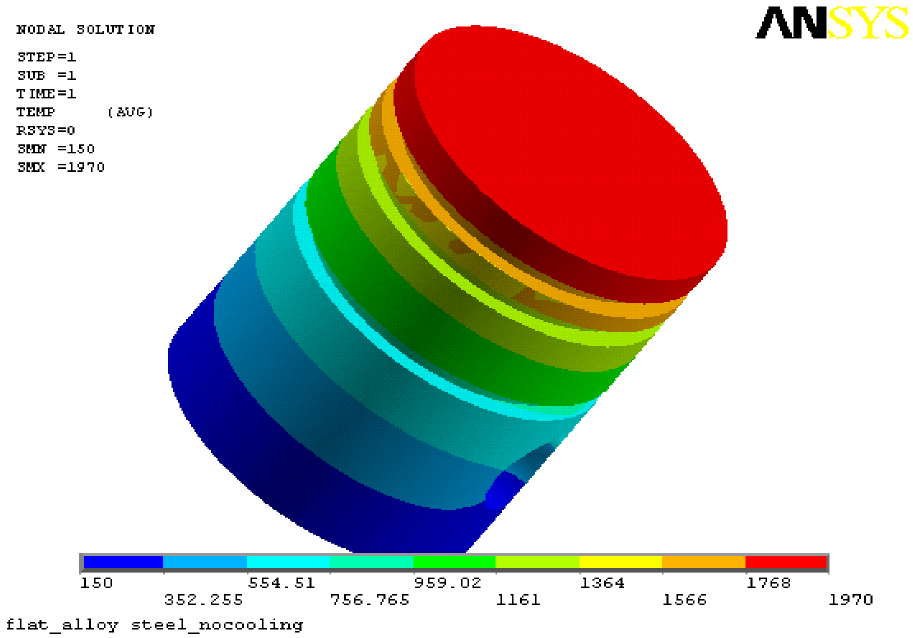


Figure 3.22 Temperature distributions of the piston made up of alloy steel **with no cooling** mechanisms in **flat** piston geometry.

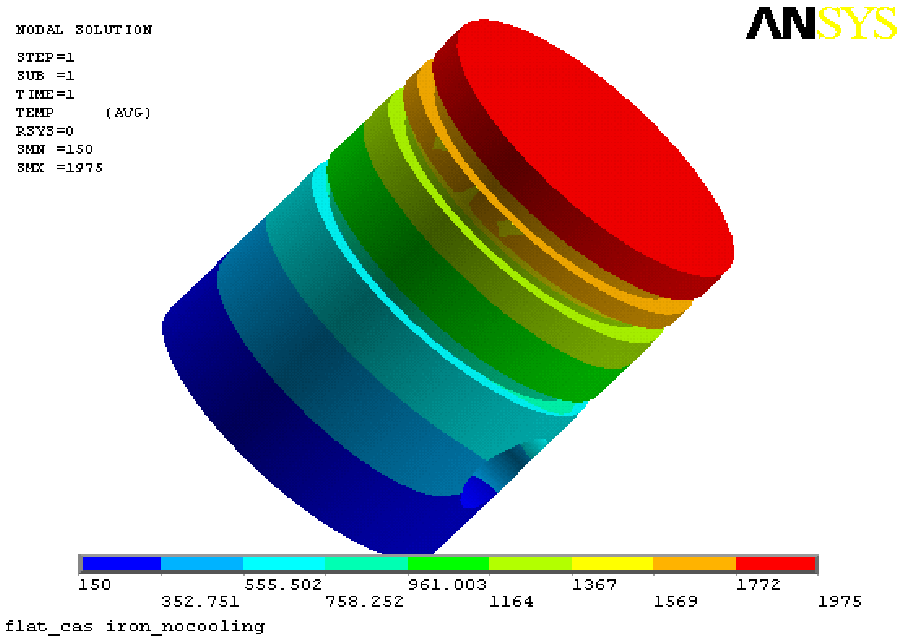


Figure 3.23 Temperature distributions of the piston made up of cast iron with **no cooling mechanisms** in **flat** piston geometry [36, 38, 41].

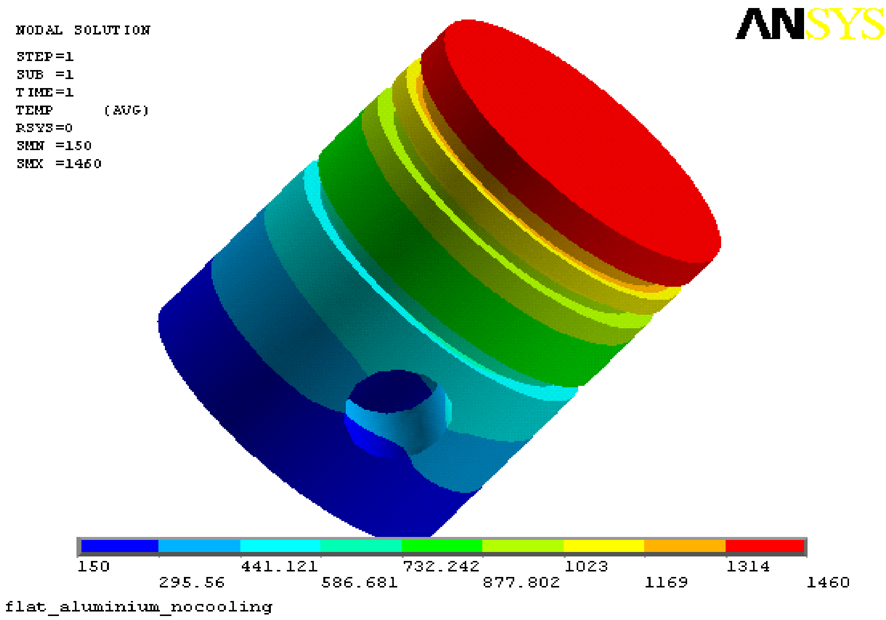


Figure 3.24 Temperature distributions of the piston made up of aluminium with **no cooling** mechanisms in flat piston geometry.

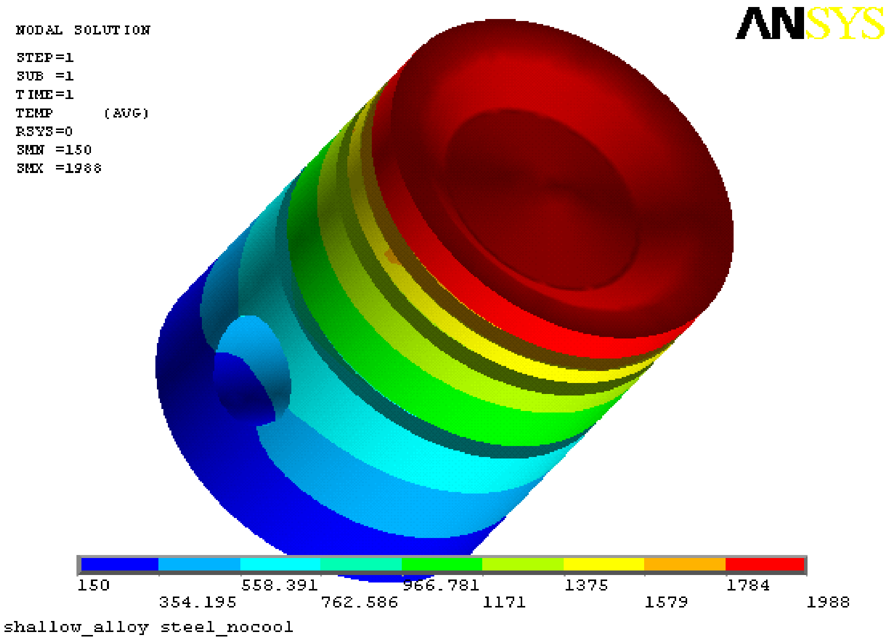


Figure 3.25 Temperature distributions of the piston made up of alloy steel **with no cooling** mechanisms in **shallow** piston geometry.

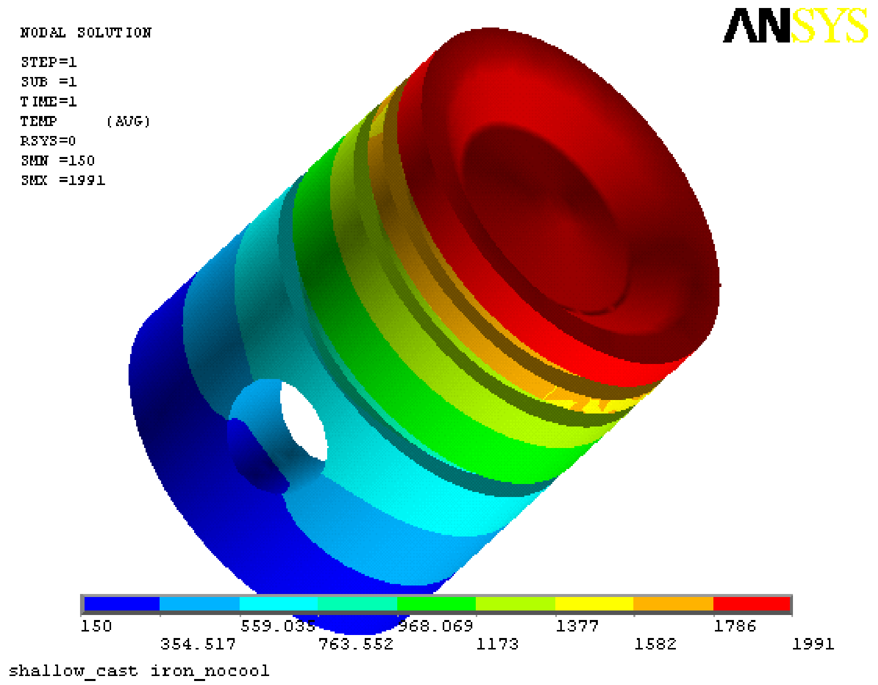


Figure 3.26 Temperature distributions of the piston made up of cast iron **with no cooling** mechanisms in **shallow** piston geometry.

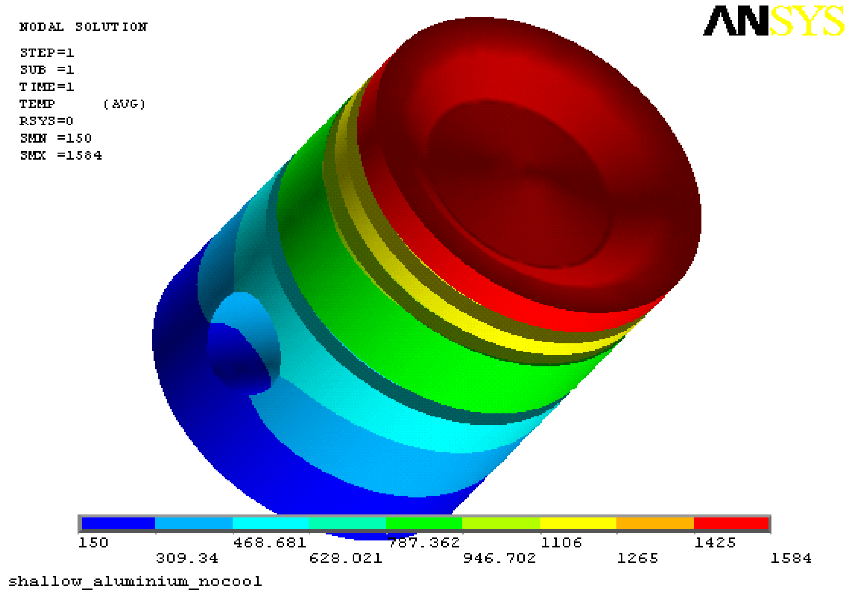


Figure 3.27 Temperature distributions of the piston made up of cast iron **with no cooling** mechanisms in **shallow** piston geometry.

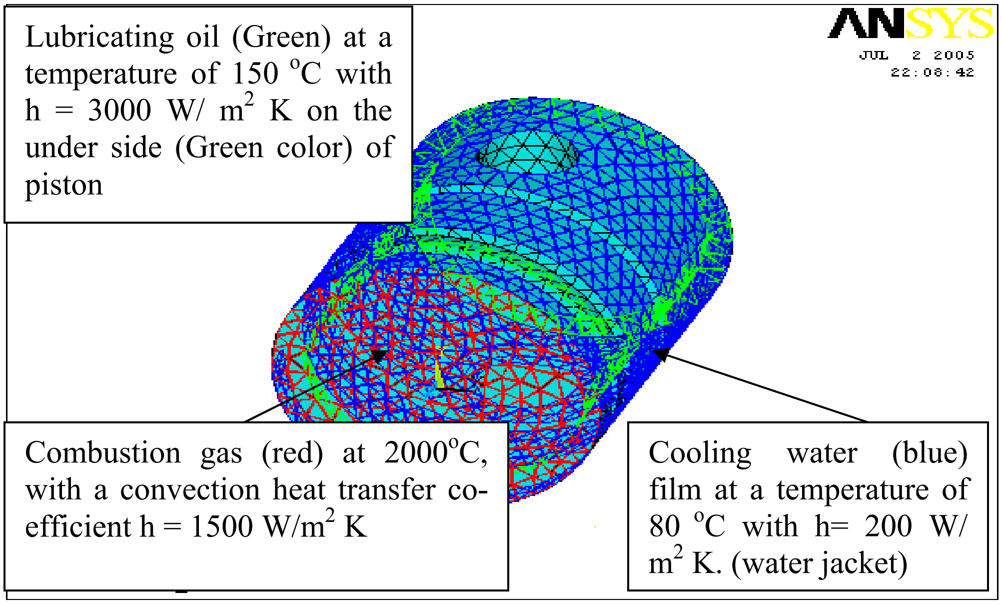
As can be seen from the above plots, the lesser the thermal conductivity more the heat accumulation in the piston material. The maximum temperature reached in different piston geometry and for different material is summarized below.

Shape of the piston crown	**Piston material**	**Thermal conductivity (k) in W/mK**	**Peak material temperature in ^oC**
Flat	Alloy steel	52	**1460**
Flat	Cast iron	46	**1975**
Flat	Aluminium	220	**1970**
Bowled	Alloy steel	52	**1789**
Bowled	Cast iron	46	**1821**
Bowled	Aluminium	220	**1363**
Shallow	Alloy steel	52	**1988**
Shallow	Cast iron	46	**1991**
Shallow	Aluminium	220	**1584**

Table 3.1 Summary of thermal analysis with no cooling mechanisms simulated

3.12 Physical boundary conditions – Thermal loading with cooling

In this third stage of the thermal analysis, cooling mechanisms were simulated in addition to the combustion heat load. The boundary conditions are combustion gas is at a temperature of 2000 ^oC with a convection heat transfer coefficient of 1500W/m^{2}K and cooling loads are water film with a bulk temperature of 80^o C with a convection heat transfer coefficient h = 200 W/m^2 K [36, 7, 27,34]. The other cooling mechanism is lubricating oil at a temperature of 150 ^{o} C with a convection heat transfer co-efficient h = 3000 W/m^{2}K.



Lubricating oil (Green) at a temperature of 150 °C with h = 3000 W/ m^2 K on the under side (Green color) of piston

Combustion gas (red) at 2000°C, with a convection heat transfer co-efficient h = 1500 W/m^2 K

Cooling water (blue) film at a temperature of 80 °C with h= 200 W/ m^2 K. (water jacket)

Figure 3.28 Thermal boundary condition of the piston **with cooling** mechanism

3.13 Results and their physical interpretation (with cooling)

After applying the thermal boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing Fourier heat transfer equations for each and every element and these formulated governing equations were solved for the temperatures from which all the other quantities such temperature gradient, and thermal flux can be calculated. Temperature distribution is shown below.

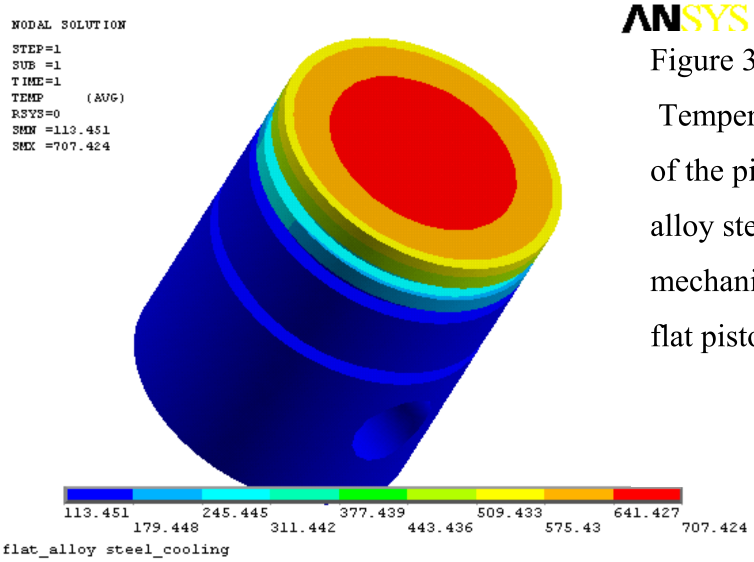


Figure 3.29

Temperature distributions of the piston made up of alloy steel with **cooling** mechanisms simulated in flat piston geometry.

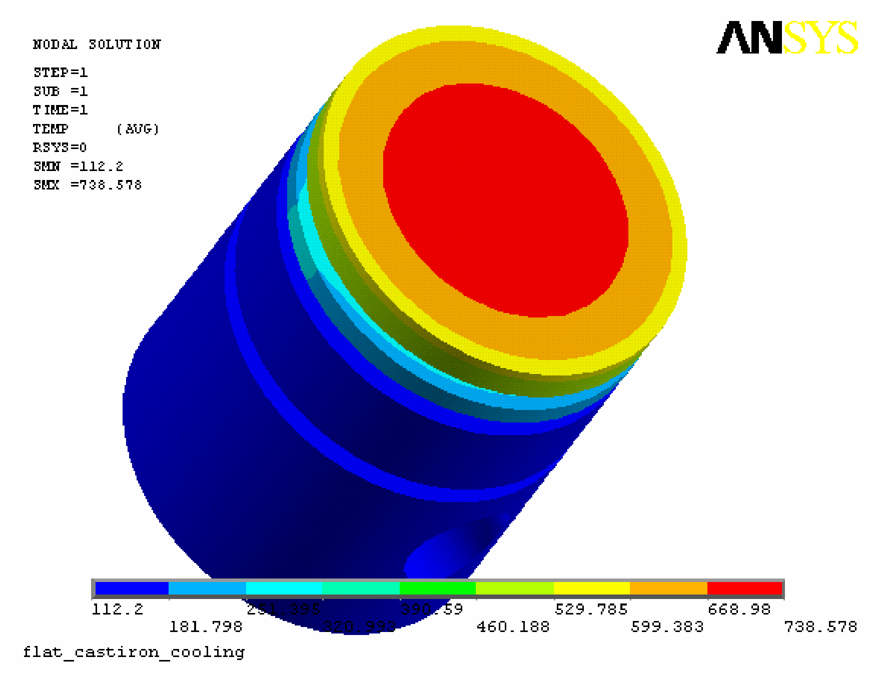


Figure 3.30 Temperature distributions of the piston made up of cast iron with **cooling** mechanisms simulated in flat piston geometry.

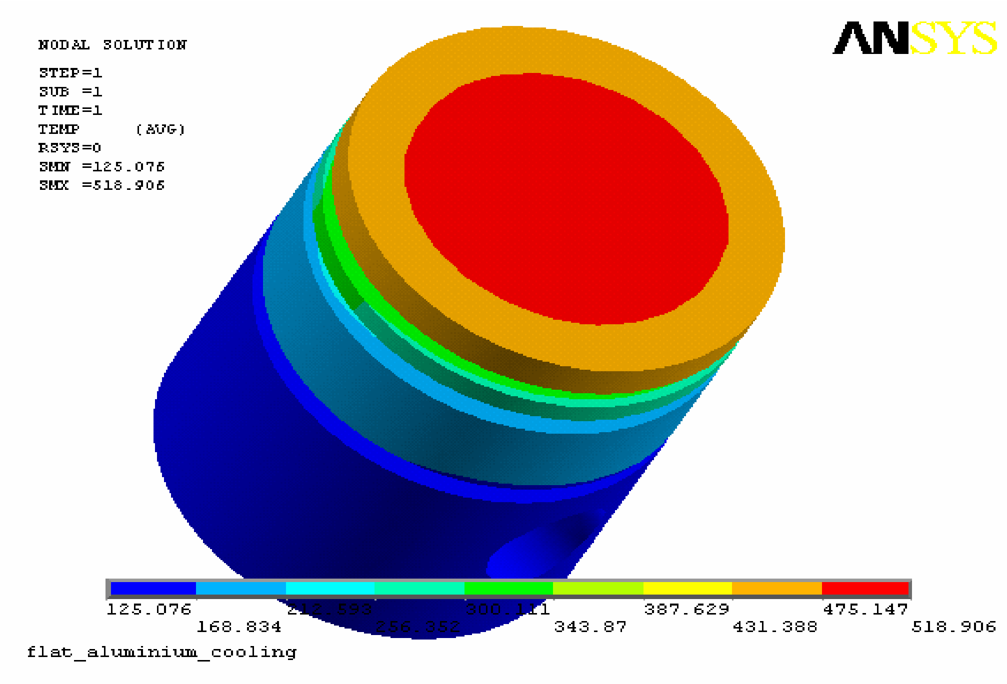


Figure 3.31 Temperature distributions of the piston made up of aluminium with **cooling** mechanisms simulated in flat piston geometry.

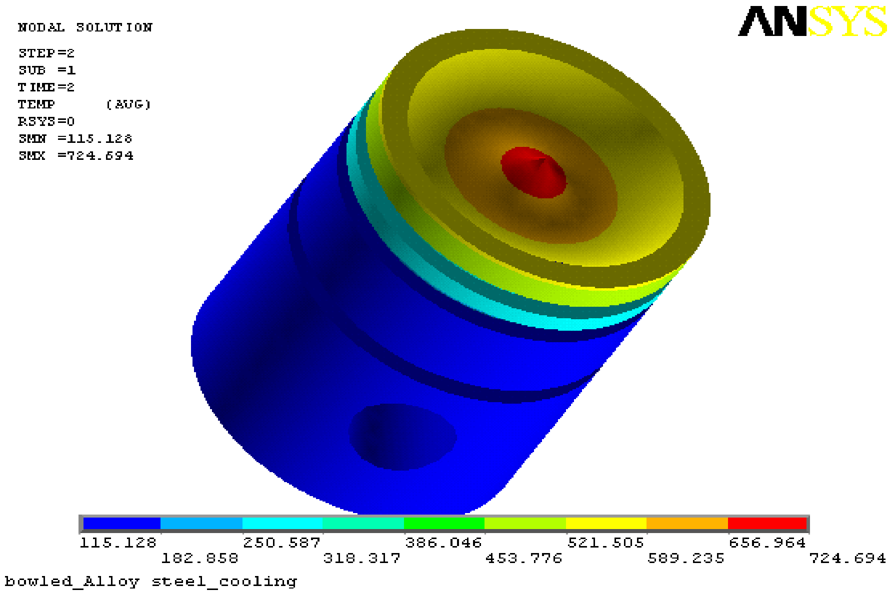


Figure 3.32 Temperature distributions of the piston made up of alloy steel with **cooling** mechanisms simulated in bowled piston geometry.

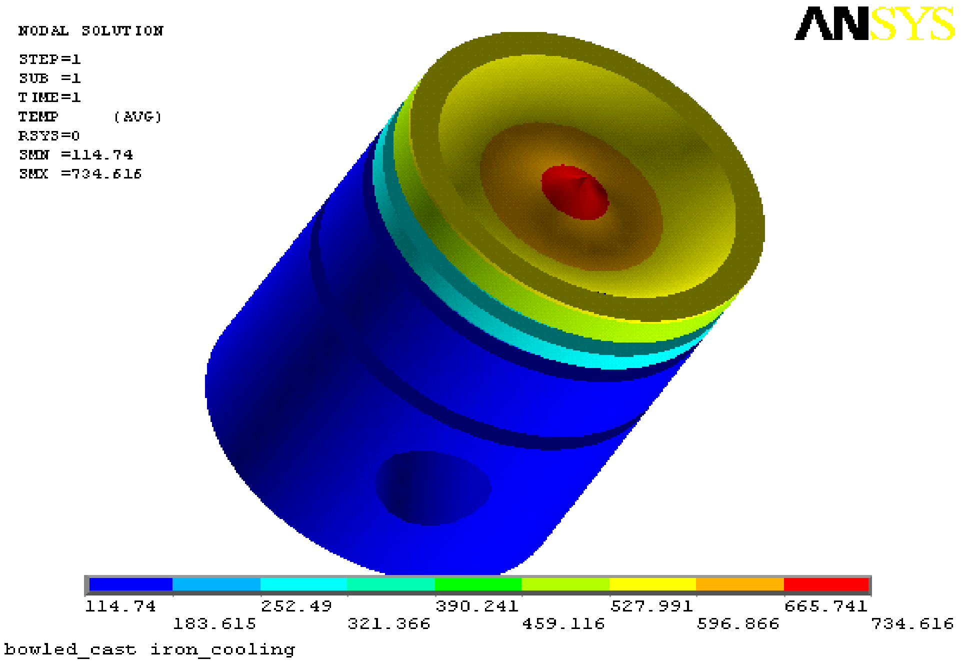


Figure 3.33 Temperature distributions of the piston made up of cast iron with **cooling** mechanisms simulated in bowled piston geometry.

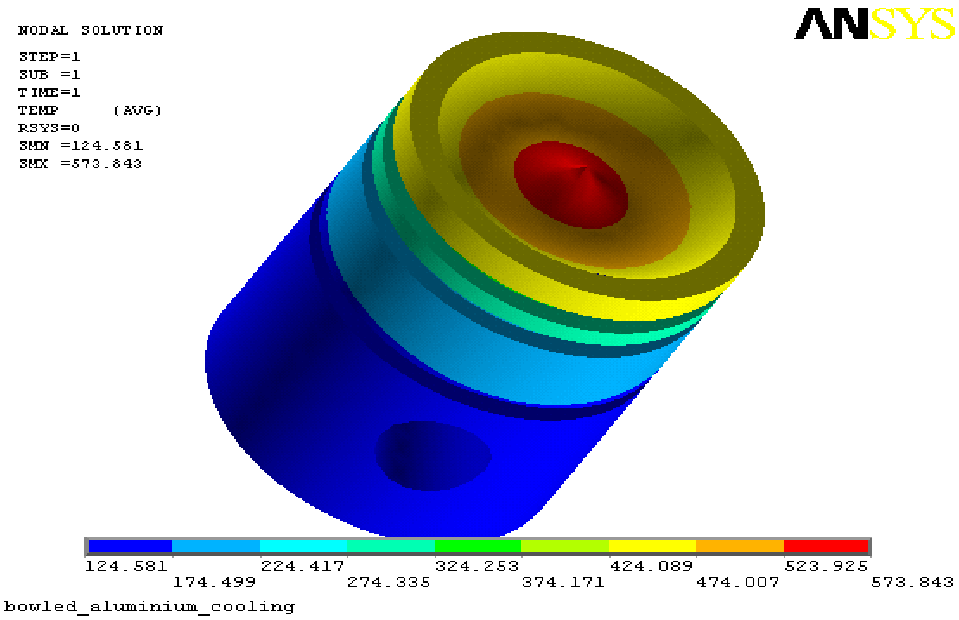


Figure 3.34 Temperature distributions of the piston made up of aluminium with **cooling** mechanisms simulated in bowled piston geometry.

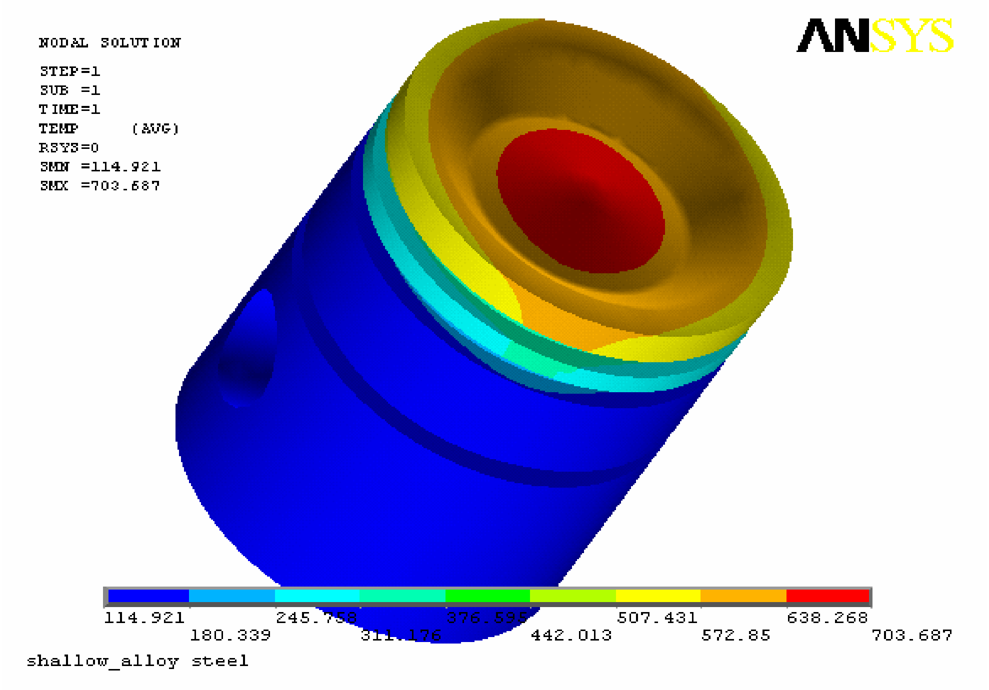


Figure 3.35 Temperature distributions of the piston made up of alloy steel with **cooling** mechanisms simulated in shallow piston geometry.

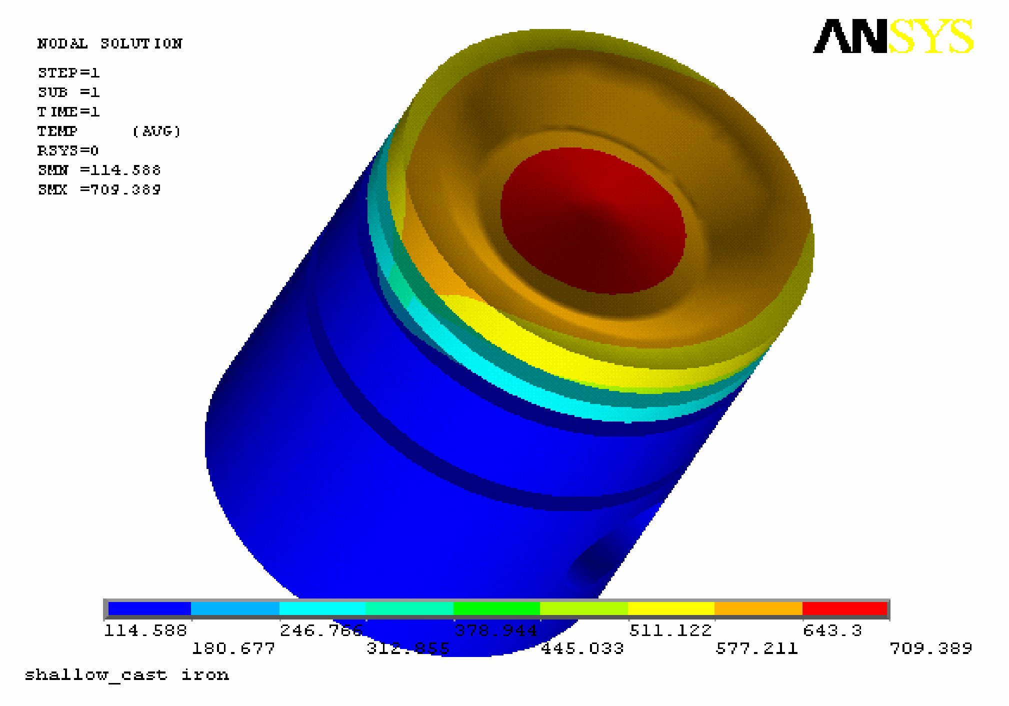


Figure 3.36 Temperature distributions of the piston made up of cast iron with **cooling** mechanisms simulated in shallow piston geometry.

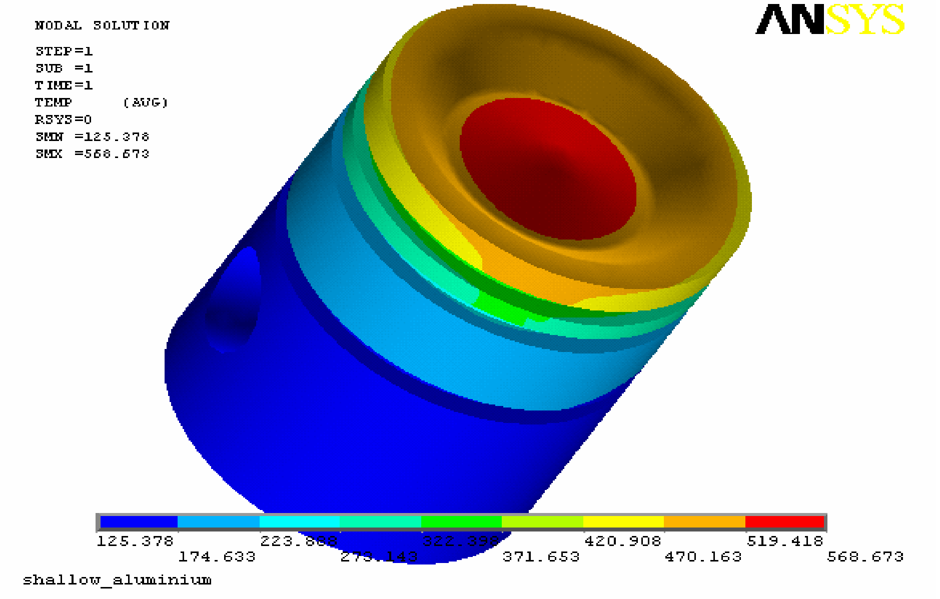


Figure 3.37 Temperature distributions of the piston made up of aluminium with **cooling** mechanisms simulated in shallow piston geometry.

As can be seen from the above temperature plots, By having cooling mechanisms simulated, temperatures were brought down to very low value in most of the regions .Now that most of the regions in the piston are in controlled temperatures. Therefore it is necessary to have dedicated cooling mechanisms for piston without which material melting may not be a surprise. It is also interesting to note that more heat is always concentrated at the centre of the piston. This is so in all piston geometries. This is because of the fact that heat accumulated at the centre need to travel a long distance to dissipate the heat. Here is the summary of material temperatures for the different analysis that were carried out.

Shape of the piston crown	**Piston material**	**Thermal conductivity (k) in W/mK**	**Peak material temperature without cooling in ^{0}C**	**Peak material temperature with cooling in ^{0}C**
Flat	Alloy steel	52	**1460**	**707**
Flat	Cast iron	46	**1975**	**718**
Flat	Aluminium	220	**1970**	**518**
Bowled	Alloy steel	52	**1789**	**724**
Bowled	Cast iron	46	**1821**	**734**
Bowled	Aluminium	220	**1363**	**573**
Shallow	Alloy steel	52	**1988**	**703**
Shallow	Cast iron	46	**1991**	**709**
Shallow	Aluminium	220	**1584**	**568**

Table 3.2 Comparison of both (with and without) the thermal analysis

As can be seen from the above plots, it is very clear that cooling mechanisms are essential for an engine. Most of the regions in the piston were brought to a controlled temperature when the cooling mechanisms were simulated. Aluminium piston runs very cooler than the alloy steel and cast iron made pistons. As it has been already mentioned, this simulation is basically **peak moment simulation** in which critical moments (at the point of combustion) were simulated. So very practically speaking, temperature in the engine will have to rise to these levels even with water and oil cooling [1, 7, 34, 36]. An engine running at very stable condition, the surface temperature of the piston is around 500-700 °C. This range is in fact varies during burning cycle. This range is maximum during the combustion moments.

3.14 Summary

- Piston is a reciprocating element in an internal combustion engine, which is subjected to both thermal and structural loads. So both thermal and structural analyses were carried out.
- Analysis carried out was basically a **peak moment simulation,** in which conditions prevailing at the point of combustion were simulated.
- Structural load is gas pressure and thermal load is heat energy released by the burning fuel and convection water and oil cooling
- Combustion pressure induces the structural stresses in the piston
- The very main thermal function of the piston is to conduct away the necessary amount of heat to the cooling mediums such as water and oil in order to survive itself other it would have to melt.
- Piston should have to retain the sufficient of heat in order to sustain the combustion and higher thermal efficiency.
- The peak surface temperature of the piston material when there is no cooling is about 1980°C against 518 °C when cooling was provided.

CHAPTER – 4

Finite Element analysis of Engine block

(Structural and thermal aspects)

Engine block is heart of an internal combustion engine as it forms base of an internal combustion engine. The following are the important structural and thermal functions of the engine block [3, 4, 6, 7, 1]. They are

- It forms the combustion chamber in which burning is taking place.
- It forms the bearing surface on which piston slides
- It takes the structural stresses induced due to the gas pressure
- It takes the thermal stresses induced due to the heat load
- Engine block combustion chamber surfaces should be kept at a sufficient temperature in order to sustain the combustion and also to have good thermal efficiency.
- Engine block conducts away the necessary amount of heat otherwise it may melt which is not uncommon.

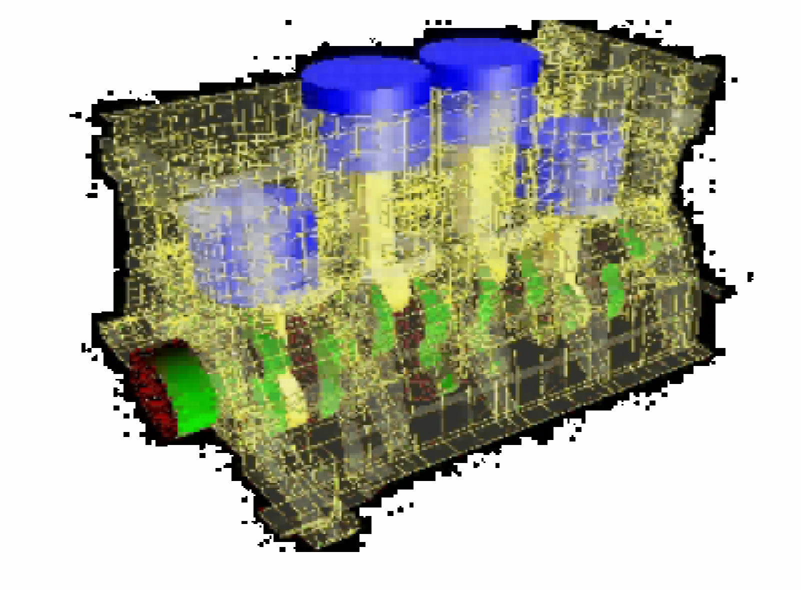


Figure 4.1 Typical diesel Engine block [Source: www.ADAMSengine.com]

Engine blocks are normally made up of cast iron or aluminum. Cast iron is used as an engine block material because it has good wear resistance, good damping capacity and also the fact that it is good in compression which are the important quality that an engine block material should possess. Aluminum (alloy 1100) is used as an engine block material because it is a good heat transporter (high thermal conductivity k =222 W/m K) and one of the less dense material [5]. There is a point of difficulty while designing an engine block is that more heat conduction through the cylinder walls will facilitate the functioning of the engine block but this would affect the thermal efficiency of the engine and there may be a problem that combustion may not sustainable. Therefore there has to be a trade off between amount of heat to be retained and amount to be conducted away.

4.1 What is the need to conduct FEA analysis of engine block?

These are the possible reasons that necessitate conducting FEA analysis on engine block [1, 6, 7, 29, 31]

5. To check whether or not engine block material takes the structural stress induced due to gas load
6. To check whether or not engine block material takes the heat load (fuel energy released at the point of combustion)
7. Is surface temperature of the cylinder inner surface less than melting point of the cylinder material?
8. Is thermal stress severe?

The above stated questions can be answered only when FEA analysis is carried out on the engine block.

4.2 Physical problem- structural load

Diesel engine is a smoky and high mass burning engine. The peak combustion pressure at the point of combustion of a typical diesel engine is in the range of 70 – 120 bars depending upon the field of work [3, 17]. This peak combustion pressure induces the compression stress in the cylinder wall material. So, one of the main function of the cylinder block is to take this compression stresses.

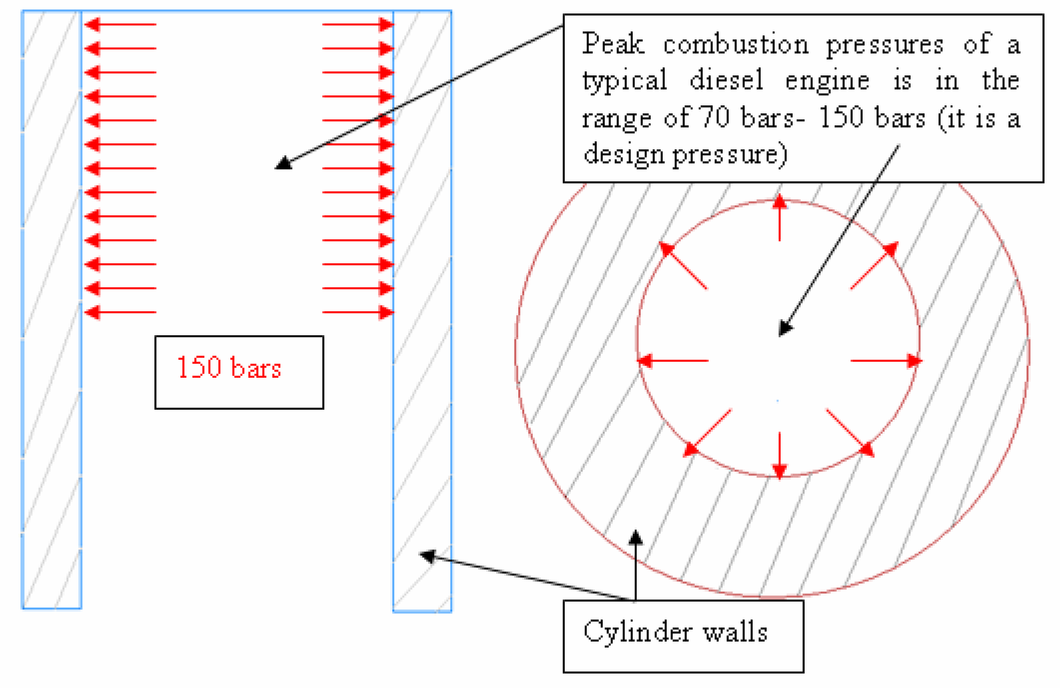


Figure 4.2 Nature of the physical problem

The main objective of conducting FEA analysis on the engine block is to assertion whether or not engine block material takes the structural stress induced due to gas pressure. When the stresses induced due to peak combustion pressures greater than the yield point of the material, it is very obvious that cylinder block will fail structurally. So, the ultimate objective is to find out what is the maximum stress induced in the cylinder block and then to compare with the materials yield strength or design strength.

4.3 Physical problem- thermal load

Combustion is taking place in a space formed by cylinder surfaces, piston top surface (crown), and by the cylinder head [50, 29, 27, 1, 6]. Cylinder wall is the ultimate heat path for transferring heat from the cylinder inner surface to the cooling medium. At the point of combustion, moving gas comes into contact with a wall; there exists a relatively stagnant gas layer which acts as a thermal insulator. The resistance of this gas layer to the heat flow is quite high. Heat transfer from the cylinder gases takes place through cylinder walls to the cooling medium. A large temperature drop is produced in the stagnant gas layer adjacent to the walls.

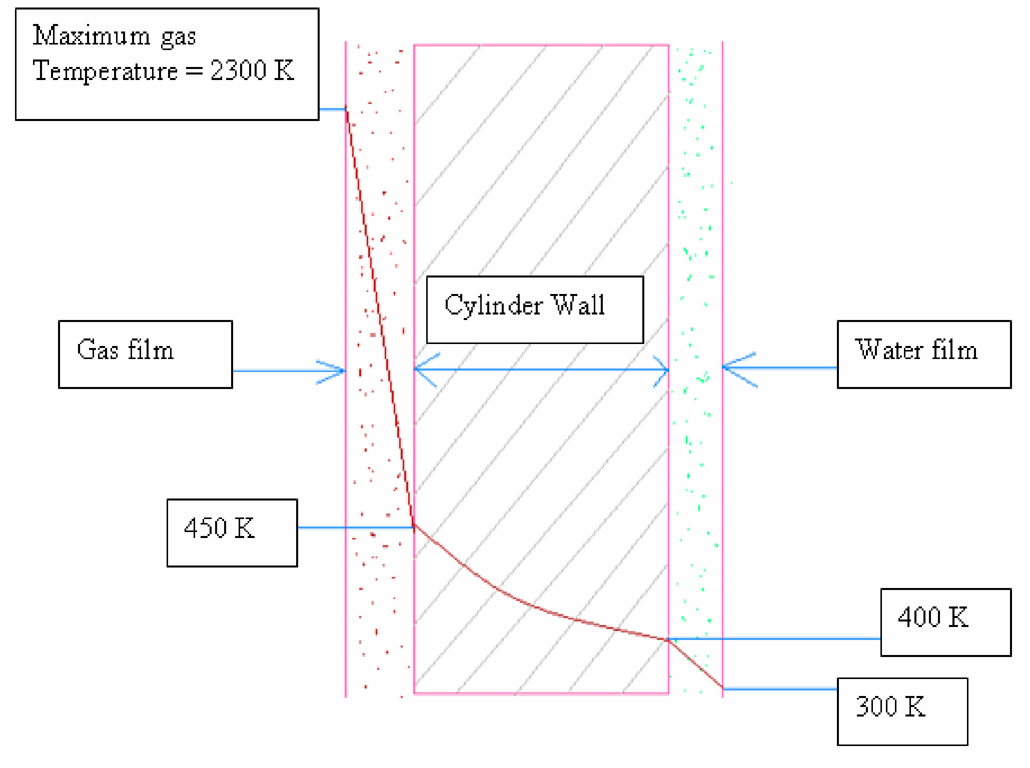


Fig 4.3 Cylinder wall temperature distribution in a properly water cooled cylinder

In a high burning engine such as diesel engine, the peak cylinder gas temperature may be 2300 K while the temperature of the inner wall surface may be only 450 K due to cooling as shown in the figure 4.3. Heat is transferred from the gases to the cylinder walls when the gas temperature is higher than the cylinder wall temperature. The rate and direction of flow of heat varies depending upon the temperature differential. If no cooling is provided, there could be no heat flow, so that whole cylinder wall would soon reach an average temperature of the cylinder gases and if this temperature is more than the melting of the temperature cylinder material, it will have to melt. So, the ultimate objective of conducting FEA analysis on engine block is to find out what are the peak material temperature and its distribution due to heat load and also to plot thermal gradient and heat flux. There are three major kinds of analysis normally performed on engine block.
They are [1, 3, 4, 6, 54]

- Static structural stress analysis
- Dynamic stress analysis

- steady state heat transfer analysis (To obtain temperature distributions, heat flow rates, heat flux and temperature gradient)
- Transient heat transfer analysis (To obtain temperature distributions with respect to time , time heat flow rates, heat flux and temperature gradient)
- Thermal stress analysis (due to temperature gradient and constraining)

In the current study, only static structural stress and steady state heat transfer analysis were performed.

4.4 Approach of the analysis

Although engine block needs **coupled structural- thermal analysis** in order to predict the thermal stresses in the engine block, in the current study both the analysis were treated separately because of the fact that engine block is not rigidly fixed and also contribution of thermal stress to net stress is negligible compared to the compressive stress [45, 41, 50, 34]. Therefore following approach was used to carry out the analysis.

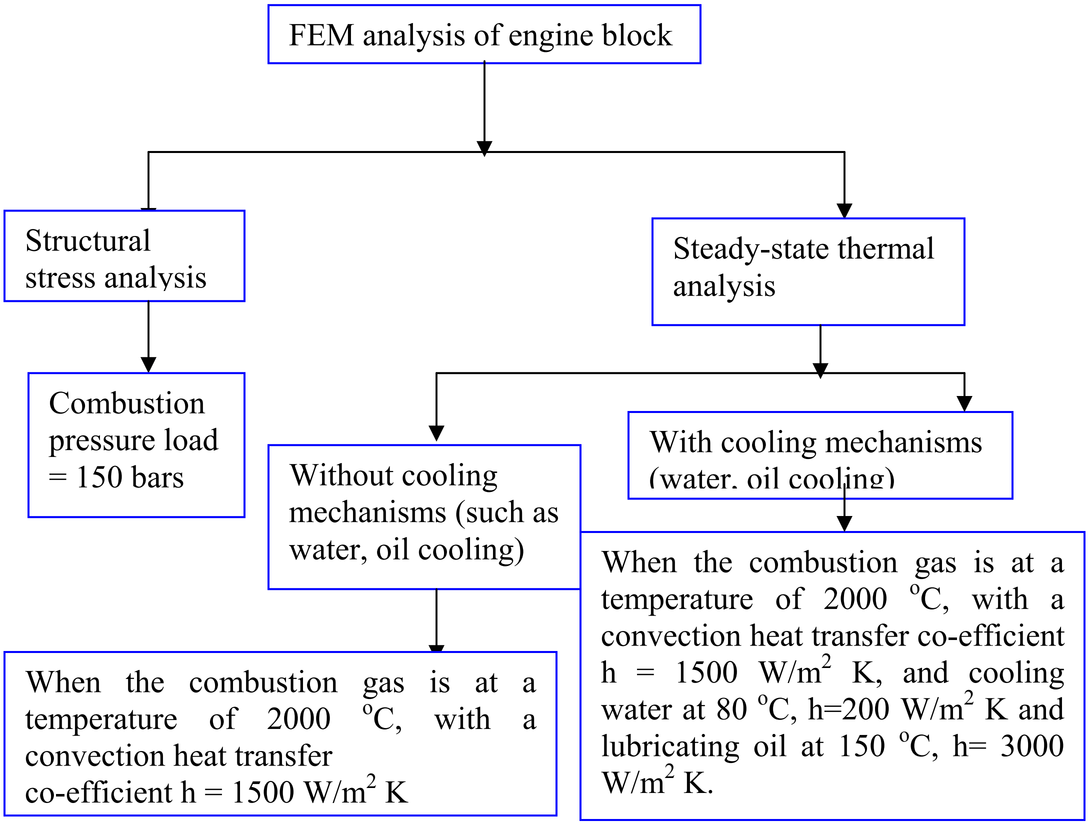


Figure 4.4 Approach of the analysis

4.5 Inputs required to carrying out the above analysis

- Approximate geometry of the engine block
- Materials properties of the engine block material such as [5]
    - Young’s modulus of cast-iron (E = 168 GPa)
    - Poisson’s ratio (μ=0.3)
    - Strength of the material of cast iron (S = 379 MPa)
    - Thermal conductivity of the material of cast iron (k = 46 W/mK)

4.6 Typical FEM procedure to carry out this analysis

The following steps are the typical procedure to carry out any finite element analysis [12, 75, 95]

- **Geometry** creation (solid modeling of engine block)
- Dividing the solid model into number of elements and then connecting these elements with each other (**meshing**) and thereby obtaining the finite element model.
- Applying boundary conditions

    1. for **structural** analysis-boundary conditions means defining combustion pressures and fixing constraints
    2. for **thermal** analysis – boundary conditions means defining gas convection load , convection water cooling load, atmospheric conditions, and convection oil cooling load.
- Solving the problem
- Results and their interpretation.

4.7 Geometry creation and meshing

Geometry of the engine block is needed to carry out this analysis. Solid model of the engine block was created in solid modeling environment called “**CATIA V5**”.Solid model is the base for carrying out any FEA or CFD analysis. Solid model of the engine block that was created in **CATIA V5** solid modeling environment is shown in the figure 3.4. The created solid model was brought in to the ANSYS analysis environment using **ANSYS-CATIA** interface. Solid model used in any FEA analysis can be very simple and at the same time it reflects the physical reality of the engine block. Therefore, small small features like rounds, chamfers, holes have not been modeled. As shown in the figure below, solid model is very simple but at the same time gives a physical reality. Solid model of the typical engine block is very big, and it has about hundreds of small features. Engine block has cooling water jackets, oil holes, holes for bolts and nuts. Solid model will have to be turned into to FEM model by dividing the solid model into number of small small elements. This process is called “**discritization**”.Discritization is the process of dividing the solid model into finite number of elements. After dividing the solid model into finite number of elements, they have to be connected to each other as the solid volume is continuous and physically connected at each and every material point.

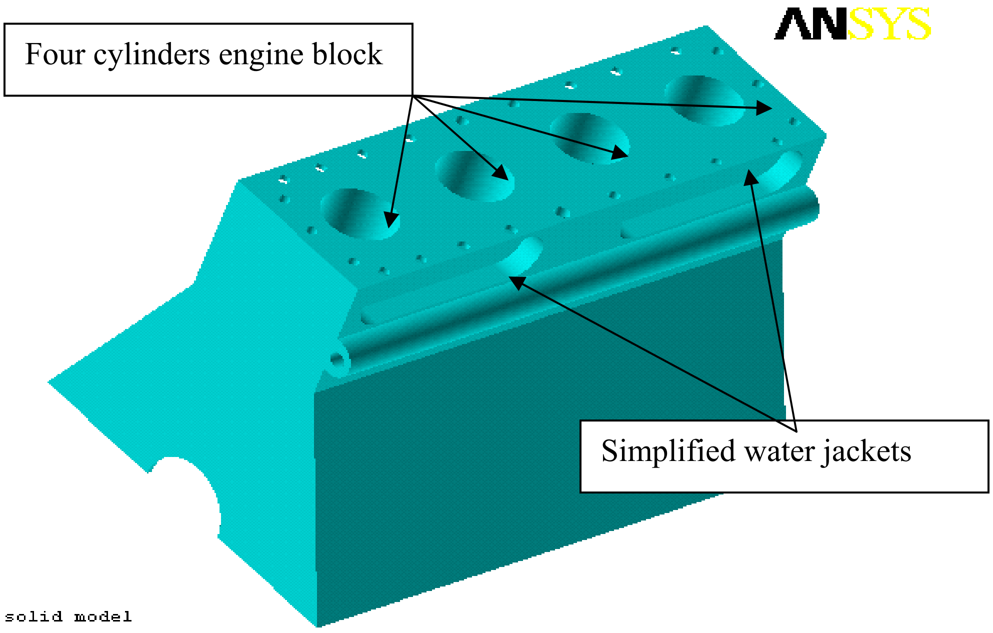


Figure 4.5 Solid model of the engine block created in CATIA V5 and then imported into ANSYS analyzing environment using ANSYS-CATIA interface.

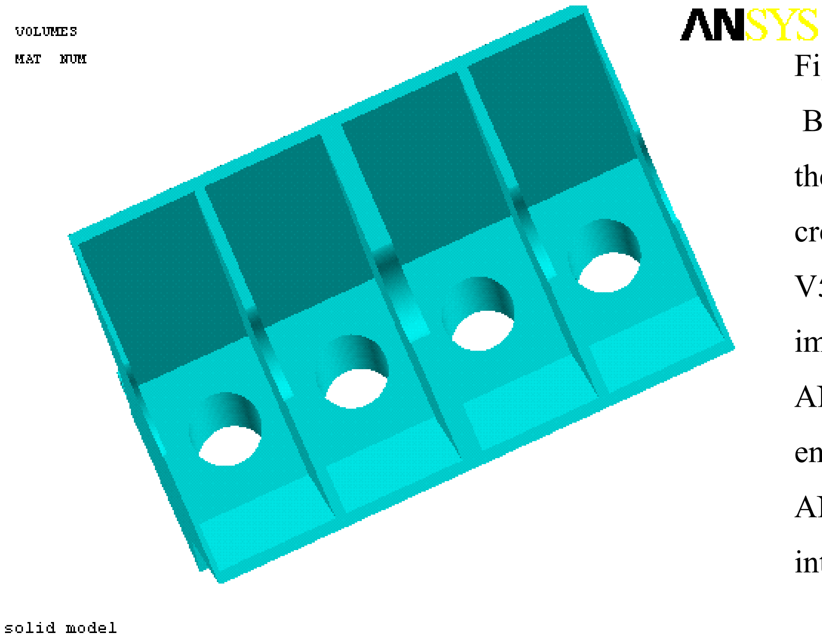
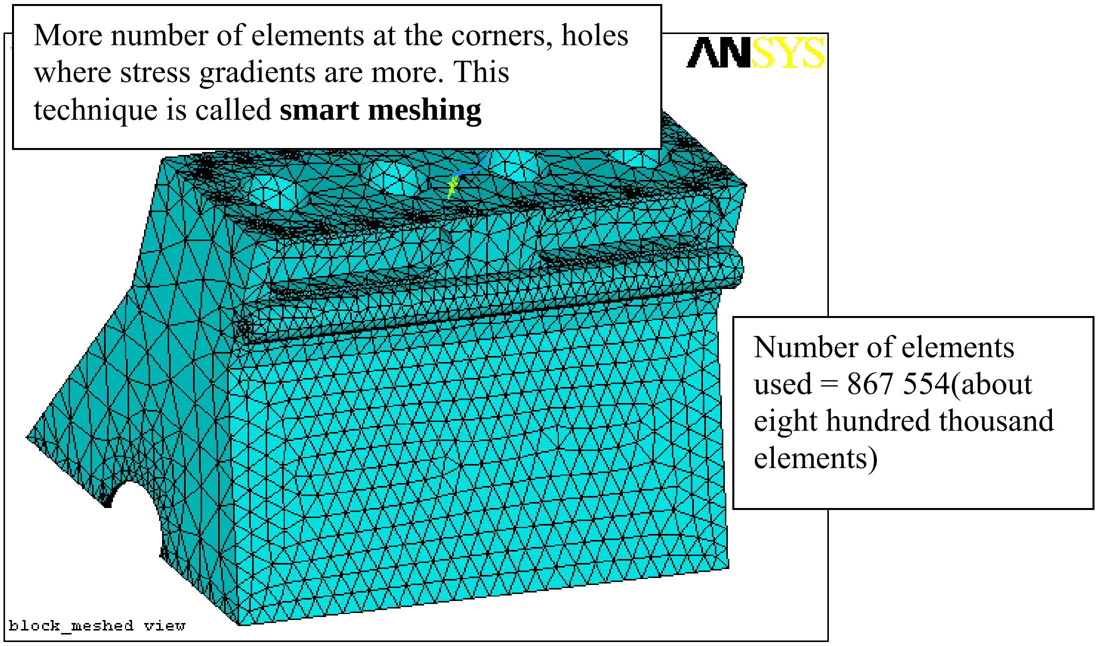


Figure 4.6 Bottom view of the engine block created in CATIA V5 and then imported into ANSYS analyzing environment using ANSYS-CATIA interface.

In simple terms meshing means that connecting the elements with each other. An element is the building block of the finite element model. So the type of element (linear element or higher order elements), number of nodes and their capabilities are important parameters for selecting an element for a particular analysis. Meshed model of the engine block is shown below.



More number of elements at the corners, holes where stress gradients are more. This technique is called **smart meshing**

Number of elements used = 867 554(about eight hundred thousand elements)

Figure 4.7 Meshed model of the engine block

In this analysis, the element used was **Tet 10 node3 SOLID 187** which is a solid element which has all six degrees of freedom (translation in all X, Y, and Z and rotations about all three axes); it also has stress stiffening and buckling capabilities. This element is shown below.

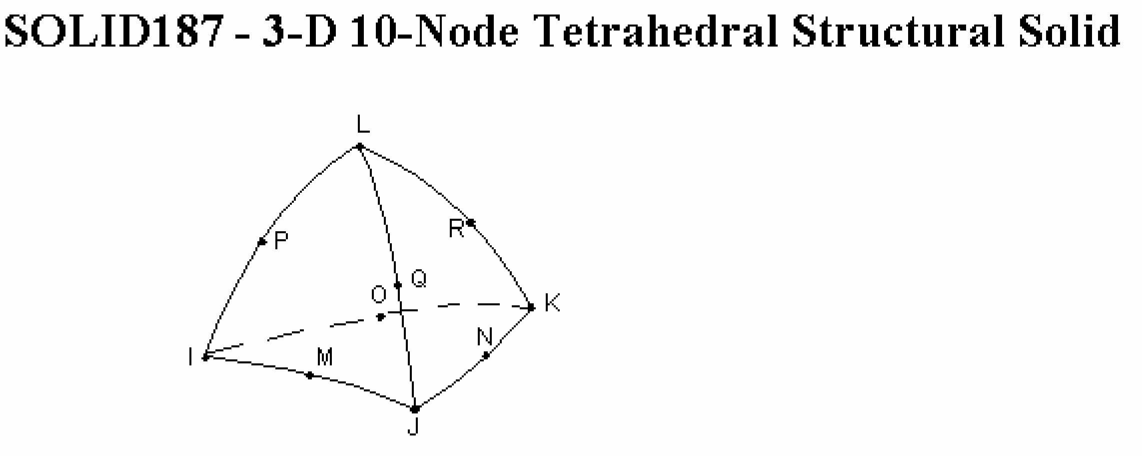
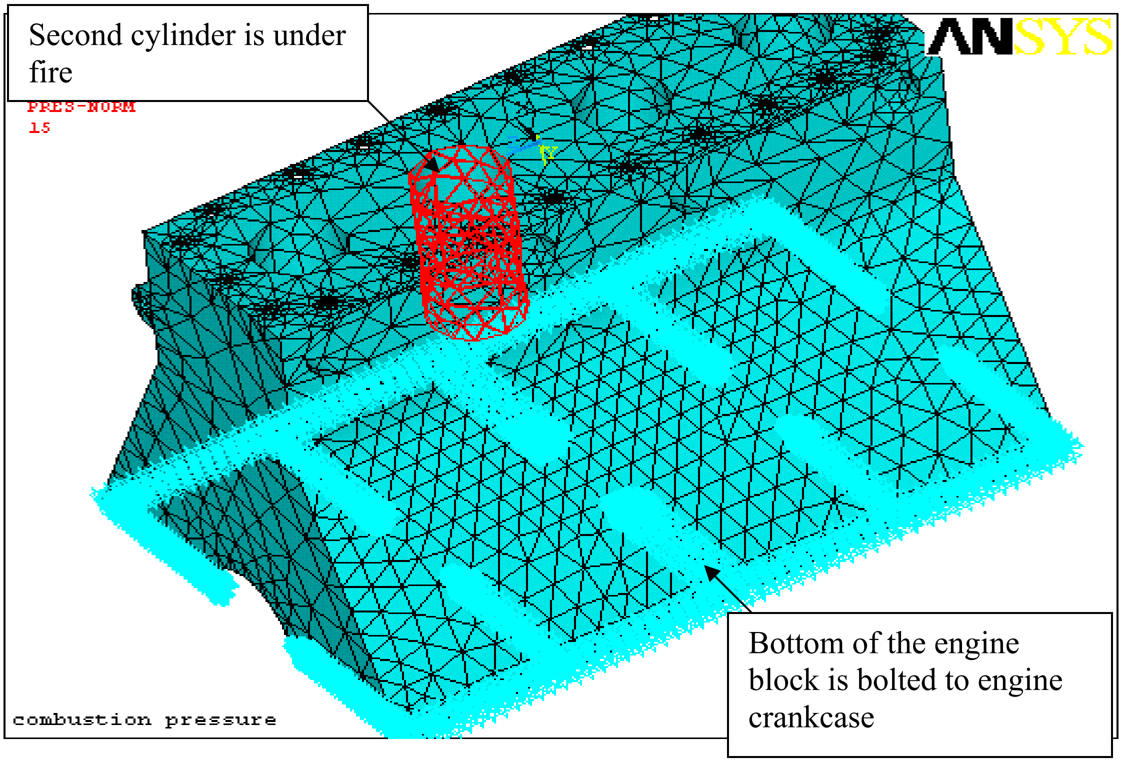


Figure 4.8 Element name – Tet 10 node SOLID 187

Meshing is an important step in FEA analysis. Mesh must be adequate enough to obtain the accurate results while keeping the computational time in mind. More the number of elements, better the accuracy, on the other hand more the number of elements more the computational time. The number of elements used in this analysis was 867 554 (about eight hundred thousands) .In order to obtain accurate information about the stress gradients, temperature gradients and temperature distributions, more number of elements have been placed at the corners, rounds, and at the holes by using ANSYS **smart meshing** capability. Smart meshing is the special feature in ANSYS meshing capability, which places more number of elements at points where stress gradient, temperature gradient are more. The ultimate purpose is to obtain more accurate information at the very sensitive places.

4.8 Physical boundary conditions – Structural loading

Structural loading simply means that applying gas pressure loads. The combustion gas pressure was applied on the inner surface of the cylinder block and bottom of the block was bolted to engine crankcase and then to the supports. The gas pressure taken for the study was 150 bars (may be for a new design).The analysis was started with the idea of applying the combustion gas pressure only in a particular cylinder. But the at the same time engine designer should not forget the fact that typical diesel engine running at **3000 rpm**,should have to sustain **200 firings in a single second.** Therefore time delay between the successive firings was neglected, and the gas pressure was applied on all the cylinders.

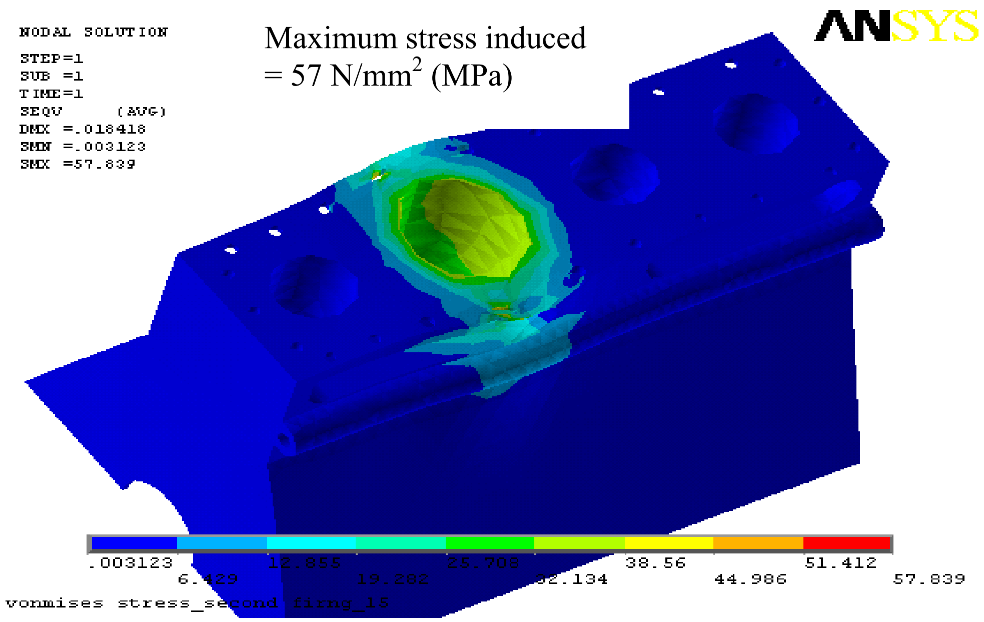

Second cylinder is under fire

Bottom of the engine block is bolted to engine crankcase

Figure 4.9 Structural boundary condition of the engine block when second cylinder is firing.

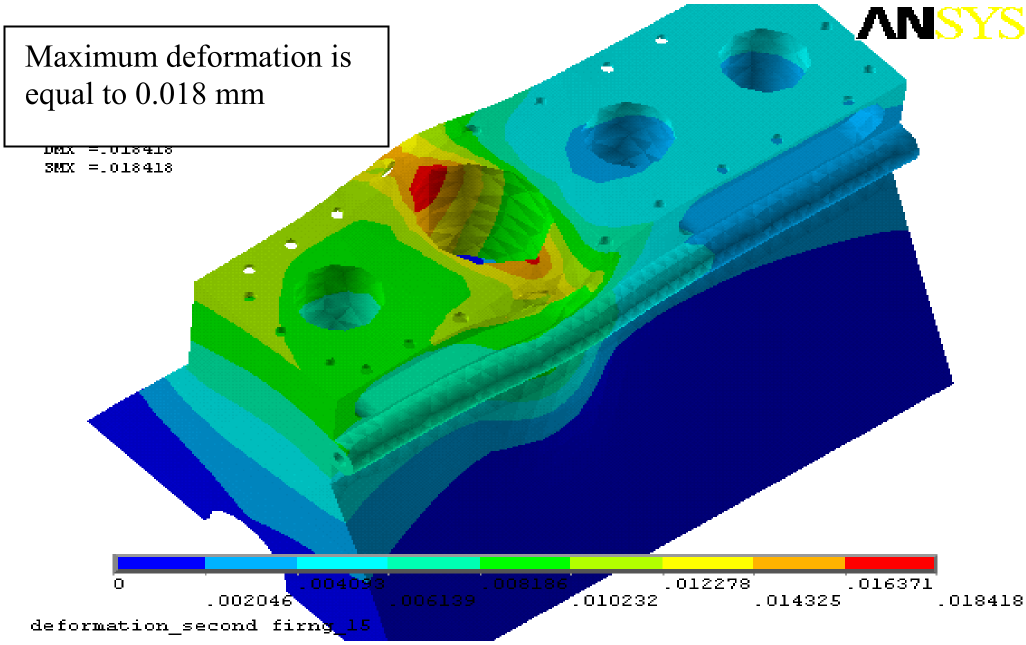
4.9 Results and their physical interpretation

After applying the boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing structural stress strain equations for each and every element and these formulated governing equations were solved for the deformations from which all the other quantities such stresses, strains etc can be calculated.



Maximum stress induced = 57 N/mm^2 (MPa)

Figure 4.10 **Von-mises stress** (equivalent stress) contour of the engine block when **second** cylinder is firing.



Maximum deformation is equal to 0.018 mm

Figure 4.11 Deformation contour of the engine block when **second** cylinder is firing.

As can be seen from the above plots, the maximum stress induced in the block is about **57 N/mm^2 which about 20 % of the strength of the cast iron** material from which engine block is made. In the next step, the load was applied on all the cylinders. The applied boundary condition is shown below.

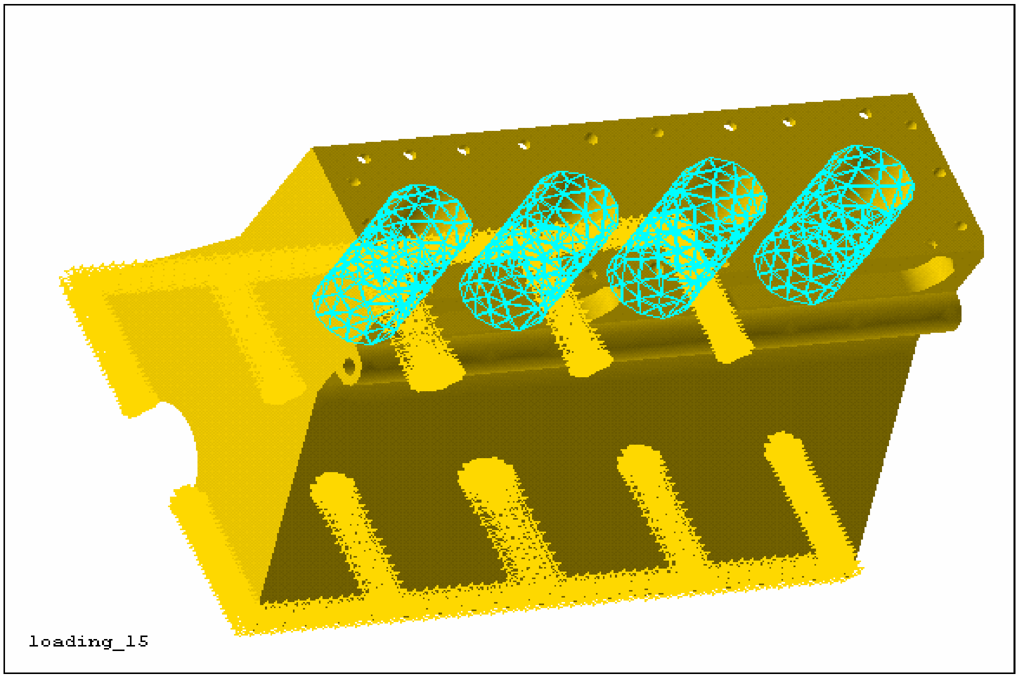
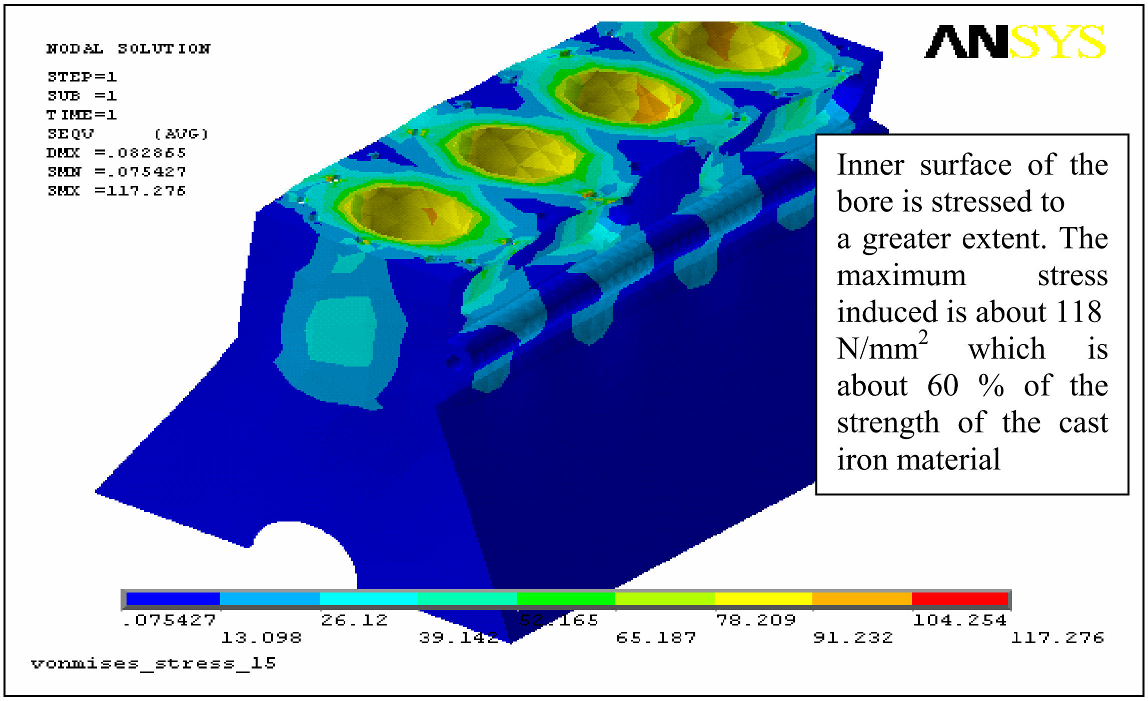


Figure 4.12 Structural boundary condition of the engine block when all the four cylinders are firing (simultaneous firing) with a peak pressure of 150 bars.



Inner surface of the bore is stressed to a greater extent. The maximum stress induced is about 118 N/mm^2 which is about 60 % of the strength of the cast iron material

Figure 4.13 **Von-mises stress** (equivalent stress) contour of the engine block when **all four cylinders are firing** with a peak pressure of **150 bars**.

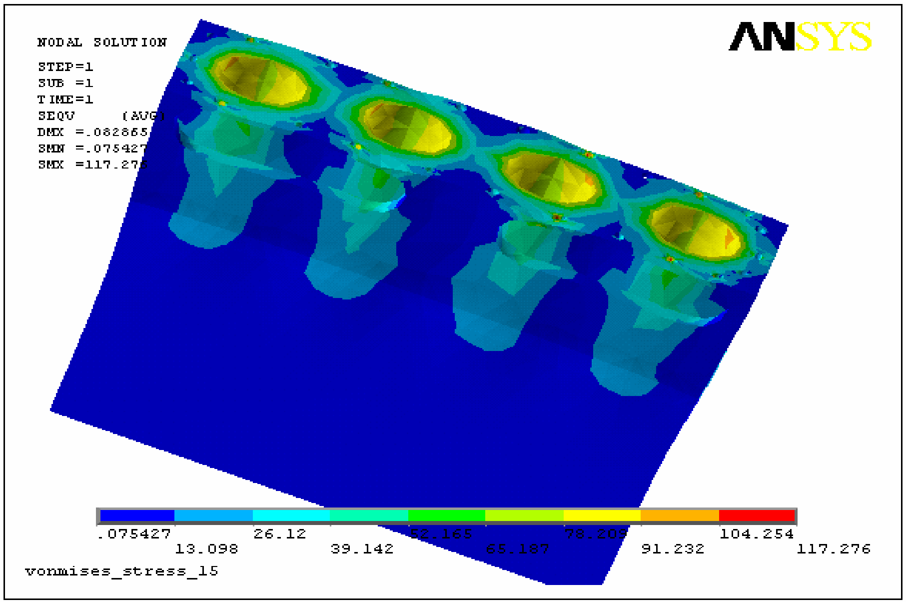
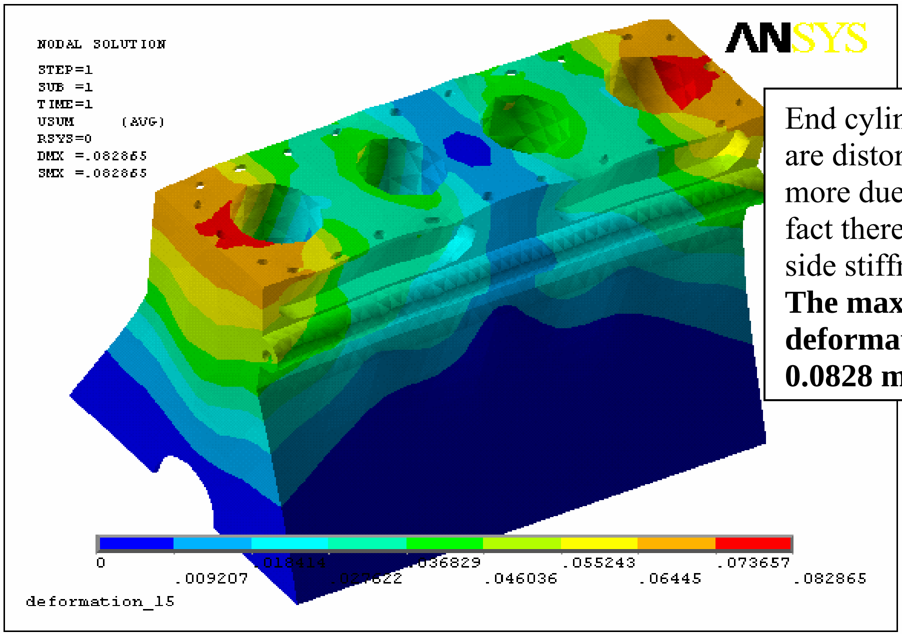


Figure 4.14 **Von-mises stress** (equivalent stress) contour of the engine block when **all four cylinders are firing** with a peak pressure of **150 bars**-different view



End cylinders are distorted more due to the fact there is no side stiffness. **The maximum deformation is 0.0828 mm.**

Figure 4.15 Deformed state of the engine block when **all four cylinders are firing** with a peak pressure of **150 bars**.

As can be seen form the above plots, inner surface of the all four cylinders are stressed to a greater stress levels. The maximum stress induced was about 118 N/mm^2 which is about 60 % of the strength of the cast iron material. The deformation profile shows that end cylinders are distorted more due to the fact that there is no side stiffness for the end cylinders as compared to the second and third cylinders.

4.10 Physical boundary conditions – Thermal loading without cooling

Engine block has important thermal functions such as transporting necessary amount of heat to the cooling mediums such as water and oil. Thermal boundary conditions are heat energy (gas at very high temperature with high heat transfer co-efficient) released when fuel is burned and convection cooling loads. The first stage in the thermal study was to simulate the engine block without any cooling mediums in order to see the temperature profile without cooling [27, 36, 39]. It is worthwhile to conduct this analysis because engine designer should fore see, how his engine block would behave if there is no cooling mediums for the engine that he designed and also that this analysis is the base from which engine designer can calculate how much cooling is necessary to keep the engine operating at its best performance state. Typical internal combustion engine leaves about 30% of the combustion energy to the cooling water. The boundary conditions are combustion gas is at a temperature of $2000\ ^{o}$C with a convection heat transfer coefficient of $1500 W/m^2K$ and other boundary condition is atmospheric boundary conditions on all the outer surface of the engine block.

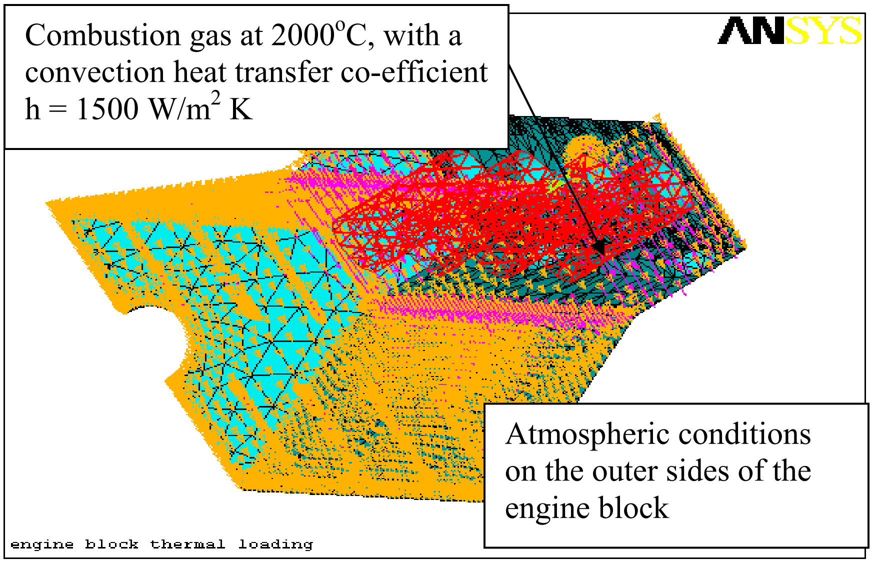
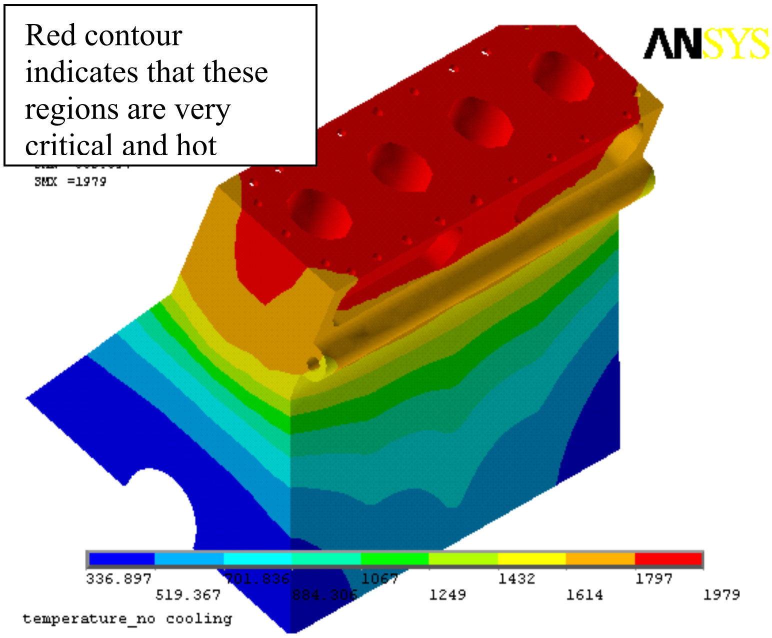


Figure 4.16 Thermal boundary condition of the engine block with no cooling mechanisms simulated.

4.11 Results and their physical interpretation (without any cooling)

After applying the thermal boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing Fourier heat transfer equations for each and every element and these formulated governing equations were solved for the temperatures from which all the other quantities such temperature gradient, and thermal flux can be calculated. Temperature distribution is shown below.



Red contour indicates that these regions are very critical and hot

Figure 4.17 Temperature distributions of the engine block with no cooling mechanisms simulated

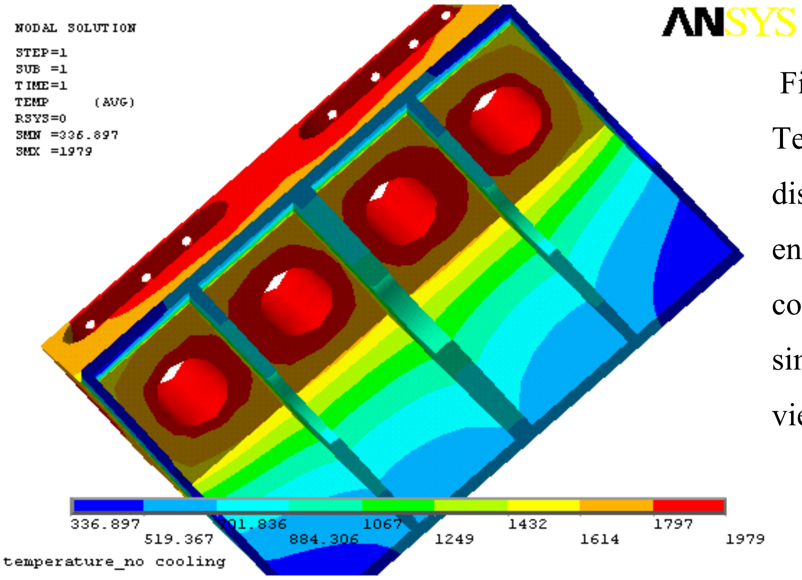


Figure 4.18 Temperature distributions of the engine block with no cooling mechanisms simulated- bottom view

As can be seen from the above temperature plots, in a steady state heat transfer condition whole engine block top surface reaches a very high temperature but the bottom regions are in low temperature as atmospheric convection heat load was applied on all the surrounding surfaces and bottom of the block. [37,39].The maximum temperature reached is about 1800 ^{o}C which is more than the melting point of the cast iron material. Therefore it is necessary to have dedicated cooling mechanisms for an engine block, without which engine block would certainly have problem to do its function.

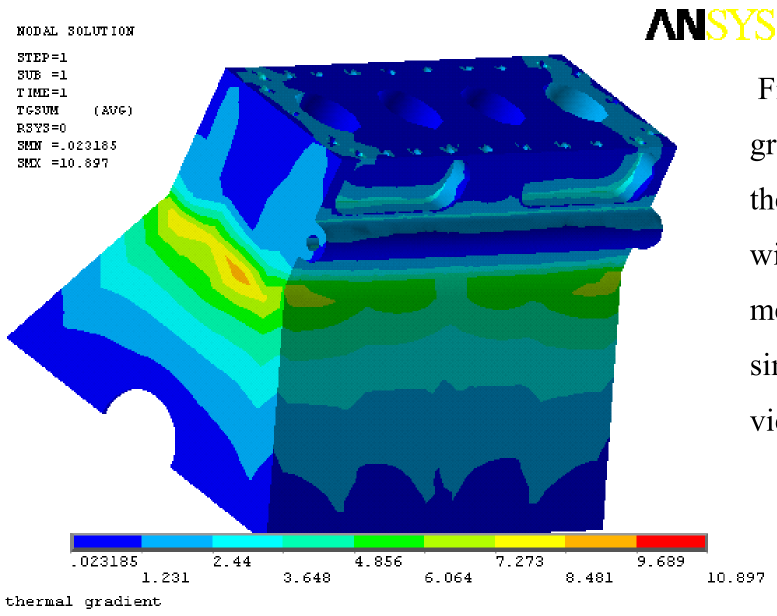


Figure 4.19 Thermal gradient contours of the engine block with no cooling mechanisms simulated-bottom view

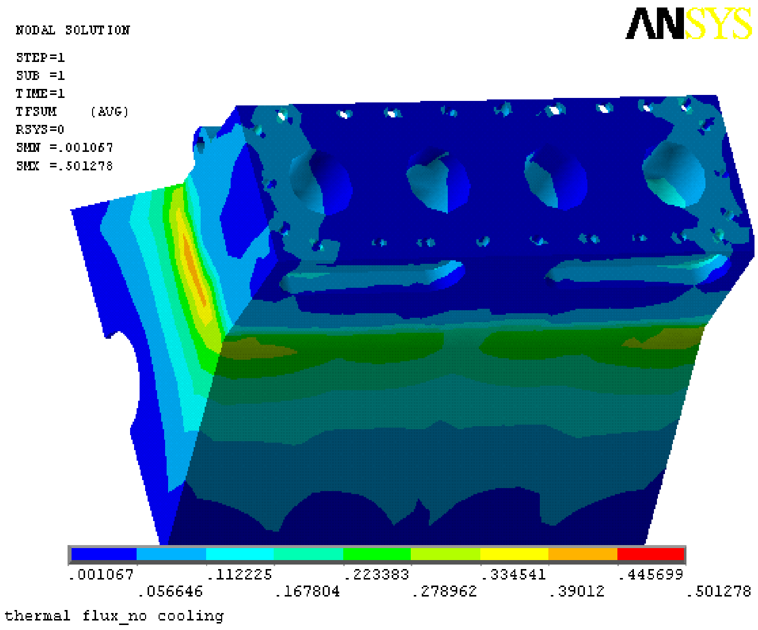
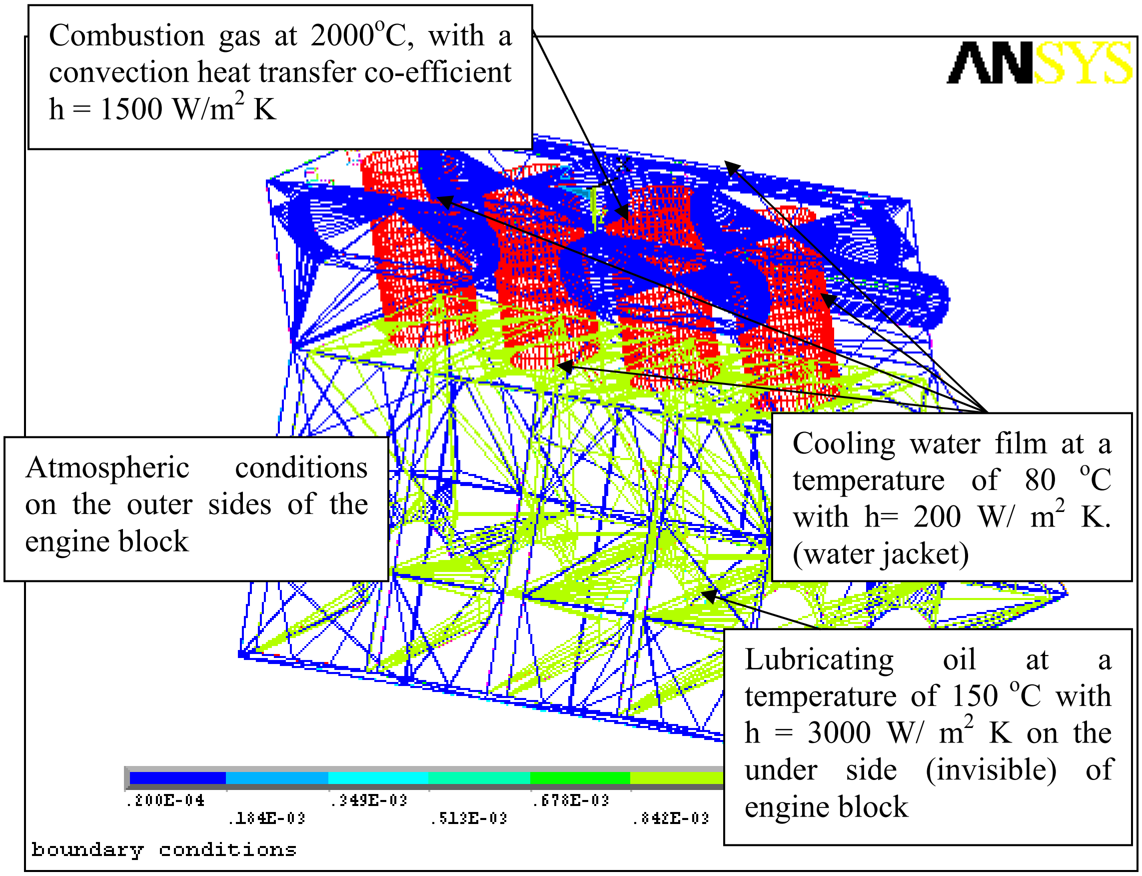


Figure 4.20 Thermal flux contours of the engine block without cooling mechanisms simulated -bottom view

4.12 Physical boundary conditions – Thermal loading with cooling

In this third stage of the thermal analysis, cooling mechanisms were simulated in addition to the combustion heat load and atmospheric cooling. The boundary conditions are combustion gas is at a temperature of 2000 °C with a convection heat transfer coefficient of 1500W/m^{2}K and other boundary condition is atmospheric boundary conditions on the outer surface of the engine block and cooling loads are water film with a bulk temperature of 80° C with a convection heat transfer coefficient h = 200 W/m^2 K. The other cooling mechanism is lubricating oil at a temperature of 150 ° C with a convection heat transfer co-efficient h = 3000 W/m^{2}K [1, 4, 3, 31, 41, 48].



Combustion gas at 2000°C, with a convection heat transfer co-efficient h = 1500 W/m^2 K

Atmospheric conditions on the outer sides of the engine block

Cooling water film at a temperature of 80 °C with h= 200 W/ m^2 K. (water jacket)

Lubricating oil at a temperature of 150 °C with h = 3000 W/ m^2 K on the under side (invisible) of engine block

Figure 4.21 Thermal boundary condition of the engine block with cooling mechanism simulated.

4.13 Results and their physical interpretation (with cooling)

After applying the thermal boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing Fourier heat transfer equations for each and every element and these formulated governing equations were solved for the temperatures from which all the other quantities such temperature gradient, and thermal flux can be calculated [1,3,41,48]. Temperature distribution is shown below.

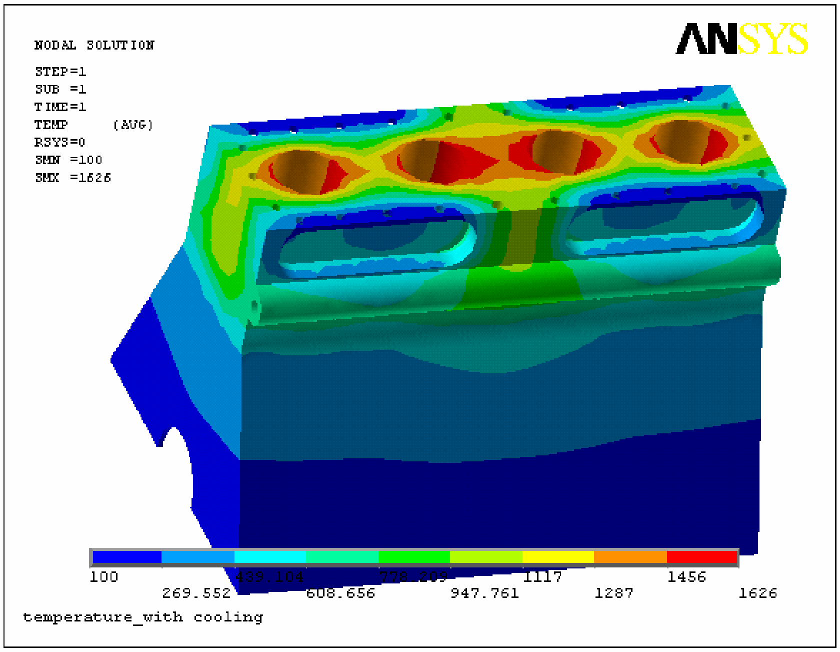


Figure 4.22 Temperature distributions of the engine block with cooling mechanisms simulated.

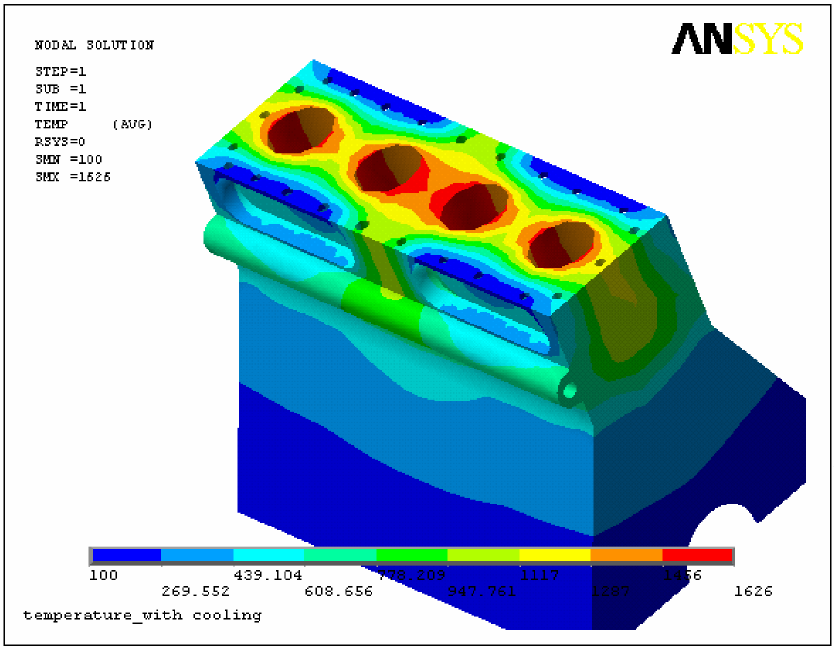


Figure 4.23 Temperature distributions of the engine block with cooling mechanisms simulated –different view

As can be seen from the above temperature plots, having cooling mechanisms in place, temperatures were brought down to very low value in most of the regions except the main combustion space. Now that most of the regions in the engine block are in controlled temperatures [46]. Only the main combustion space is still at a very high temperature this may be essential to have good thermal performance. Therefore it is necessary to have dedicated cooling mechanisms for engine block without which material melting may not be a surprise [45, 51].

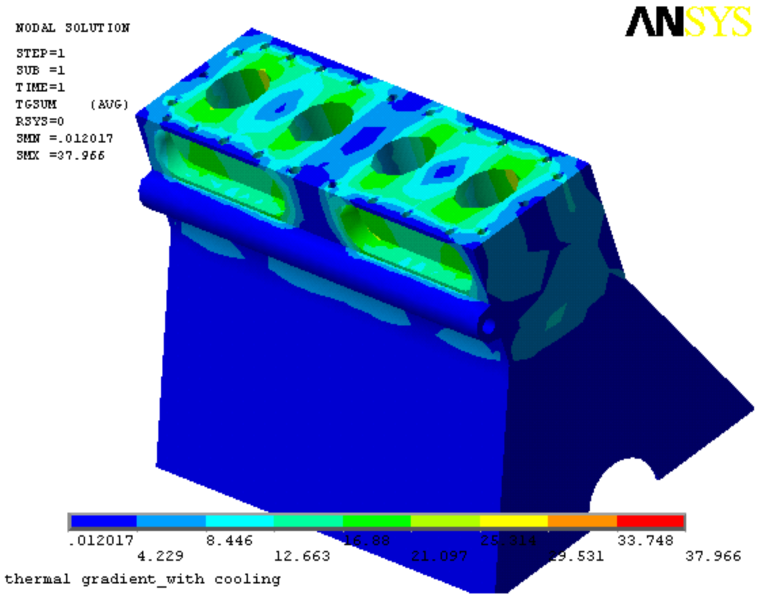


Figure 4.24 Thermal gradient contours of the engine block with cooling mechanisms simulated-bottom view

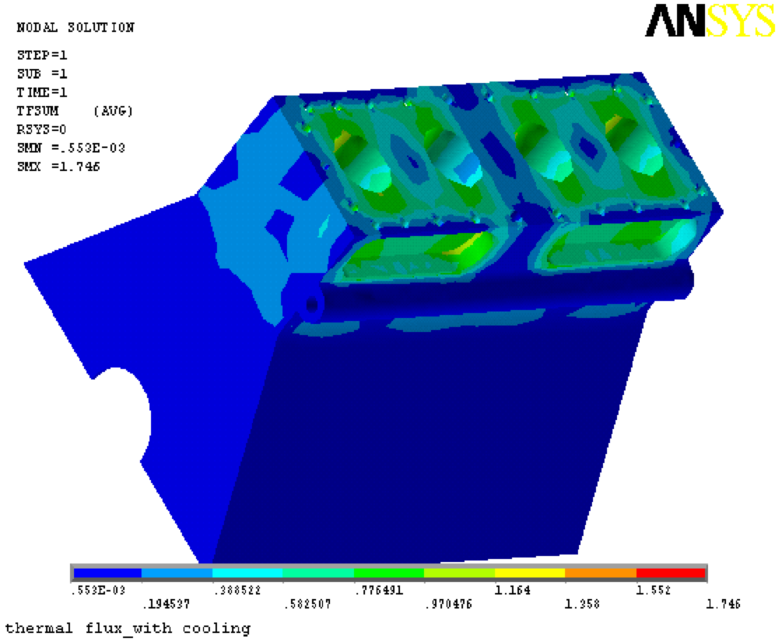


Figure 4.25 Thermal flux contours of the engine block with cooling mechanisms simulated -bottom view

As can be seen from the above plots, it is very clear that cooling mechanisms are essential for an engine. Most of the regions in the engine block were brought to a controlled temperature when the cooling mechanisms were simulated. Temperature gradient and thermal flux were identical as thermal flux is a direct function of temperature gradient. Although temperature is still 1600 ^{o}C in the combustion space, most of the regions in the engine were brought to low temperatures. As it has already been mentioned, this simulation was basically **peak moment simulation** in which critical moments (at the point of combustion) were simulated. So very practically speaking, temperature in the engine will have to rise to these levels even with water and oil cooling. An engine running at very stable condition, the temperature of inner surface of the cylinder is in the range of 400-500 ^{o}C. This range is in fact varies during burning cycle. This range is maximum during the combustion moments. Therefore the average temperature of 750 ^{o}C during the combustion period is absolutely expectable.

4.14 Summary

- Engine block is subjected to both structural and thermal loads.
- Analysis carried out was basically a “**peak moment simulation”**
- Structural load is gas pressure and thermal load is heat energy released by the burning fuel and convection water and oil cooling
- Combustion pressure induces the structural stresses in the engine block
- The very main function of the engine block is to conduct away the necessary amount of heat to the cooling mediums such as water and oil in order to survive itself otherwise it would have to melt.
- Engine block should have to retain the sufficient of heat in order to sustain the combustion and higher thermal efficiency.
- The average temperature of the upper portion of the engine block when there is no sort of cooling mechanism was about 1400 ^{o}C against the 750 ^{o}C when it enjoys water and oil and, atmospheric cooling.
- Thermal gradient is appreciable at the middle portion of the engine block may be because of the fact that middle portion of the engine block forms the main cooling path.

CHAPTER-5

Finite Element Stress Analysis of Connecting rod

Connecting rod is the kinematical link that connects the piston to the crankshaft. As the crankshaft is being the ultimate load-bearing member in an internal combustion engine, connecting rod is a force and motion-transferring member. Therefore, whatever the load that has to be transferred to the crankshaft, that load must have to go through the connecting rod. Therefore, the simulation of working conditions of the connecting rod becomes essential. The finite element stress analysis of connecting rod was performed in order to find out the compression stresses induced in it as a punishment for the gas force transfer.

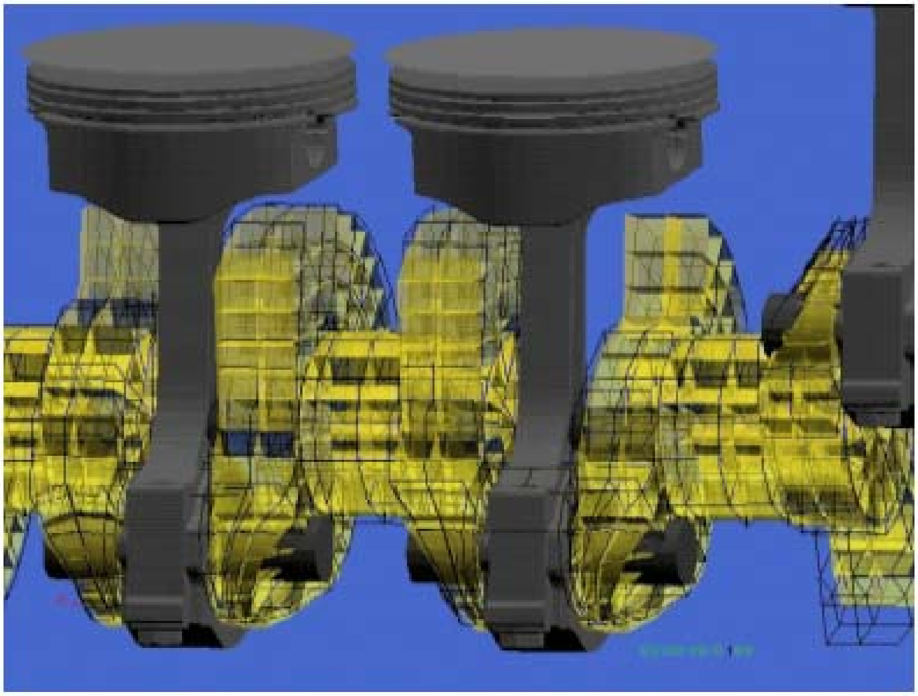


Figure 5.1 The Moment when connecting rod transfers the gas load to the crankshaft

5.1 What is the need to conduct FEA analysis of connecting rod?

These are the possible reasons that necessitate conducting FEA analysis of connecting rod [34].

- To check whether connecting rod sustains the compressive gas force that is transferred through it at the point of combustion.
- To find out the maximum stress induced due to compressive gas force
- To plot the deformed state of the connecting rod due to compressive gas force
- To find out the best configuration and suitable material for connecting rod

Engine designer will be in the position to answer these questions only when he conducts FEA analysis of connecting rod.

5.2 Physical problem

The peak combustion pressure at the point of combustion of a typical diesel engine is in the range of 70 – 120 bars depending upon the application. Therefore, the connecting rod has to be designed to sustain this gas load. This gas load induces compression stresses in the connecting rod.

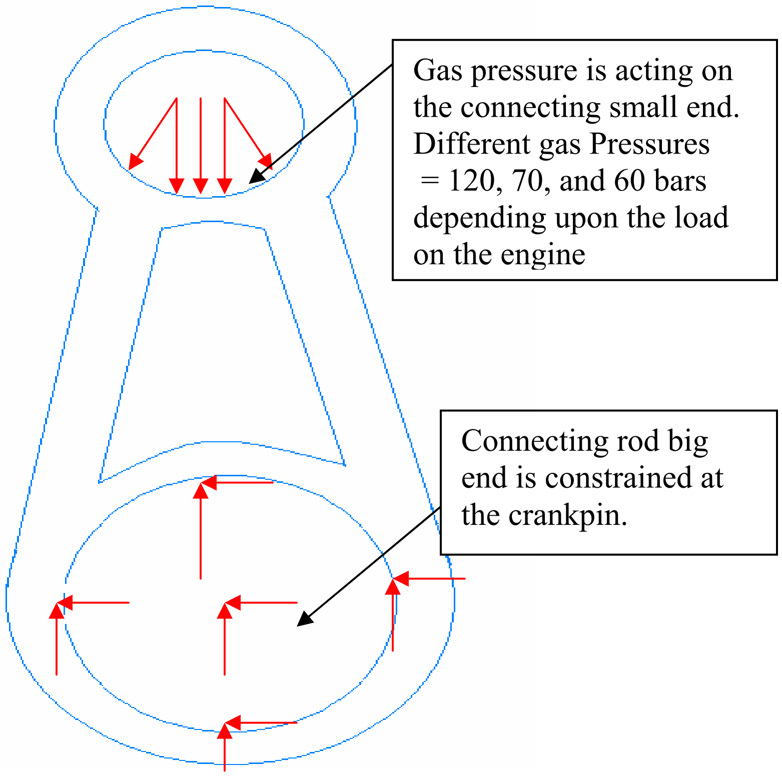


Figure 5.2 Statement of the physical problem

5.3 Approach of the analysis

Connecting is subjected to compressive forces during the explosion stroke (power stroke). The connecting has to transfer different gas a force depending upon the load on the engine [34].Though connecting rod is subjected to cyclic loads, maximum stress distortion is occurring at the moment of combustion. Therefore, current study analysis the connecting rod for different gas loads depending upon load on the engine. Therefore following approach was used to carry out the analysis.

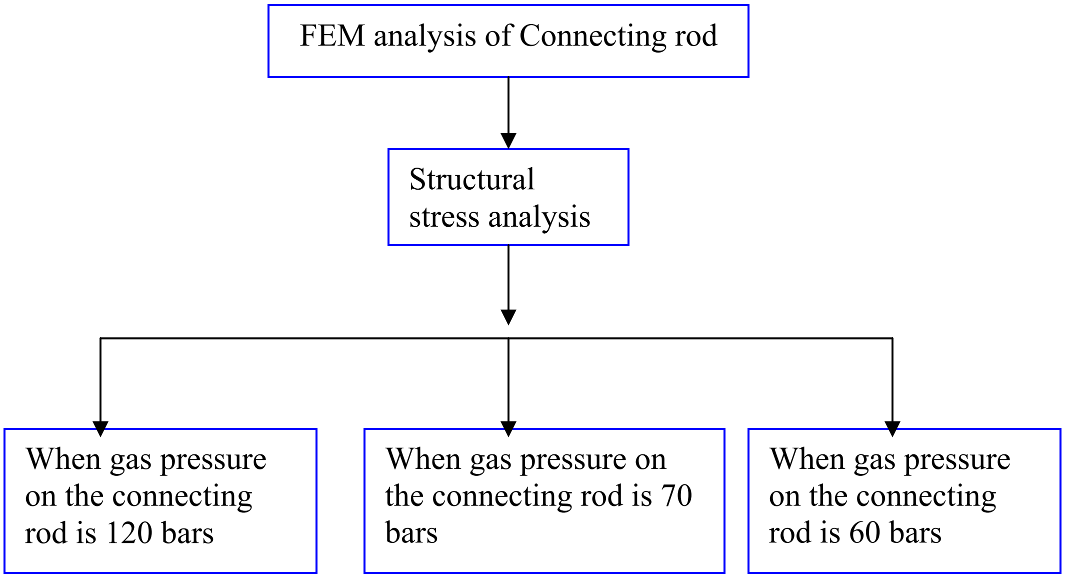


Figure 5.3 Approach of the analysis

5.4 Inputs required to carry out this stress analysis

- Approximate geometry of the connecting rod
- Materials properties of the connecting rod material such as [5]
    - Young's modulus of Alloy steel-1040-heat treated (E = 120 GPa)
    - Poisson's ratio (μ=0.30)
    - Yield strength of Alloy steel -1040-heat treated (S =375 MPa)

5.5 Typical FEM procedure to carry out this analysis

- **Geometry** creation (solid modeling of connecting rod)
- Dividing the solid model into number of elements and then connecting these elements with each other (**meshing**) and thereby obtaining the finite element model.
- Applying boundary conditions (defining pressure loads)
- Solving the problem
- Results and their interpretation.

5.6 Geometry creation and meshing

Geometry of the connecting rod is needed to carry out this analysis. Solid model of the connecting rod was created in solid modeling environment called “**CATIA V5**”.Solid model is the base for carrying out any FEA analysis. Solid model of the connecting rod that was created in **CATIA V5** solid modeling environment is shown in the figure 5.4. The created solid model of the connecting rod was **brought in** to the ANSYS analysis environment using **ANSYS-CATIA** interface. Solid model used in any FEA analysis can be very simple and at the same time it reflects the physical reality of the connecting rod. Therefore, small small features like rounds, chamfers, holes have not been modeled. As shown in the figure below, solid model is very simple but at the same time it reflects a physical reality. There are so many designs of connecting rod. The common one is two piece connecting rod in which two pieces are connected to each other with the help of bolts. Therefore, connection can be considered as a rigid and can be treated as single piece connecting rod. Solid model will have to be turned into to FEM model by dividing the solid model into number of small small elements. This process is called “**discritization**”.Discritization is the process of dividing the solid model into finite number of elements. After dividing the solid model into finite number of elements, they have to be connected to each other as the solid volume is continuous and physically connected at each and every material point.

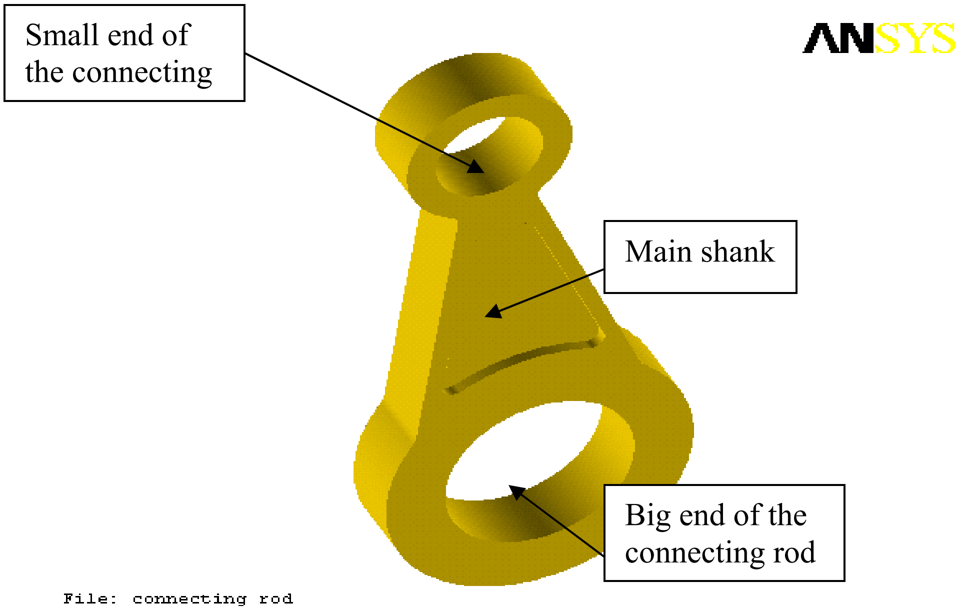


Figure 5.4 Solid model of the connecting rod created in CATIA V5 and then imported into ANSYS analyzing environment using ANSYS-CATIA interface.

In simple terms meshing means that connecting the elements with each other. An element is the building block of the finite element model. Therefore, the type of element (linear element or higher order elements), number of nodes and their capabilities are important parameters for selecting the elements for a particular analysis. Meshed model of the connecting rod is shown below.

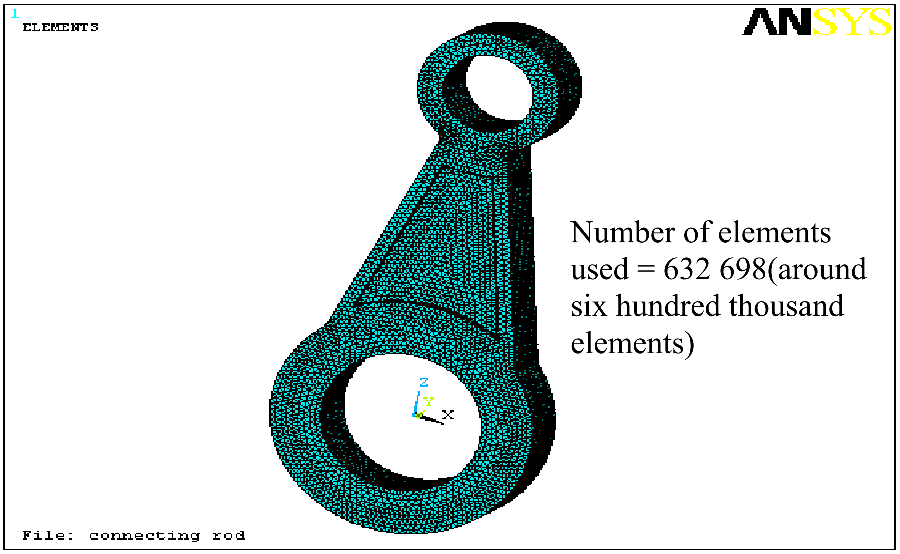
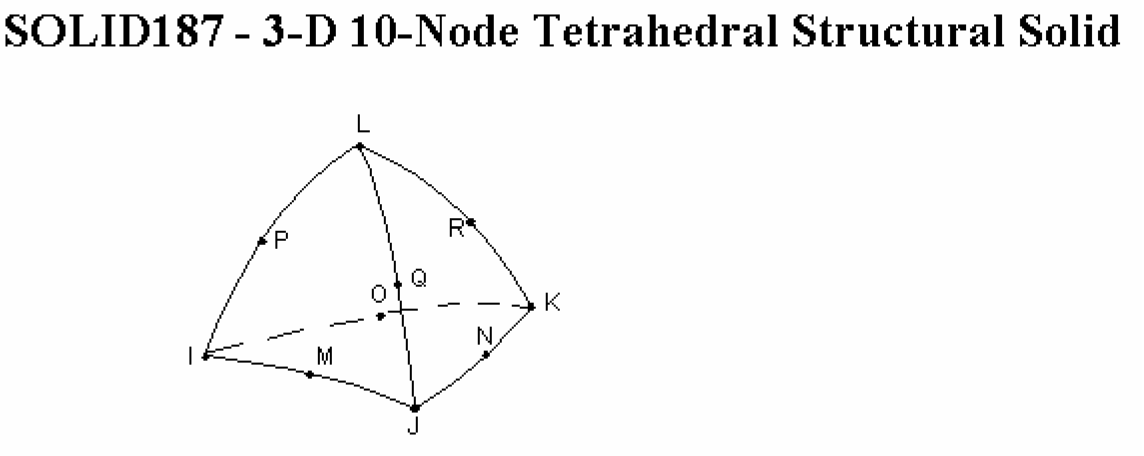


Figure 5.5 Meshed model of the connecting rod

In this analysis, the element used was **Tet 10 node3 SOLID 187** which is a solid element which has all six degrees of freedom (translation in all X, Y, and Z and rotations about all three axes); it also has stress stiffening and buckling capabilities. This element is shown below.



SOLID187 - 3-D 10-Node Tetrahedral Structural Solid

Figure 5.6 Element name – Tet 10 node SOLID 187

Meshing is an important step in FEA analysis. Mesh must be adequate enough to obtain the accurate results while keeping the computational time in mind. More the number of elements, better the accuracy, on the other hand more the number of elements more the computational time. The number of elements used in this analysis was 632 698(about eight hundred thousands) .In order to obtain accurate information about the velocity gradients and temperature gradients, more elements have been placed at the corners, rounds, and at the holes by using ANSYS **smart meshing** capability. Smart meshing is the special feature in ANSYS meshing capability, which places more number of elements at points where velocity gradient, pressure gradient, and temperature gradients are more. The ultimate purpose is to obtain more accurate information at the very sensitive places.

5.7 Physical boundary conditions

Connecting rod is subjected to compressive gas force during the combustion period. The combustion gas pressure was applied on the small end of the connecting rod and big end is constrained by the crankshaft which forms the physical support and reaction provider. Connecting rod is subjected to different gas pressures depending upon the load on the engine [34]. There fore connecting was virtually simulated for three different gas loads. The gas pressures were 120, 70, and 60 bars (diesel engine peak combustion pressures).The applied boundary conditions are shown below. Gas pressure was not applied on the complete inner circular surface of the small end of the connecting rod; it was applied only on the bottom semi circular surface only [34]. Similarly in the big end, only upper semi circular surface was constrained in the all the three directions as it is the physical reality.

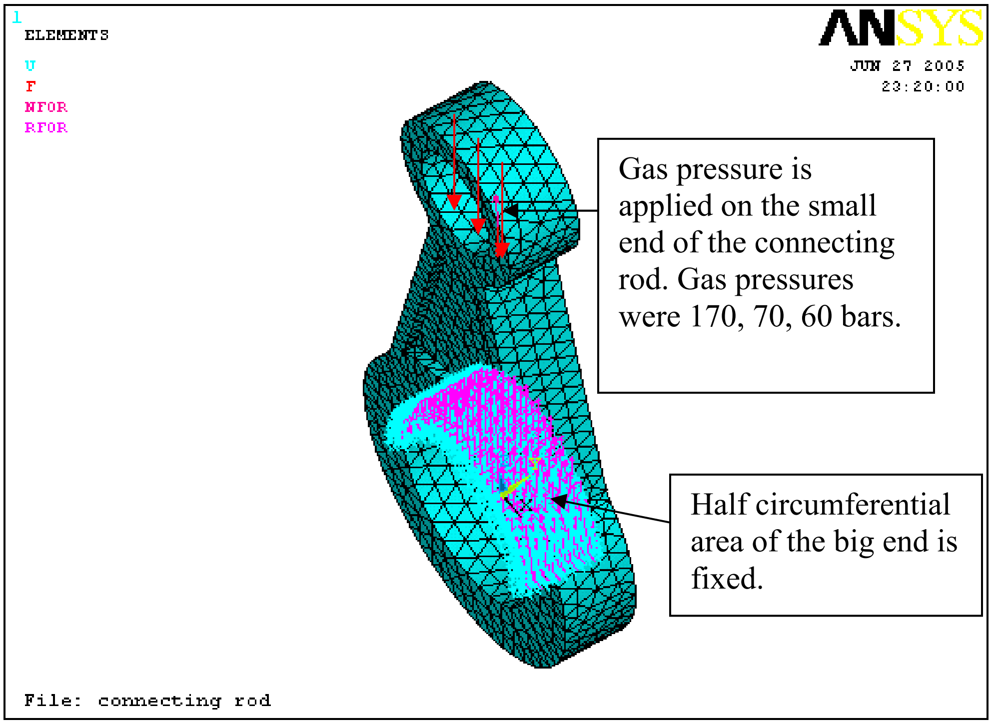
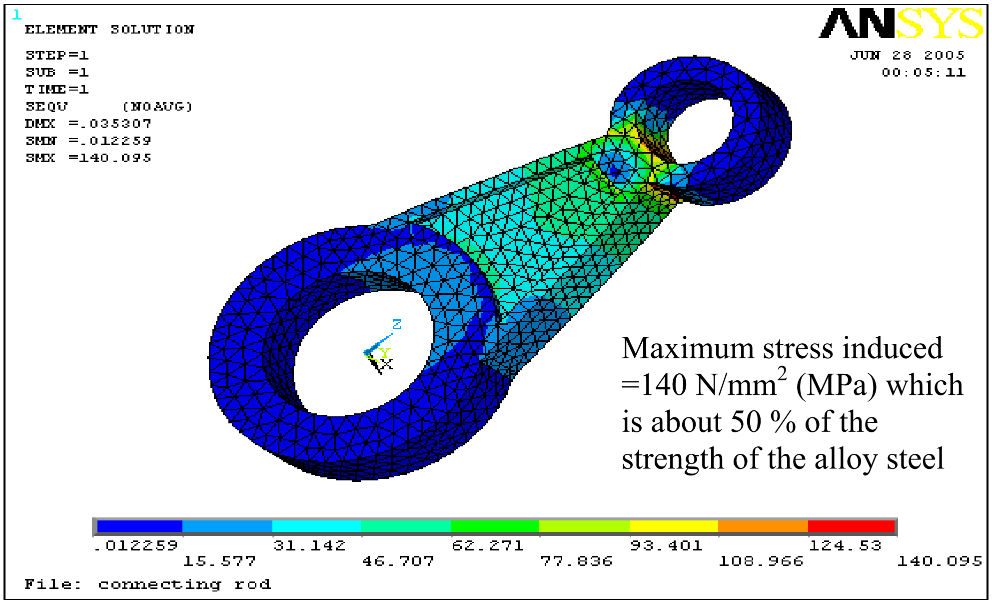


Figure 5.7 Physical boundary condition of the connecting rod.

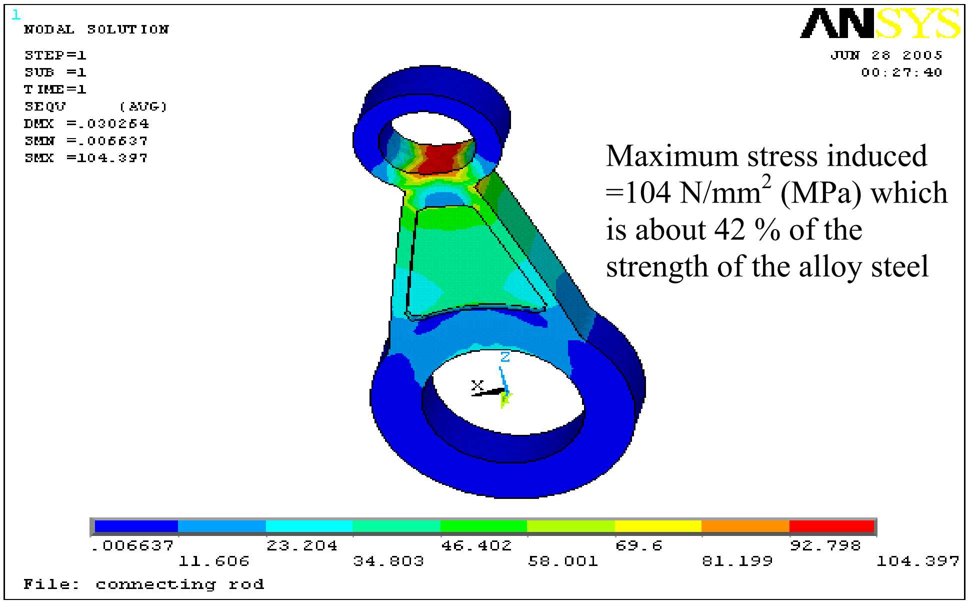
5.8 Results and their physical interpretation

After applying the boundary conditions, the problem was solved by the ANSYS Iterative Solver. ANSYS solver formulates the governing structural stress strain equations for each and every element and these formulated governing equations were solved for the deformations from which all the other quantities such stresses, strains etc can be calculated.



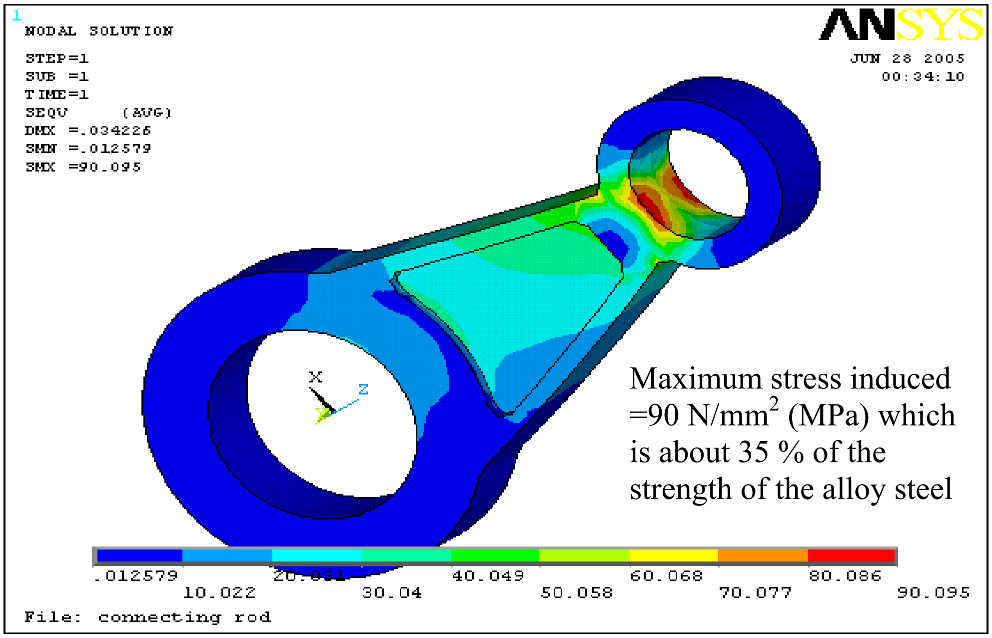
Maximum stress induced =140 N/mm^2 (MPa) which is about 50 % of the strength of the alloy steel

Figure 5.8 **Von-mises stress** (equivalent stress) contour of the connecting rod when gas load was 120 bars.



Maximum stress induced =104 N/mm^2 (MPa) which is about 42 % of the strength of the alloy steel

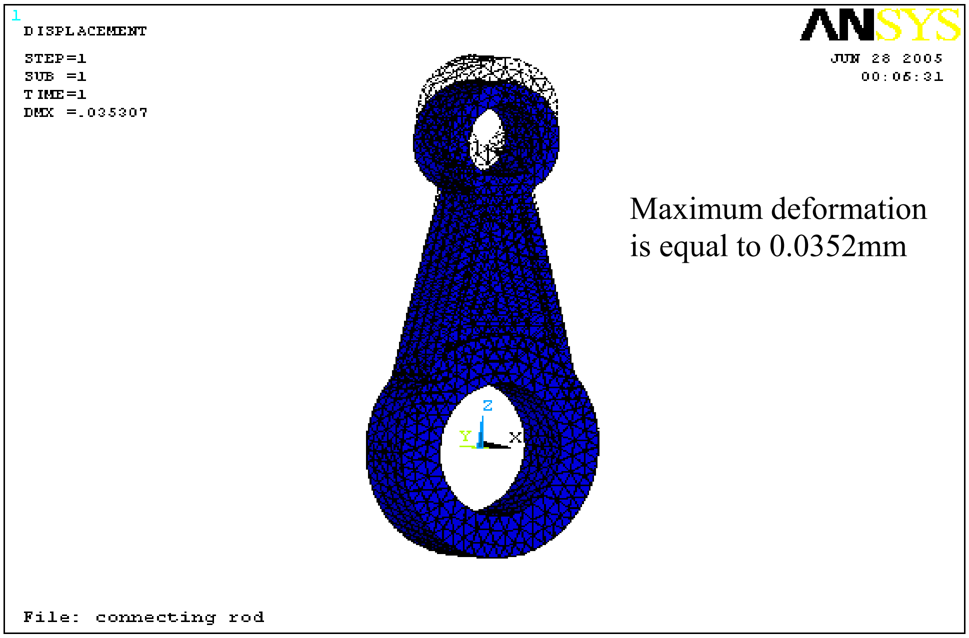
Figure 5.9 **Von-mises stress** (equivalent stress) contour of the connecting rod when gas load was 70 bars.



Maximum stress induced =90 N/mm^2 (MPa) which is about 35 % of the strength of the alloy steel

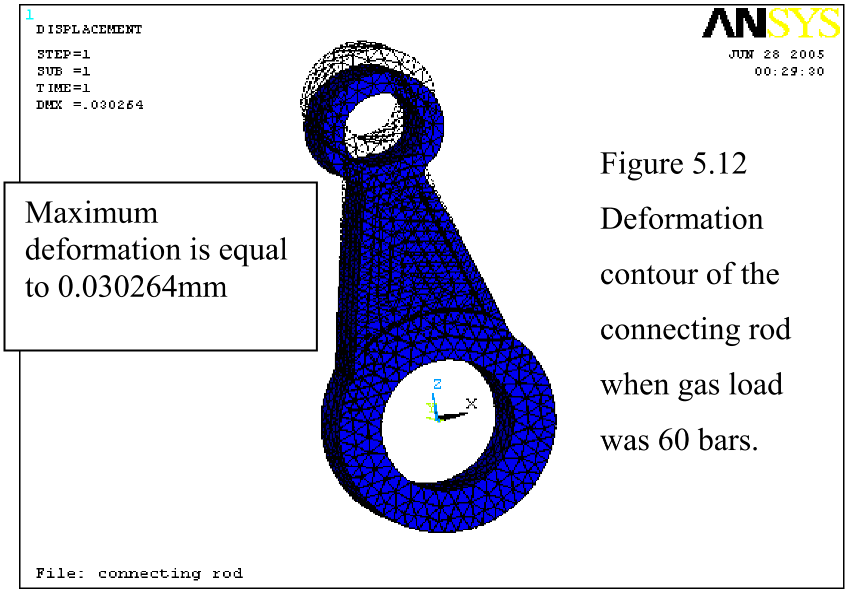
Figure 5.10 **Von-mises stress** (equivalent stress) contour of the connecting rod when gas load was 60 bars.

As can be seen from the above plots, the maximum compressive stresses induced in the rod were about 140, 104, and 90 **N/mm^2** for the gas loads of 120, 70, and 60 bars respectively, which are less than 50 % strength of the of the **steel alloy-1040** taken for the study. There fore there has been a factor of safety of 2 which is partially acceptable.



Maximum deformation is equal to 0.0352mm

Figure 5.11 Deformation contour of the connecting rod when gas load was 120 bars.



Maximum deformation is equal to 0.030264mm

Figure 5.12 Deformation contour of the connecting rod when gas load was 60 bars.

As can be seen from the above plots, deformation and stresses are varying directly with the gas load, which were physically expected [34]. An FEA simulation engineer can simulate different combustion pressures and different geometry of the connecting rod as a sensitive design studies (design changes).

5.9 Summary

- Connecting rod is a structural member designed mainly to take compressive stresses.
- Connecting rod will fail structurally if stress on the connecting rod is greater than the yield strength of the material from which rod is made
- FEA simulation can be conducted on the connecting rod to see what are the peak stresses and deformations induced due to the application of gas loads.
- Solid model of the connecting was created in CATIA and then imported into the ANSYS analyzing environment using CATIA-ANSYS interface.
- Connecting rod was simulated for different gas loads as rod is subjected to different gas forces depending upon load on the engine.
- Maximum stresses induced were about 140, 104, and 90 **N/mm^2** for the gas loads of 120, 70, and 60 bars respectively, which are less than 50 % strength of the of the **steel alloy-1040** taken for the study.

CHAPTER -6

Computational Fluid Dynamics (CFD) analysis Of Exhaust Manifold

Exhaust Manifold is a simple pipe, which carries the burned gases. In a multi cylinder engine, individual pipes collect the combustion products from various cylinders and leads to a single out let pipe. Exhaust manifold is subjected to both thermal and flow boundary conditions (loading) [49, 54, 65].

6.1 Basic functions of the exhaust manifold

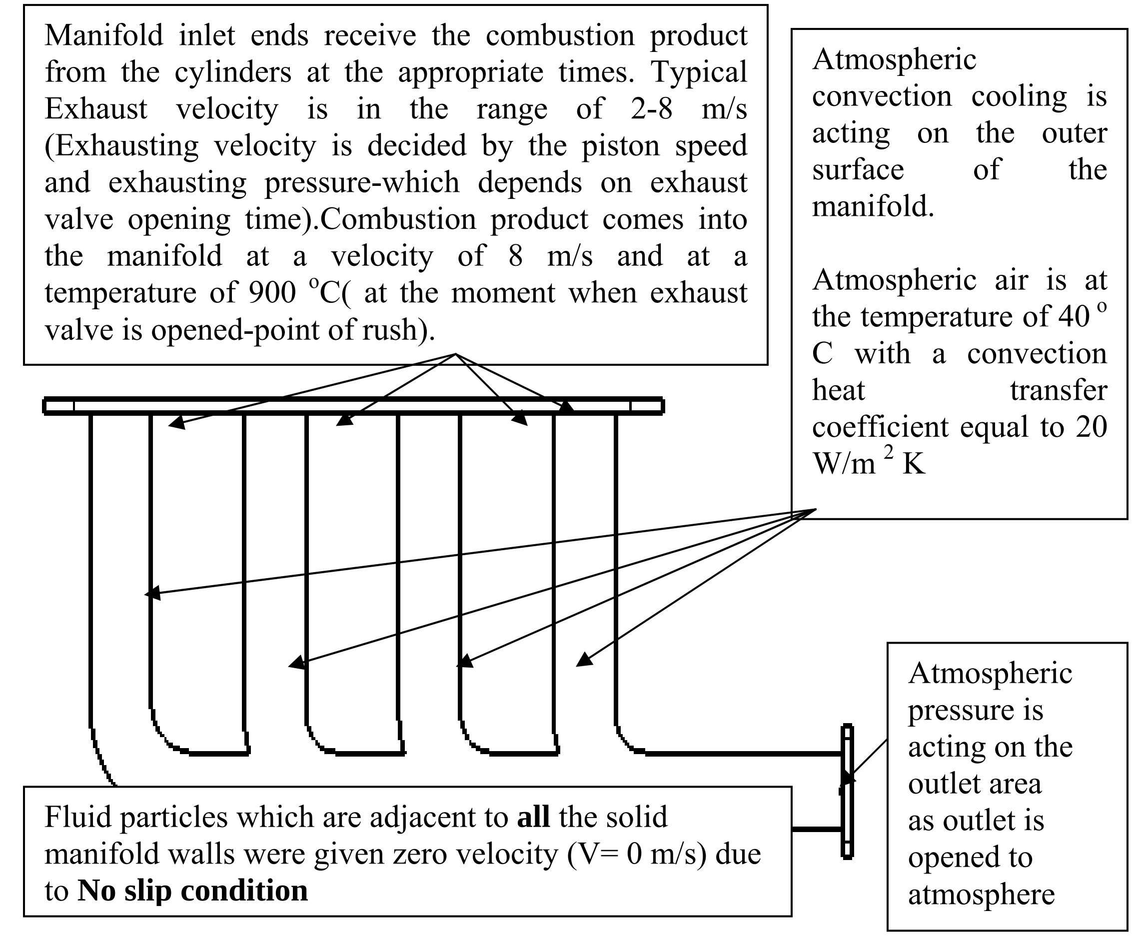
- It is a pipe which carries away the burned gases out of the engine cylinders
- It insulates the exhaust gases from the surrounding so that catalytic converter functions well as function of the catalytic converter depends on the prevailing temperature in the manifold, which in turn indirectly depends on amount of heat flows out of manifold due to heat transfer to the surrounding.
- It conducts away sufficient amount of heat so that inner surface temperature of the exhaust manifold is less than the melting point of the exhaust manifold.

Figure 6.1 Exhaust manifold for a three-cylinder engine

Exhaust flow is very complex phenomenon as it involves both thermal and fluid aspects. In this analysis, both were taken into account. Velocity, pressure and temperature distributions in the manifold are primary importance [79,80]. This velocity, pressure and temperature distributions are, normally, obtained by carrying out the CFD analysis. Hence, CFD analysis was carried out and velocity and pressure profiles were obtained.

6.2 Physical Problem

The physical problem is shown in the figure 6.2.Exhaust manifold is a pipe, which carries the burned gases out of the engine. The combustion products (with some velocity, pressure, and temperature) flow through the manifold. The typical velocity of the combustion products at the inlet is in the range of 2-8 m/s, which is decided by the piston velocity and exhausting pressure [49]. In a typical internal combustion engine, exhausting pressure is more than then atmospheric pressure in order to facilitate the exhaust flow otherwise They have expelled out and outlet pressure is equal to atmospheric pressure.



Manifold inlet ends receive the combustion product from the cylinders at the appropriate times. Typical Exhaust velocity is in the range of 2-8 m/s (Exhausting velocity is decided by the piston speed and exhausting pressure-which depends on exhaust valve opening time).Combustion product comes into the manifold at a velocity of 8 m/s and at a temperature of 900 ^{o}C(at the moment when exhaust valve is opened-point of rush).

Atmospheric convection cooling is acting on the outer surface of the manifold.

Atmospheric air is at the temperature of 40 ^{o} C with a convection heat transfer coefficient equal to 20 W/m ^{2} K

Atmospheric pressure is acting on the outlet area as outlet is opened to atmosphere

Fluid particles which are adjacent to **all** the solid manifold walls were given zero velocity (V= 0 m/s) due to **No slip condition**

Figure 6.2 Statement of the physical flow problem

There are three major kinds of analysis normally performed on exhaust manifold.
They are [49, 54, 65]

- Conduction heat transfer analysis (To obtain temperature distributions, heat flow rates, heat flux and temperature gradient)
- Thermal stress analysis (due to temperature gradient and constraining)
- CFD analysis to plot **velocity, pressure and temperature** distributions throughout the manifold

In the current study, only CFD analysis was performed.

6.3 Objectives of this analysis [49]

- To plot the velocity distributions of the combustion products going through the exhaust manifold
- To plot the pressure distributions of the combustion products going through the manifold
- To visualize how the combustion products flow through the exhaust manifold
- To check out the fluid flow volume (by identifying any unnecessary flow volume in the manifold)
- Results of this analysis can be used to optimize the manifold geometry.

6.4 Inputs required carrying out the CFD analysis [49, 61, 54, 79, 80]

- Approximate geometry of the manifold
- Flow properties of combustion products such as
    - Density of the combustion products (ρ) = 1.23 kg/m^3
    - Viscosity of the combustion products (n)= 2e-5 N-s/m^2
    - Velocity with which combustion products enter into the manifold (V)
    - Temperature of the combustion products (T)
    - Thermal conductivity of the combustion product (k) in W/mK

6.5 Typical CFD procedure to carry out this analysis

- Geometry creation (solid modeling of Exhaust manifold)
- Dividing the manifold solid flow volume (solid model) into number of elements and then connecting these elements with each other (meshing) and thereby obtaining the finite element model.
- Applying boundary conditions (defining velocity, pressure,convetion and temperature boundary conditions)
- Solving the problem
- Results and their interpretation.

6.6 Geometry creation and meshing

Geometry of the exhaust manifold is needed to carry out CFD analysis. Solid model of the exhaust manifold was created in solid modeling environment called “CATIA V5”.In a typical CFD analysis, solid model represents the flow volume not the pipe through which fluid is flowing .Solid model is the base for carrying out any FEA or CFD analysis. Solid model of the exhaust manifold that was created in **CATIA V5** solid modeling environment is shown in the figure 6.3. The created solid model of the exhaust manifold was brought in to the ANSYS analysis environment using ANSYS-CATIA interface. Solid model used in any CFD analysis can be very simple and at the same time it reflects the physical reality of the manifold. Therefore, small small features like rounds, chamfers have not been modeled. As shown in the figure below, solid model is very simple but at the same time it gives a physical reality [79, 80]. Solid model will have to be turned into to FEA model by dividing the flow volume (solid model) into number of small small elements. This process is called “**discritization**”.Discritization is the process of dividing the solid model into finite number of elements. After dividing the solid model into finite number of elements, they have to be connected to each other, as the flow volume is continuous and physically connected at each material point.

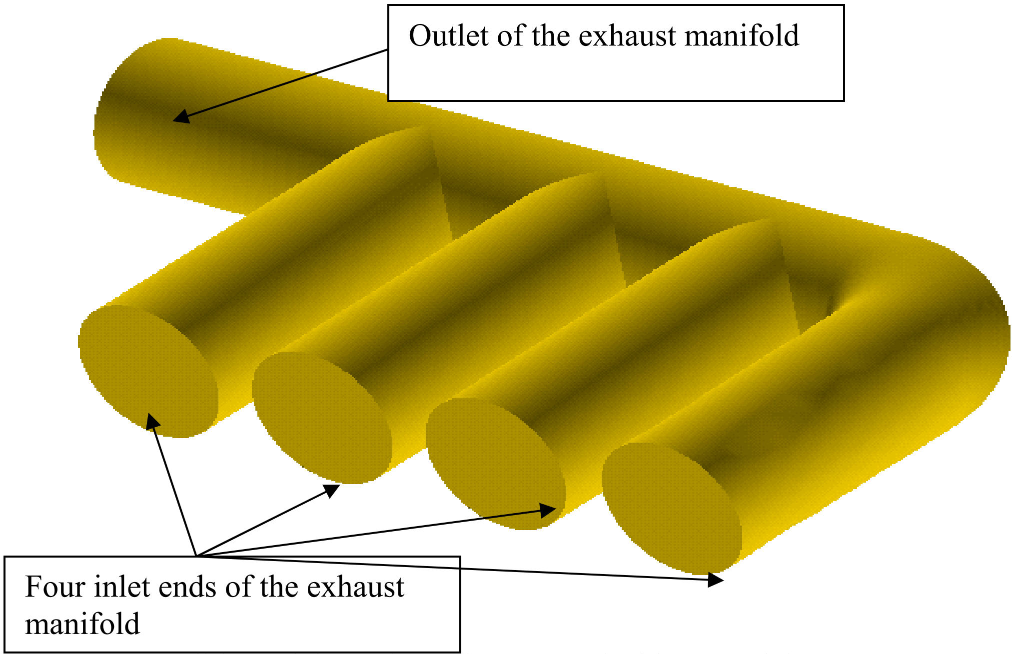


Figure 6.3 Solid model of the exhaust manifold created in CATIA V5 and then imported into ANSYS analyzing environment.

In simple terms meshing means that connecting the elements with each other. An element is the building block of the finite element model. Therefore, the type of element (linear element or higher order elements), number of nodes and their capabilities are important parameters for selecting the elements for particular analysis. In this analysis, the element used was **3D FLOTRAN 142** which is a 3D element used in 3D CFD analysis.

FLUID142 Geometry

Fluid element that was for the analysis is shown below.

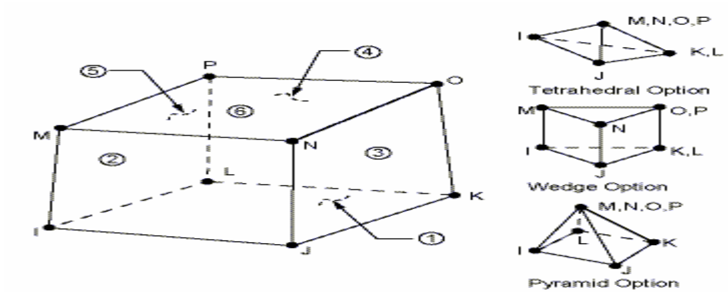


Figure 6.4 Element name – 3D FLOTRAN 142

Since exhaust manifold geometry has complex curves, smart meshing capability was used to mesh the solid model. Since velocity and pressure gradient may be significant enough, fine meshing was used in order to accurately capture the flow quantities. The meshed model of exhaust manifold is shown in figure 6.5.

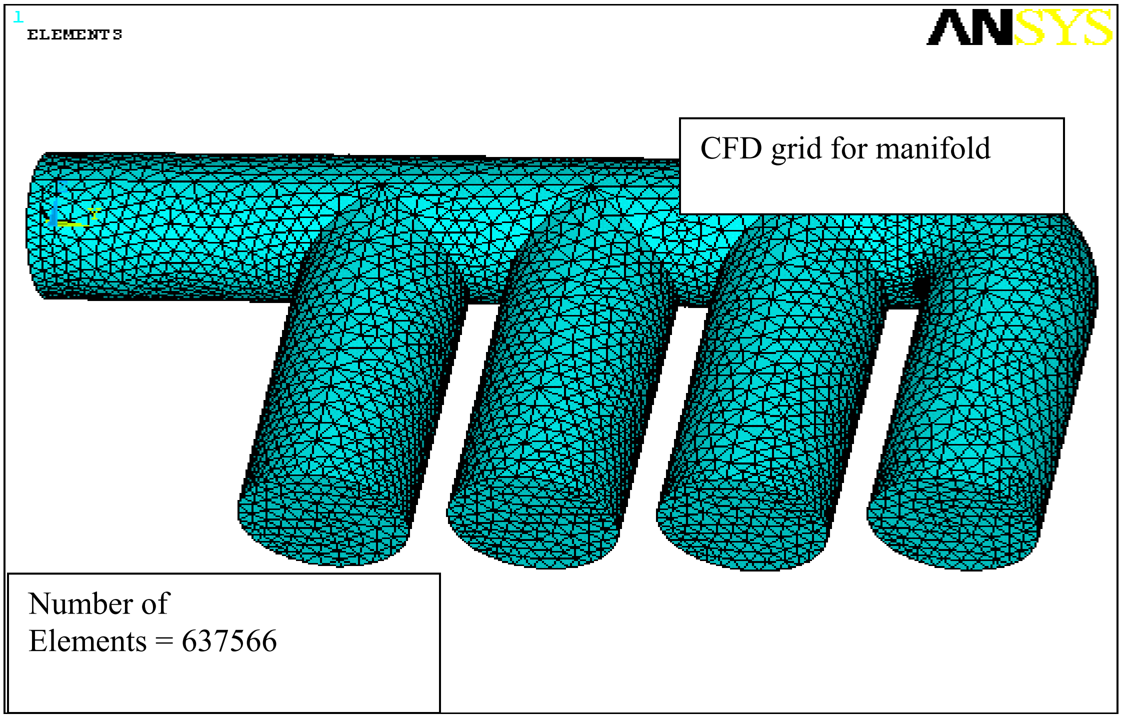
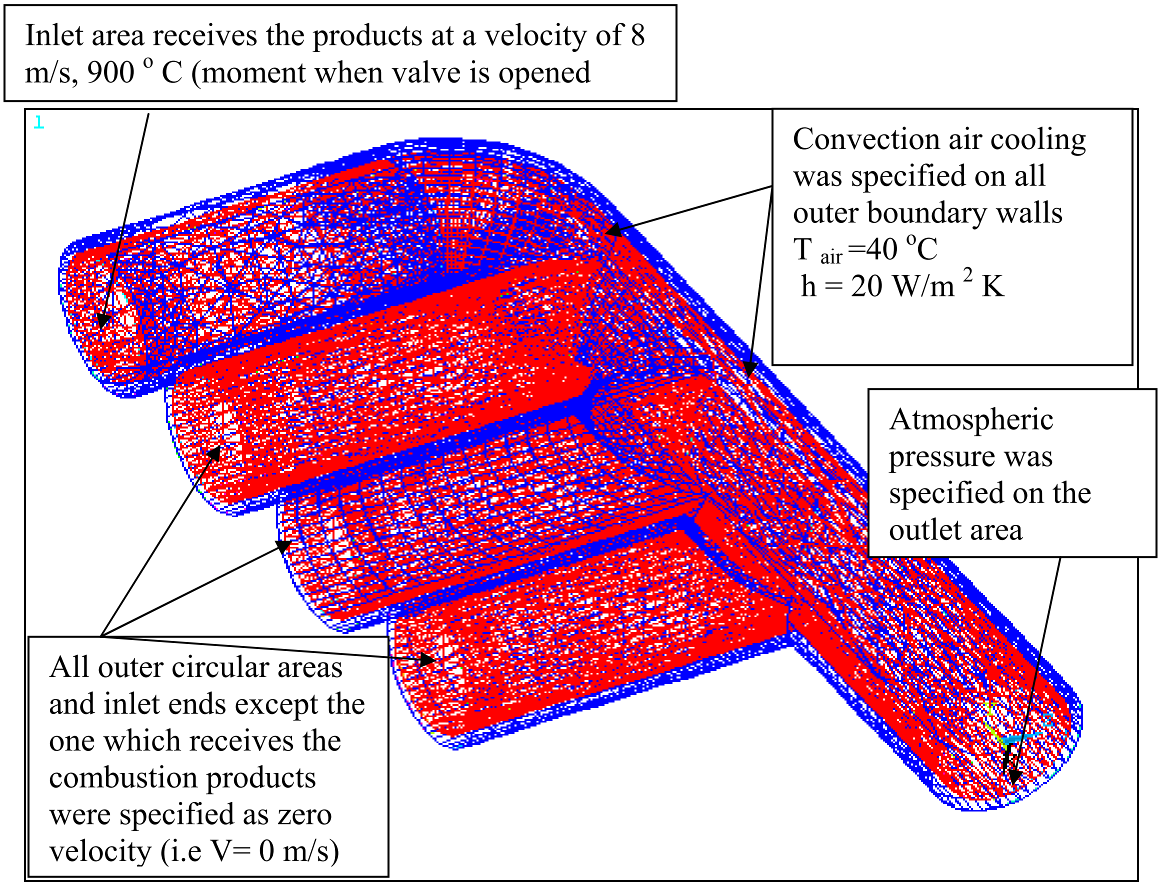


Figure 6.5 Meshed model of the exhaust manifold

Meshing is an important step in CFD analysis. Mesh must be adequate enough to obtain the accurate results while keeping the computational time in mind. More the number of elements, better the accuracy on the other hand more the number of elements more the computational time. The number of elements used in this analysis was 637566 (six hundred thousands) .In order to obtain accurate information about the velocity and temperature distributions, more number of elements have been placed at the corners, rounds by using ANSYS **smart meshing** capability [73, 74]. Smart meshing is the special feature in ANSYS meshing capability, which places more number of elements at points where velocity gradient, pressure gradient, and temperature gradients are more. The ultimate purpose is to obtain more accurate information at the very sensitive places.

6.7 Physical boundary conditions (loading)

The inlet ends of the manifold receive the combustion products and outlet area of the manifold lets the combustion products to leave the manifold. Inlet velocity of the combustion products was taken to be 8 m/s.Inlet velocity was specified as per the firing order. All the outer circumferential surfaces and front faces except the inlet end were given zero velocity as they are boundary walls (V=0m/s). Atmospheric pressure conditions have been specified at the outlet end of the manifold as it is opened to atmosphere. Exhaust manifold was simulated by assuming that exhausting is taking place in the first cylinder and then in the third cylinder [49, 54, 79, 80, 67]. Since combustion product is mixture of hydrocarbons, carbon compounds, and air, the average density of 1.23 kg/m^3 was assumed.Viscocity of he combustion product was assumed to be 2e-5N-s/m^2 [49, 51, 54, 65].



Inlet area receives the products at a velocity of 8 m/s, 900 ^{o} C (moment when valve is opened

Convection air cooling was specified on all outer boundary walls
T_{air} =40 ^{o}C
h = 20 W/m ^{2} K

Atmospheric pressure was specified on the outlet area

All outer circular areas and inlet ends except the one which receives the combustion products were specified as zero velocity (i.e V= 0 m/s)

Figure 6.6 Physical boundary conditions on the exhaust manifold

6.8 Approach of the analysis

As we know that exhausting velocity varies directly with the piston speed and exhausting pressure. More the engine speed more is the exhausting velocity. Exhausting velocity varies directly with the cylinder pressure at the moment when exhaust valve is opened. In order to simulate these facts, following approach was used. There fore analysis was carried out for following inlet exhausting velocities

- **When inlet velocity is equal to V= 10 m/s,(10000 mm/s)**
- **When inlet velocity is equal to V=7 m/s, (7000 mm/s)**
- **When inlet velocity is equal to V=5 m/s, (5000mm/s)**

After applying the boundary conditions, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing fluid flow equations for each element and those formulated governing equations were solved for the velocities, temperatures and pressures [65].

6.9 Results and their physical interpretation

Objectives of this analysis are

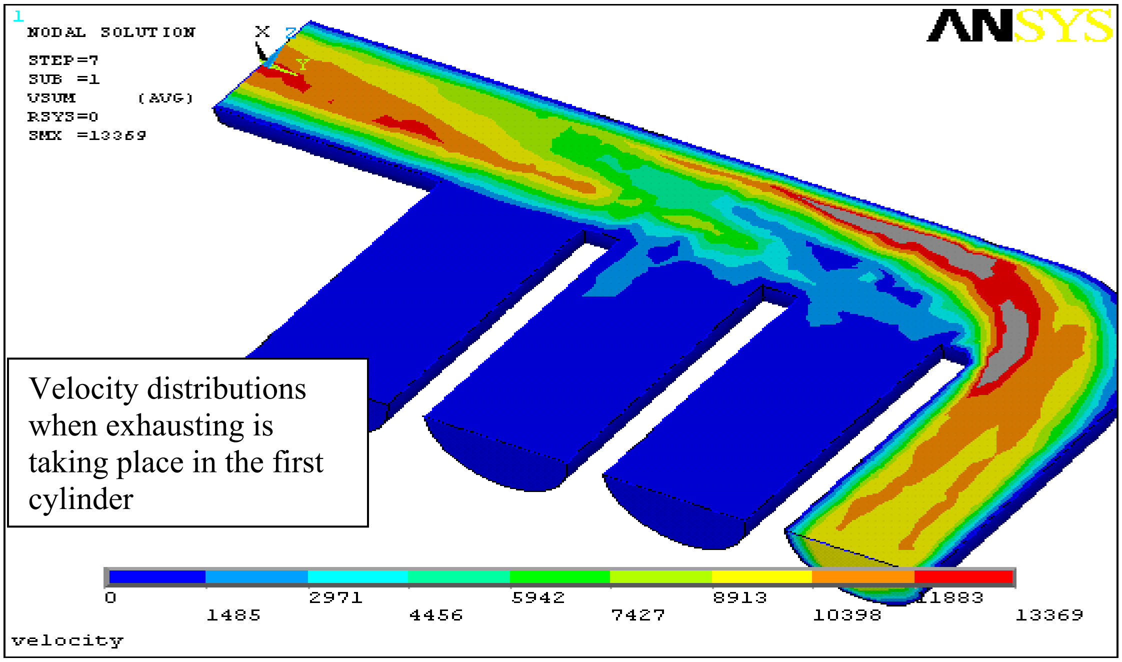
To visualize how combustion products go through the manifold
To observe the flow resistance of the manifold
To observe where it looses its velocity
To plot the temperature distributions in the manifold
To plot the velocity vector contours, this simply indicates the flow tendencies of the combustion products through the manifold
To plot the pressure distributions of the combustion products going through the manifold
To optimize the flow volume (by identifying any unnecessary flow volume in the manifold)

There are few important points, which are to be kept in mind while designing an exhaust manifold .They is

- Flow resistance should be minimum (or flow coefficient should be high)
- geometry should be in such a way that it provides equal exhausting duration for each cylinders so that there is no back flow
- Geometry should be in such a way that there is no back pressure built up.
- Flow geometry must be fully utilized by the flowing combustion products.

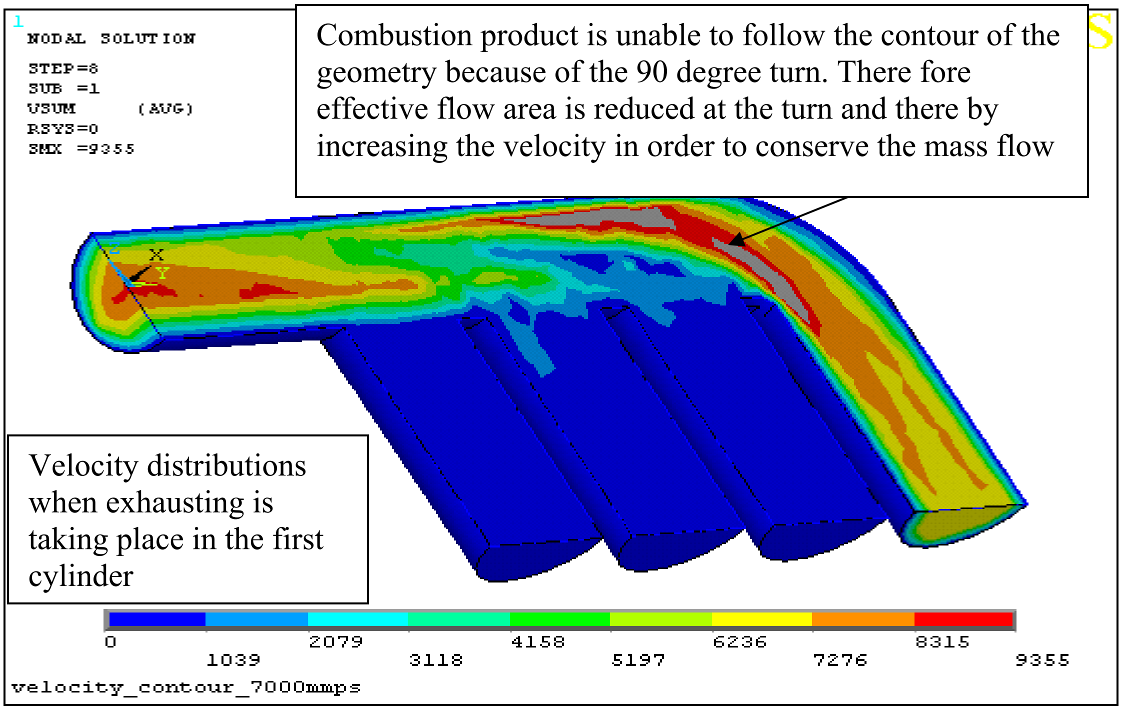
These are the guiding principles to be kept in mind while designing an exhaust manifold. In order to obtain the above mentioned objectives, one should know the pressure, velocity and temperature distributions in the exhaust manifold [49, 65, 54].

The plots that were obtained from the analysis were velocity distributions, temperature distributions, velocity vector plots, and pressure distributions. All the above observations would help the engine designer to design an exhaust manifold with optimized flow geometry, with minimum flow resistance. Analysis results and their physical interpretation are explained below.



Velocity distributions when exhausting is taking place in the first cylinder

Figure 6.7 Velocity distributions in the exhaust manifold when the inlet velocity is 10 m/s



Combustion product is unable to follow the contour of the geometry because of the 90 degree turn. There fore effective flow area is reduced at the turn and there by increasing the velocity in order to conserve the mass flow

Velocity distributions when exhausting is taking place in the first cylinder

Figure 6.8 Velocity distributions in the exhaust manifold when the inlet velocity is 7 m/s

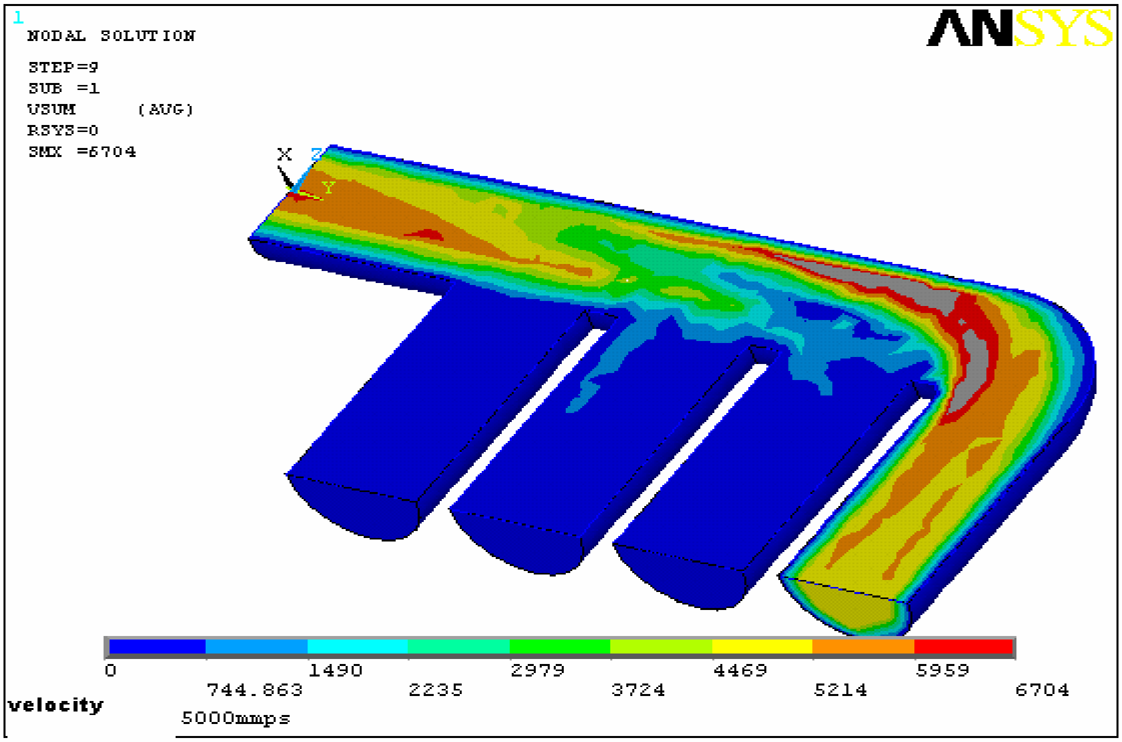
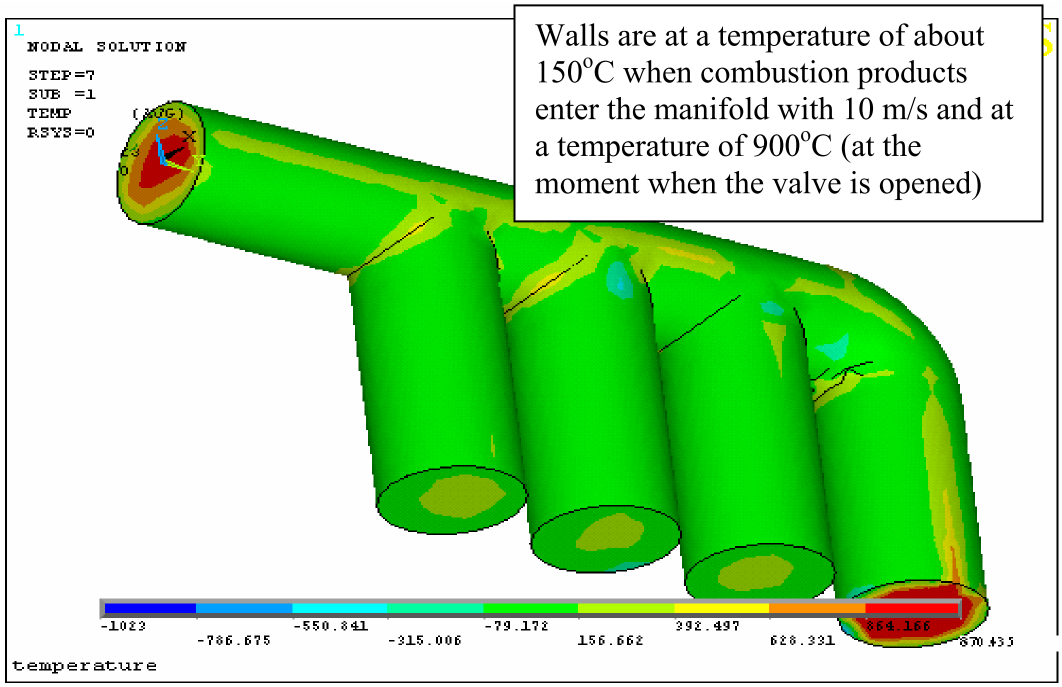


Figure 6.9 Velocity distributions in the exhaust manifold when the inlet velocity is 5 m/s

As can be seen from the above velocity distributions, it is interesting to note that only main core of the manifold pipe is effectively used for the fluid flow. Combustion particles, which are nearer to the walls, are nearly stationary because of **no slip conditions.** It is very clear that boundary layer thickness is appreciable. There is no flow in the other three runners because exhausting is taking place only in the first cylinder. Combustion product is unable to follow the contour of the geometry because of the 90-degree turn [49, 51]. There fore effective flow area is reduced at the bend and there by increasing the velocity in order to conserve the mass flow. There has been some increase in the velocity of the combustion products because effective flow area is reduced because of stationary combustion particles at the walls. There were indications that combustion product tries to enter into the second and third cylinder manifold runners. Though analyses were carried out for different incoming velocities, there is no change in the velocity flow profile.

The grey region indicates that the velocity is more than the maximum velocity shown. It is also clear that velocity is more than the incoming product velocity at few places such as at corner and at most, of the regions velocity is less than the incoming velocity. There fore at this point of time, one important inference can be derived from the showed velocity contours is that this flow geometry has to be modified because of the fact that combustion product has been taking very difficult turn at the 90 degree bend. There fore this angle has to be increased. In order to simulate the exhaust flow according to the firing order, the analysis was carried out on the same manifold, with same boundary conditions except that flow is exhausting is taking place in the third cylinder manifold runner. When exhaust products come out of the engine, it has portion of fuel energy, kinetic energy and pressure energy. Temperature distribution in the manifold is one of the important information to an engine designer. Temperature profiles have been shown below. Temperature profile shows the temperature of combustion particles at each and every point in the manifold.



Walls are at a temperature of about 150°C when combustion products enter the manifold with 10 m/s and at a temperature of 900°C (at the moment when the valve is opened)

Fig 6.10 Temperature distributions in the manifold when the inlet velocity is 10 m/s

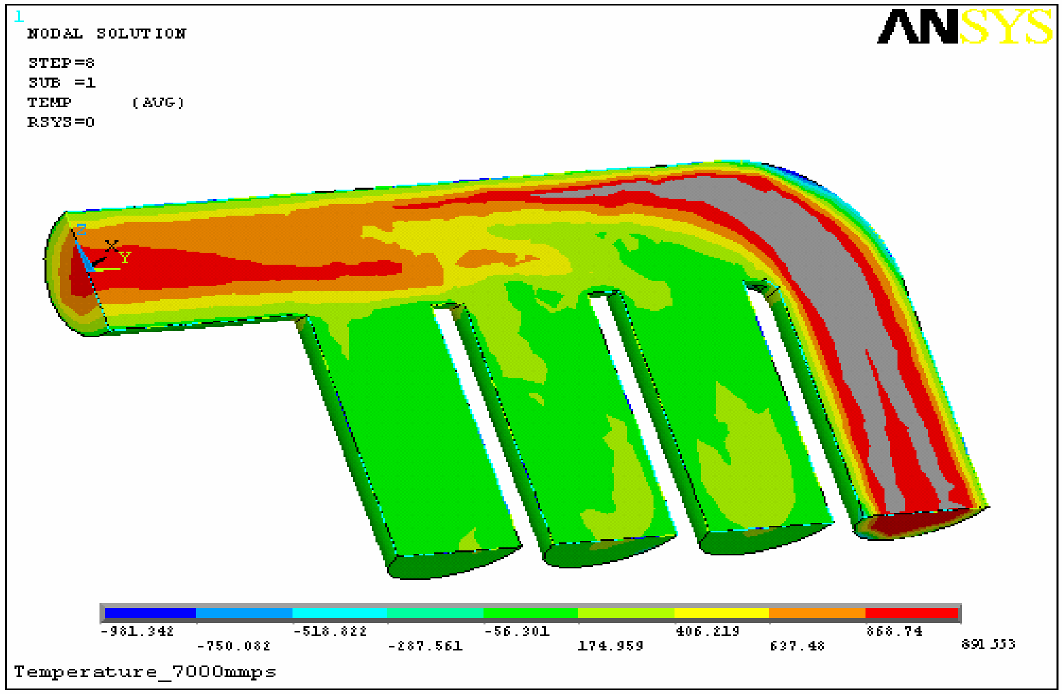


Fig 6.11 Temperature distributions in the manifold when the inlet velocity is 7 m/s

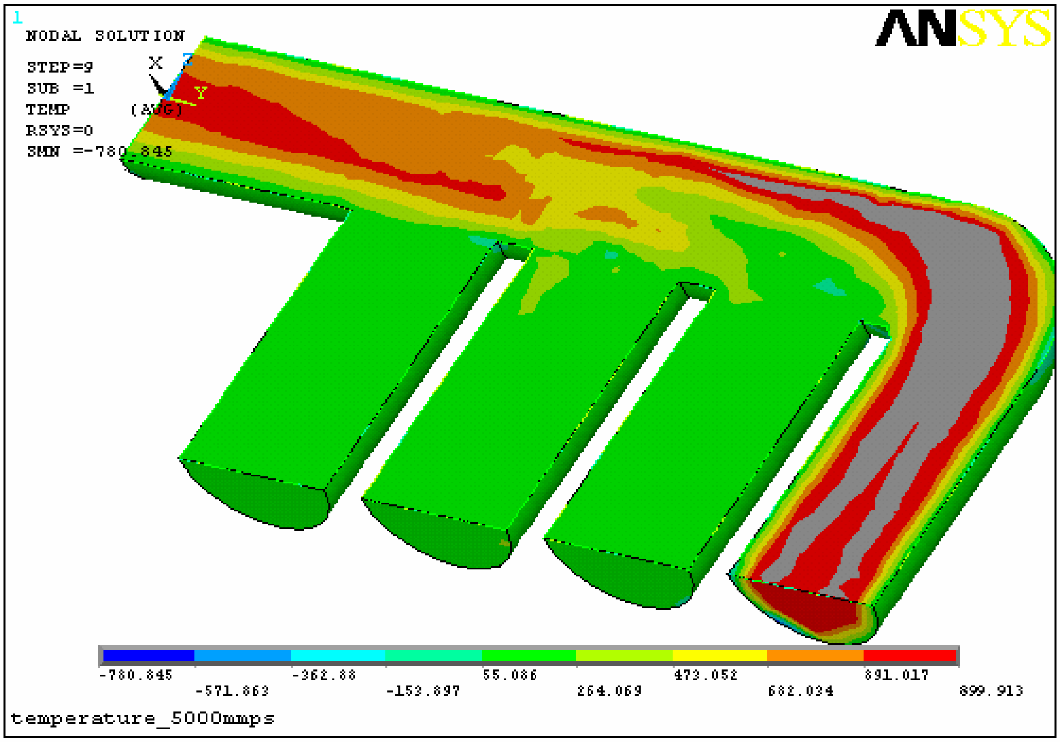


Fig 6.12 Temperature distributions in the manifold when the inlet velocity is 5 m/s

As it is very clear from the above plots, very high temperature is prevailing at the inner core where temperature is about 800°C.The solid walls are at the temperature of about 150 °C and other three manifold runners are at a temperature of about 150 °C. The point of concern is that there should not be more heat loss which would affect the functions of the after treatment devices such as catalytic converters because of the fact these devices function well when the temperature of the products is above the **light off temperature** of hydro carbons (HC) and carbon monoxide (CO).The light of temperature of HC is 280 °C and for CO it is 260 °C [55, 61]. The very next important flow information to an engine designer is pressure distributions in the manifold for different inlet velocities are shown below. Pressure profile is a measure of flow resistance of the manifold geometry.

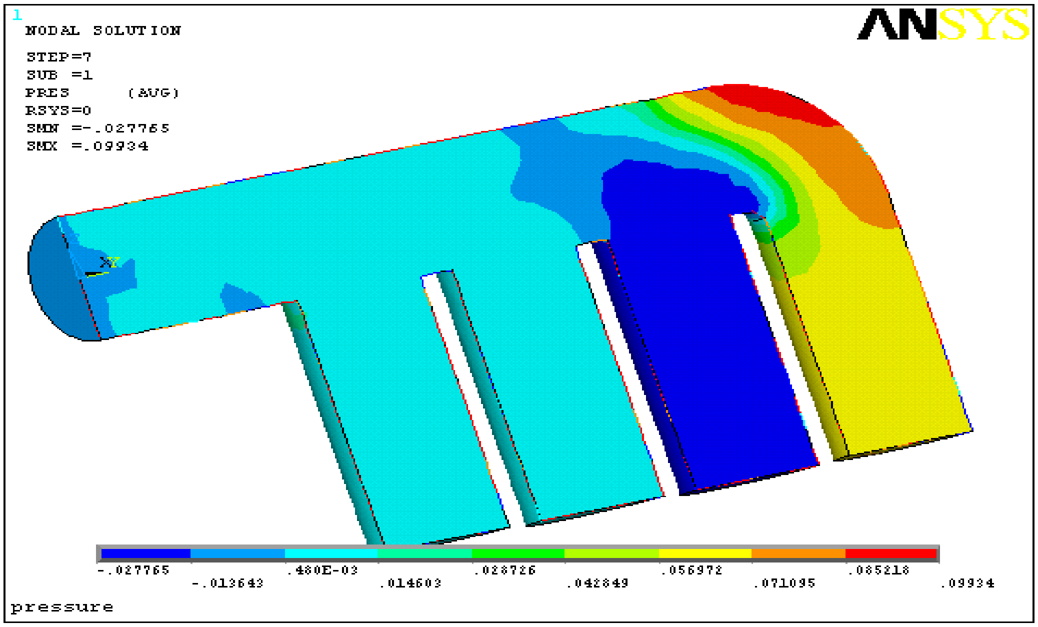
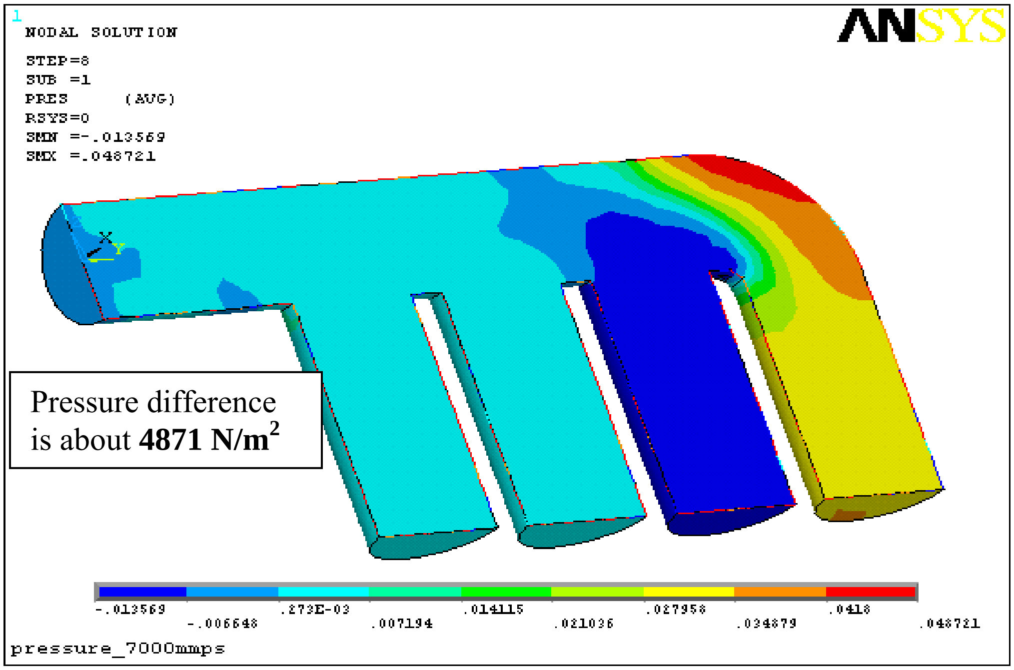
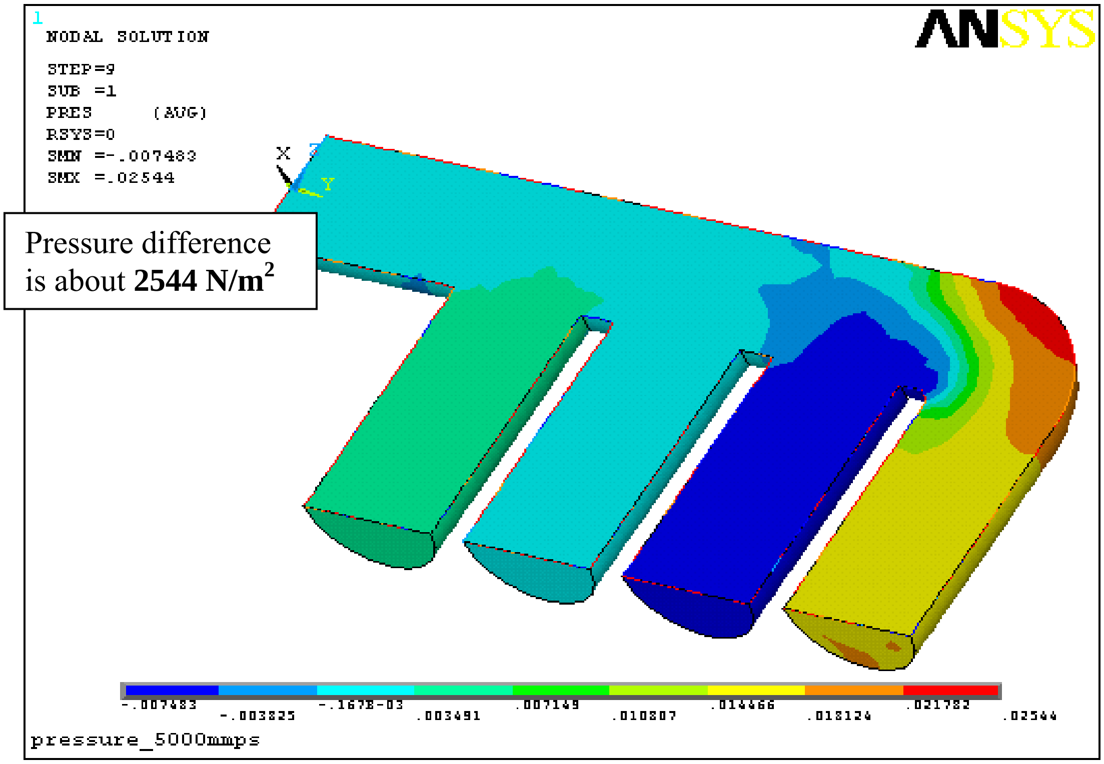


Fig 6.13 Pressure distributions in the manifold when exhausting velocity is 10 m/s



Pressure difference is about **4871 N/m^2**

Fig 6.14 Relative pressure distributions in the manifold when exhausting velocity is 7 m/s



Pressure difference is about **2544 N/m^2**

Fig 6.15 Relative pressure distributions in the manifold when exhausting velocity is 5 m/s

As can be seen from the above plots, more pressure force is required to increase the velocity of the flowing fluid (i.e. pressure differential increase with increase in inlet velocity).There has been a pressure built up at the point where fluid takes 90 degree bend. The pressure built up is due to direct hit on the solid walls as fluid enters the manifold. Since combustion product does not have any tendency to flow into the second cylinder runner, low pressure is prevailing there, as flow volume is empty. In order to simulate the firing order of the engine, combustion products made to flow through the third cylinder runner for which boundary conditions were same. As like the first one, velocity, pressure and temperature profiles have obtained and their physical interpretations have been given below.

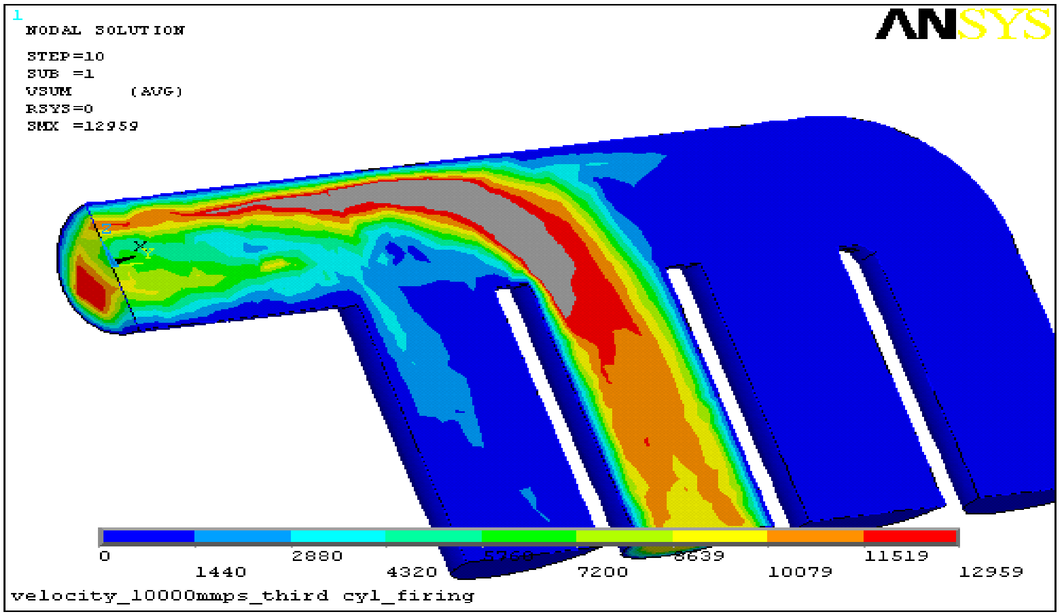


Figure 6.16 Velocity distributions in the exhaust manifold when exhausting is taking place in the third cylinder

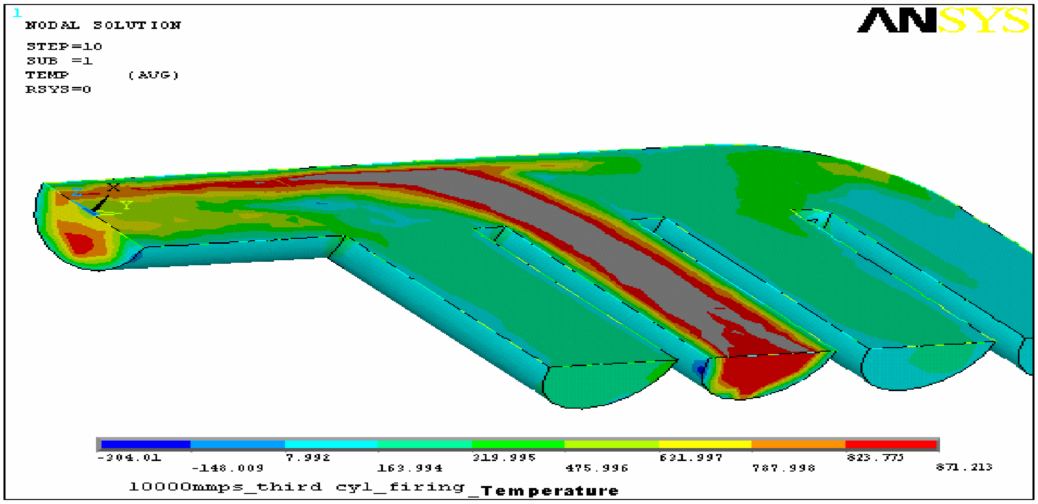
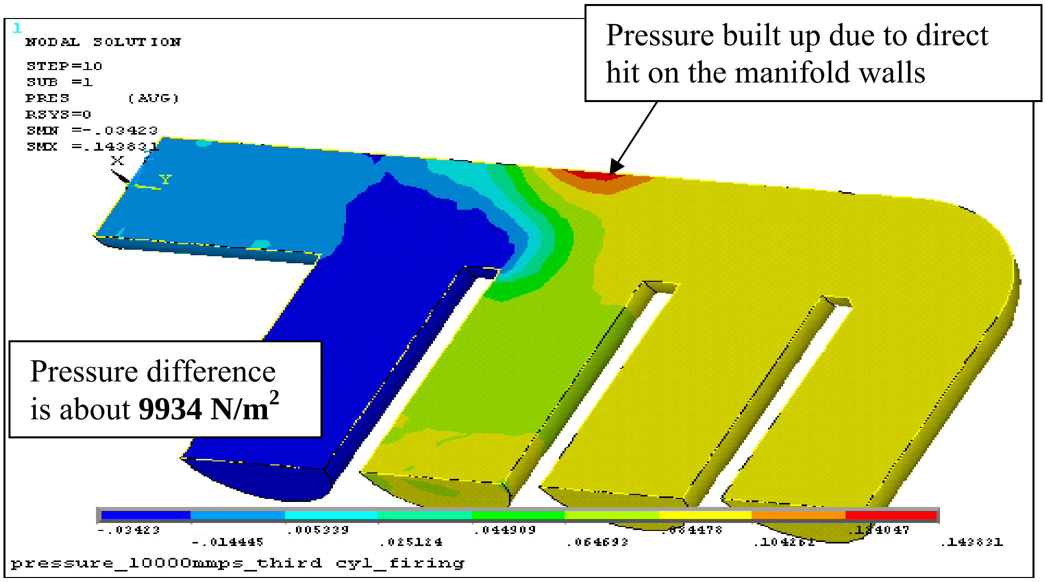


Figure 6.17 Temperature distributions in the exhaust manifold when exhausting is taking place in the third cylinder.

Velocity profile shows that flow is taking place in the third cylinder runner. Fluid is taking very hard turn at the bend. Maximum velocity reached is 12.9 m/s.Temperature profile shows that only small portion in the manifold is very hot and other regions are at very low temperature. The pressure distributions in the manifold when exhausting is taking place in the third cylinder runner is shown below.



Pressure built up due to direct hit on the manifold walls

Pressure difference is about **9934 N/m^2**

Fig 6.18 pressure distributions in the manifold when exhausting is taking place in the third cylinder

It is interesting to note from the above plot that low pressure is created in the fourth manifold runner when flow is taking place in the third runner [49,61]. This may be because of the fact that flow volume is empty and also because vacuum creation due to flow in the downstream.

6.10 Summary

- Manifold is a pipe which carries the burned gases out of the engine firing cylinders
- Veloity,pressure and temperature distributions are matter of concern
- Manifold was simulated for typical flow velocity of 10 m/s,7 m/s and 5 m/s (the purpose is to simulate the different engine speeds)
- Solid models were utilized to carry out CFD analysis
- Velocity vector plot helps us to understand ‘how combustion products flow through the manifold’ and **fluid flow tendencies** at varies points along the downstream.
- Pressure difference (pressure difference between the inlet and outlet sections) is varying directly with the flow entering velocity.
- Geometry is not good enough and it has to be optimized based observed facts, as fluid takes hard turning at the bend it is unable to utilize the full geometry.

CHAPTER - 7

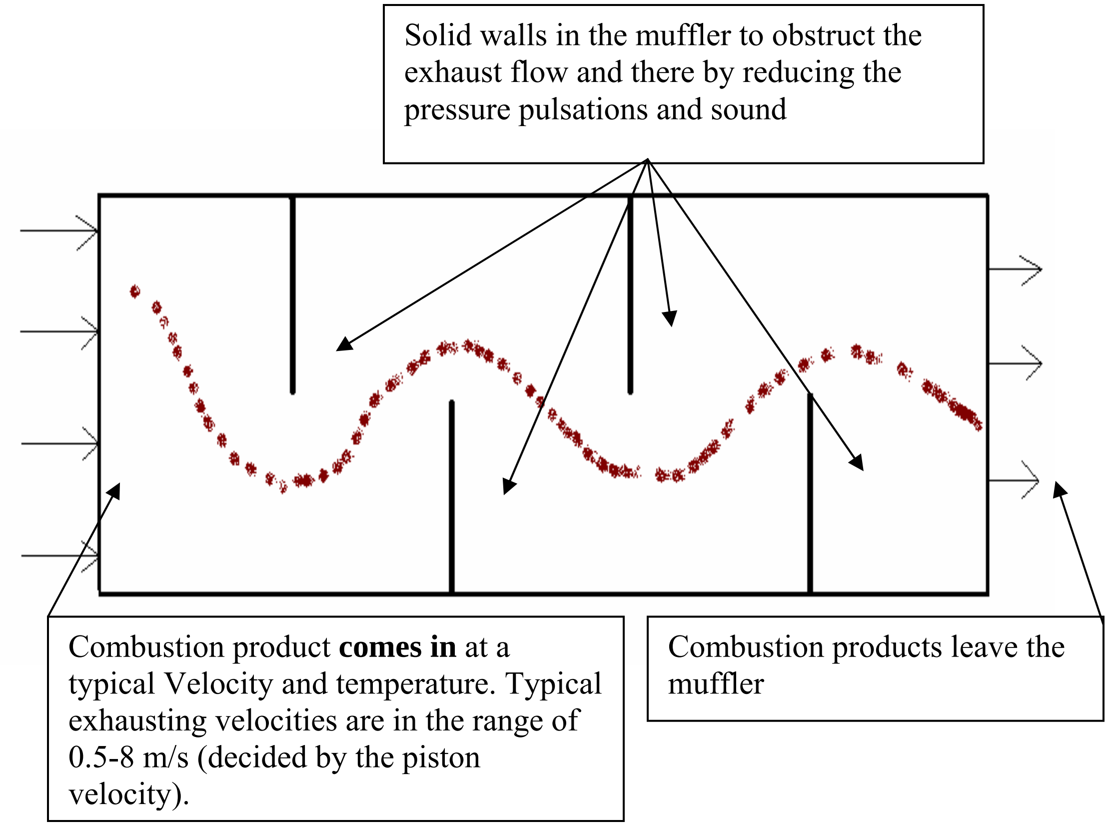
Computational Fluid Dynamics (CFD) analysis Of Exhaust Muffler

Muffler is a fluid flow device, which is used in internal combustion engine to reduce the pressure and sound of the exhaust flow products by passing the exhaust products through so many flow obstacles, which reduces the pressure pulsations, and sound energy of the exhaust products. If muffler were not in place, then there would be a strong pressure pulsation, which induces the sound. In one way, muffler acts as a damper for pressure and sound. This pressure pulsation is as obvious as exhaust pressure is little greater than the environmental pressure [49, 54]. The sound energy of the exhaust products is reduced by providing obstacles, which the combustion products have to overcome there by loosing its energy. There fore velocity and pressure of the combustion products are matters of importance as they strongly affect the pressure pulsations. These velocity and pressure profiles are, normally, obtained by carrying out the CFD analysis. Hence, CFD analysis was carried out and velocity and pressure profiles are obtained.

Figure 7.1 Exhaust mufflers

7.1 Statement of the physical Problem

The physical problem is shown below. Muffler is a fluid flow device. The combustion products flow through the muffler. Muffler is a very complicated geometry as it has number baffle plates to obstruct the combustion products and there by removing the pressure and sound energy. When combustion products passes through this complex geometry, sound and pressure energies are dissipated there by letting the combustion product with no pressure pulsations. The physical problem in hand is very complex as two physical phenomenons' (sound and flow energies) are involved in it. The typical velocity of the combustion products at the inlet is in the range of 0.5-8 m/s, which is decided by the piston velocity and exhausting pressure [79, 80].



Solid walls in the muffler to obstruct the exhaust flow and there by reducing the pressure pulsations and sound

Combustion product **comes in** at a typical Velocity and temperature. Typical exhausting velocities are in the range of 0.5-8 m/s (decided by the piston velocity).

Combustion products leave the muffler

Figure 7.2 Statement of the physical problem

Now that the objective is to visualize the flow through the muffler, to observe that how the combustion product suffers while flowing through muffler.

7.2 Objectives of this analysis [49,57]

- To plot the velocity distributions of the combustion products going through the muffler
- To plot the pressure distributions of the combustion products going through the muffler
- To visualize the flow of combustion products through the muffler
- To check out the flow volume (by identifying any unnecessary flow volume in the muffler)

7.3 Inputs required carrying out the CFD analysis [67, 79, 80]

- Approximate geometry of the muffler
- Flow properties of combustion products such as
    - i.Density of the combustion products (ρ)= 1.23 kg/m^3
    - ii.Viscosity of the combustion products (n)=2e-5N-s/m^2
    - iii.Velocity with which combustion products enter into the muffler (V)
    - iv.Temperature of the combustion products (T)

7.4 Typical Procedure to carry out this analysis [65]

- Geometry creation (solid modeling of muffler)
- Dividing the muffler solid flow volume (solid model) into number of elements and then connecting those elements with each other (meshing) and thereby obtaining the finite element model.
- Applying boundary conditions (loading-velocity, pressure loading)
- Solving the problem
- Results and their interpretation.

7.5 Geometry creation and meshing

Geometry creation simply means that the creation of solid model of the muffler. Solid model of the muffler was created in ANSYS solid modeling environment. In a typical CFD analysis, solid model represents the flow volume not the pipe through which fluid is flowing .Solid model is the base for carrying out any FEA or CFD analysis. Solid model of the exhaust muffler that has been created in ANSYS solid modeling environment is shown in figure 6.3 Solid model which is to be used in any CFD analysis can be very simple and at the same time it reflects the physical reality of the muffler. Therefore, small small features like rounds, chamfers have not been modeled. As shown in the figure 6.3, solid model is very simple but at the same time gives a physical reality. Solid model can be turned into to FEM model by dividing the flow volume (solid model) into number of small small elements. This process is called "**discritization**".Discritization is the process of dividing the solid model into finite number of elements. After dividing the solid model into finite number of elements, they have to be connected to each other as the flow volume is continuous and physically connected at each material point.

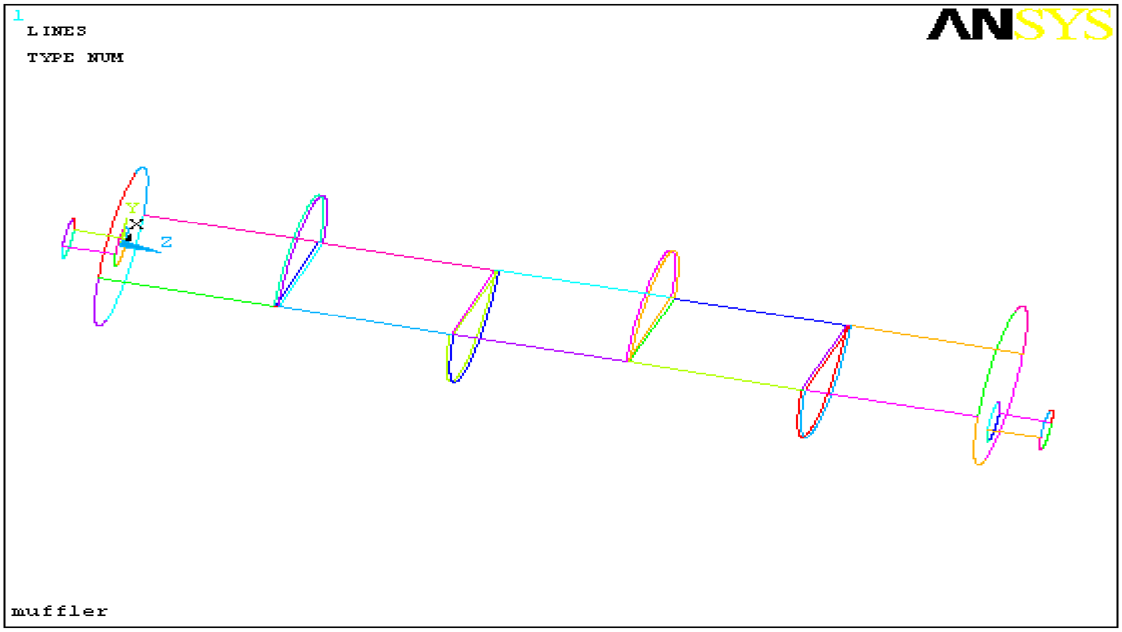


Figure 7.3 Wire frame model of plot of the exhaust muffler

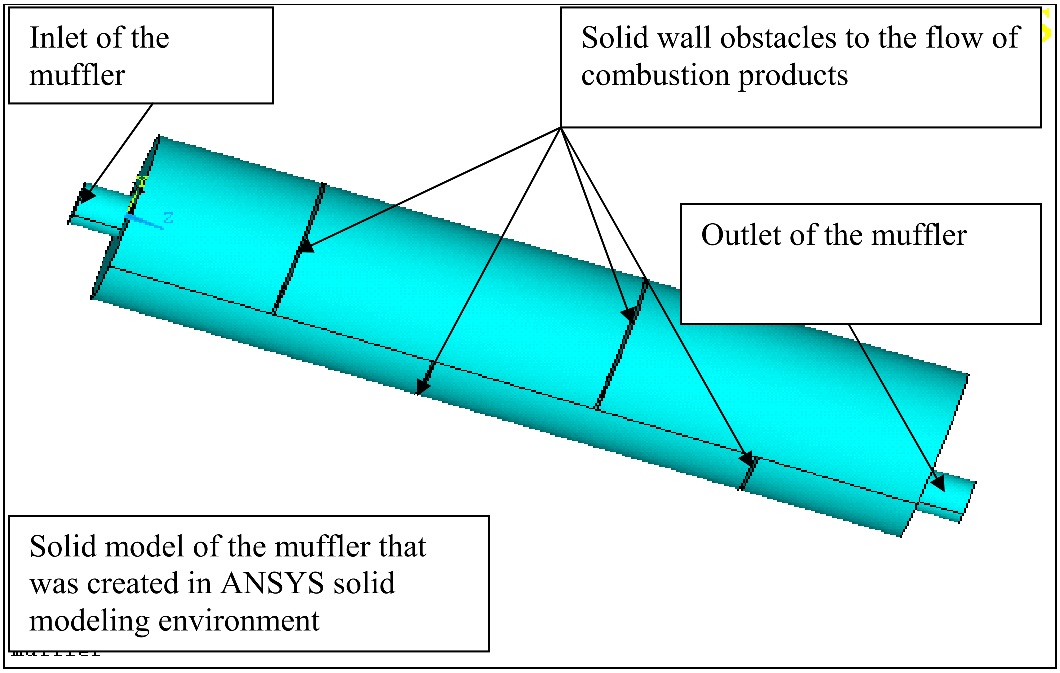


Figure 7.4 Solid model of the muffler created in ANSYS solid modeling environment

In simple terms meshing means that connecting the elements with each other. An element is the building block of the finite element model. Therefore, the type of element (linear element or higher order elements), number of nodes and their capabilities are important parameters for selecting the elements for particular analysis. In this analysis, the element used was 3D **FLOTRAN 142** which is a 3D element used in 3D CFD analysis.

FLUID142 Geometry

Fluid element that was for the analysis is shown below.

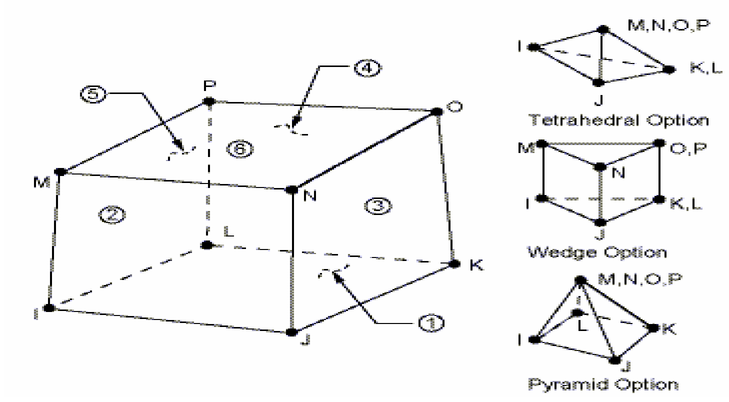
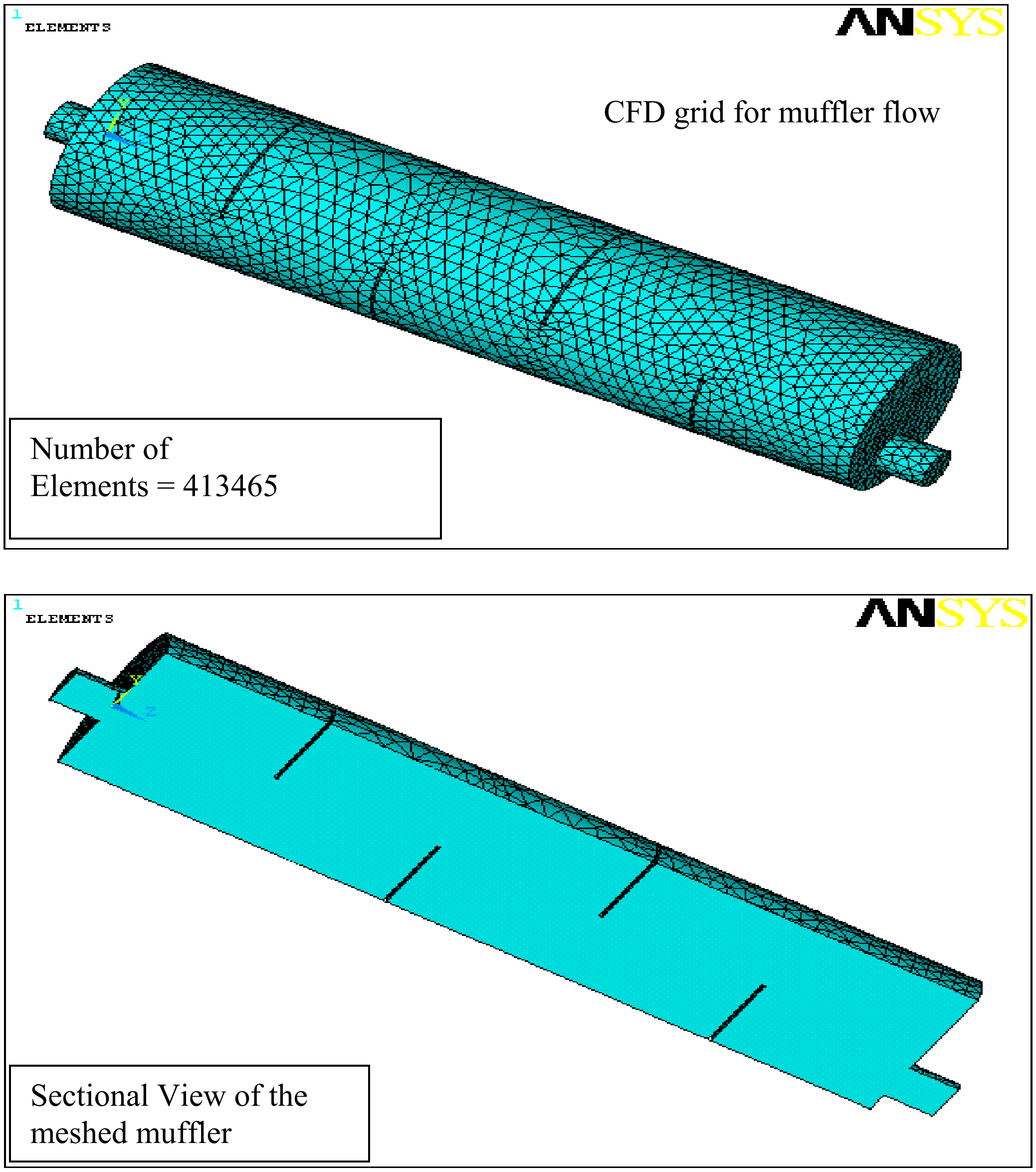


Figure 7.5 Element name – 3D FLOTRAN 142

Meshed model of the muffler is shown below.



CFD grid for muffler flow

Number of
Elements = 413465

Sectional View of the
meshed muffler

Figure 7.6 Meshed models of the muffler

Meshing is the important step in CFD analysis. Mesh must be adequate enough to obtain the accurate results while keeping the computational time in mind. More the number of elements, better the accuracy on the other hand more the number of elements more the computation time. The number of elements used in this analysis was 413465(four hundred thousands).

7.6 Physical boundary conditions (loading)

The inlet area of the muffler receives the combustion products and outlet area of the muffler lets the combustion products to leave the muffler. All the outer surfaces and front faces except the inlet and outlet areas are specified as boundary walls and atmospheric pressure was applied on the outlet area [67, 79, 80, 57].

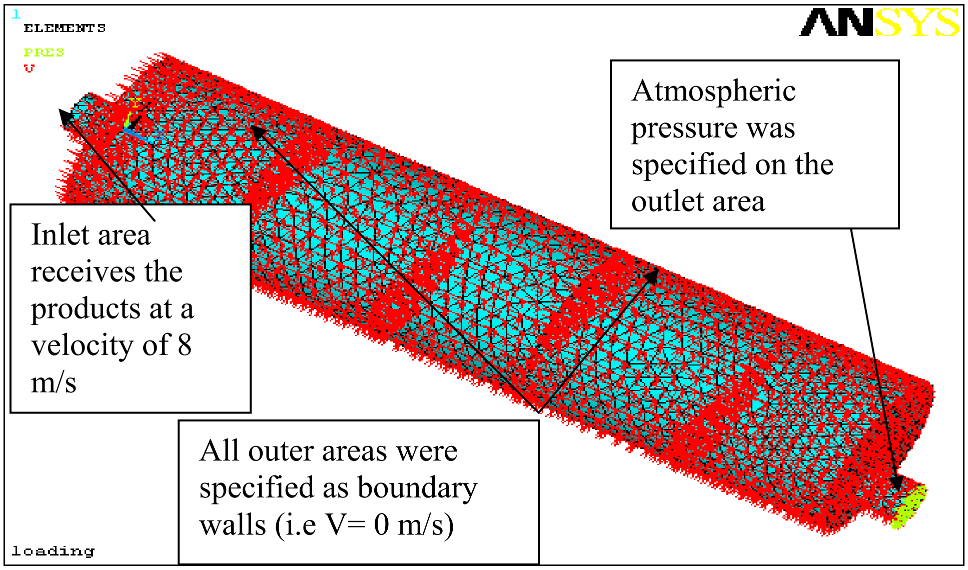


Figure 7.7 Physical boundary conditions on the muffler

After applying the boundary, the problem was solved by the ANSYS Solver. ANSYS solver formulates the governing fluid equations for each element and those formulated governing equations were solved for the pressures and velocities.

7.7 Results and their physical interpretation

Objectives of this analysis are

To visualize how combustion products go through the muffler

To observe the flow resistance of the muffler

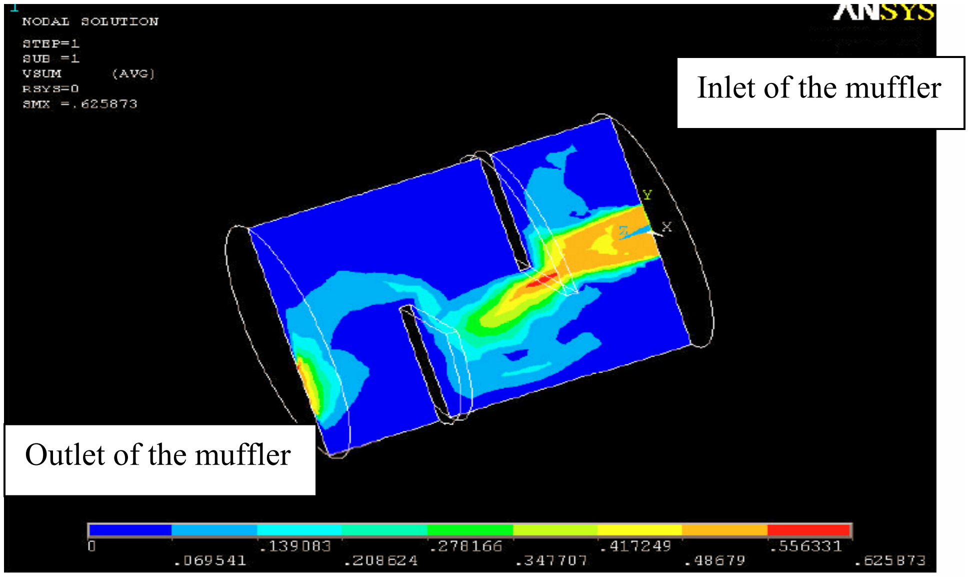
To observe where it looses its velocity

To plot the velocity vector contours, this simply indicates the flow tendencies of the combustion products through the muffler

To plot the pressure profile of the combustion products going through the muffler

To check out the flow volume (to find out any unnecessary flow volume in the muffler)

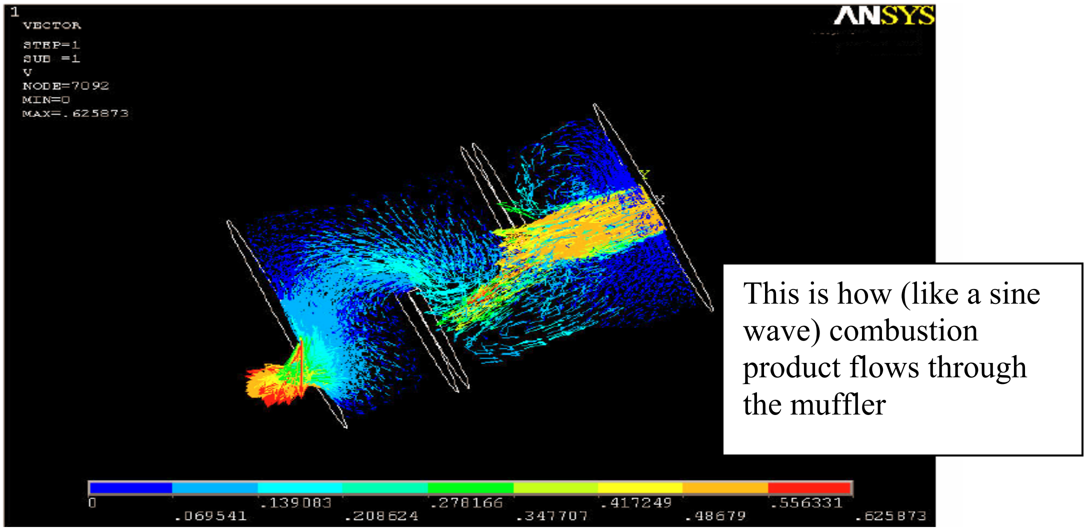
All the above observation would help the engine designer to design an exhaust muffler with optimized flow geometry, good flow resistance that is the prime function of the muffler. The plots that were obtained from the analysis were velocity contours, velocity vector plots, and pressure plots [57]. Two kinds of muffler geometry were used for the analysis. One is with two obstacles and other one is with four walls obstacles.



Inlet of the muffler

Outlet of the muffler

Figure 7.8 Velocity distributions of the combustion products at a particular section in a muffler with two walls



This is how (like a sine wave) combustion product flows through the muffler

Figure 7.9 velocity vectors of the combustion products at a particular section in a muffler with two walls

As seen from the above vector plots, it is very interesting to note that the first solid wall, directly impacts combustion products come into the muffler, and then moves down; again, it encounters the next obstacle and takes an upward flow and then combustion products were forced to flow towards outlet by the circumference walls [57]. This is how combustion products flow through the muffler.

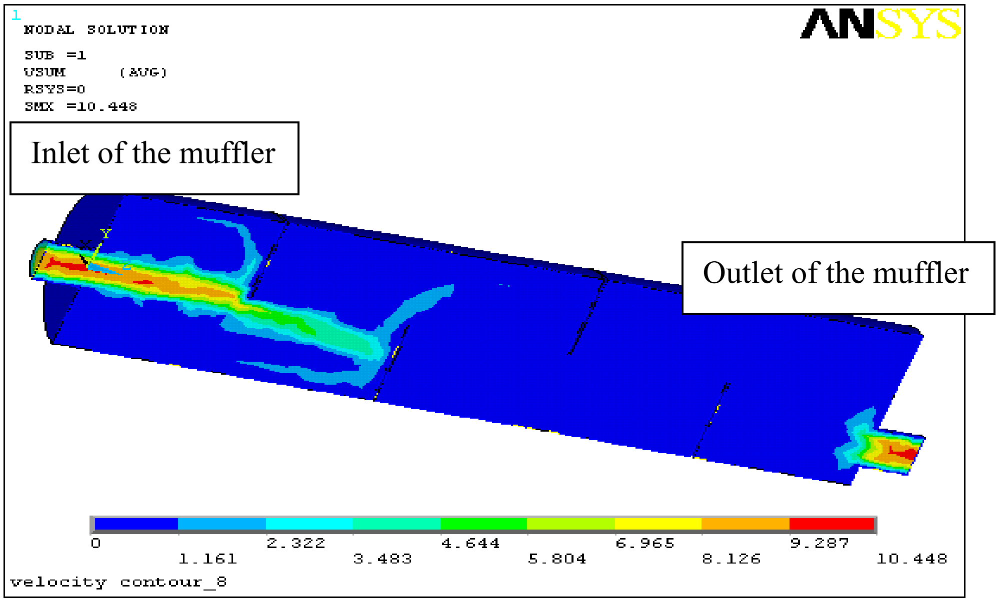


Figure 7.10 Velocity distributions of the combustion products at a middle section in a muffler with four solid walls

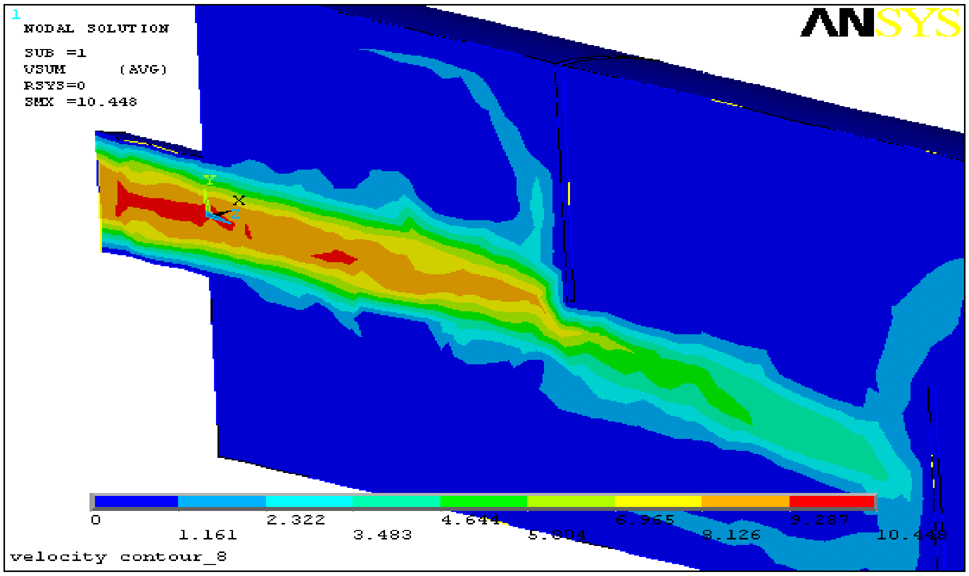


Figure 7.11 Close-up-view of velocity distribution at the inlet portion of the muffler

As it is very clear from the above figure that combustion products flow through a very small portion of the muffler and that combustion products are stagnant in most of the regions [57]. It is worth to note that velocity of the combustion products reduces very rapidly from incoming velocity of 8 m/s to roughly about 2 m/s.

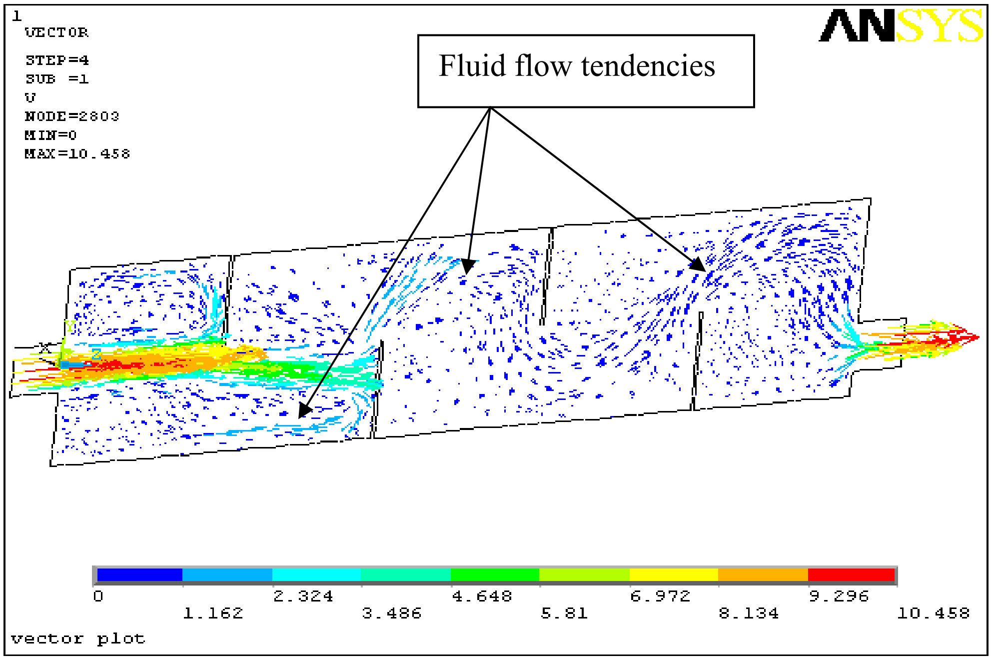


Figure 7.12 Velocity vector plot (flow tendency)

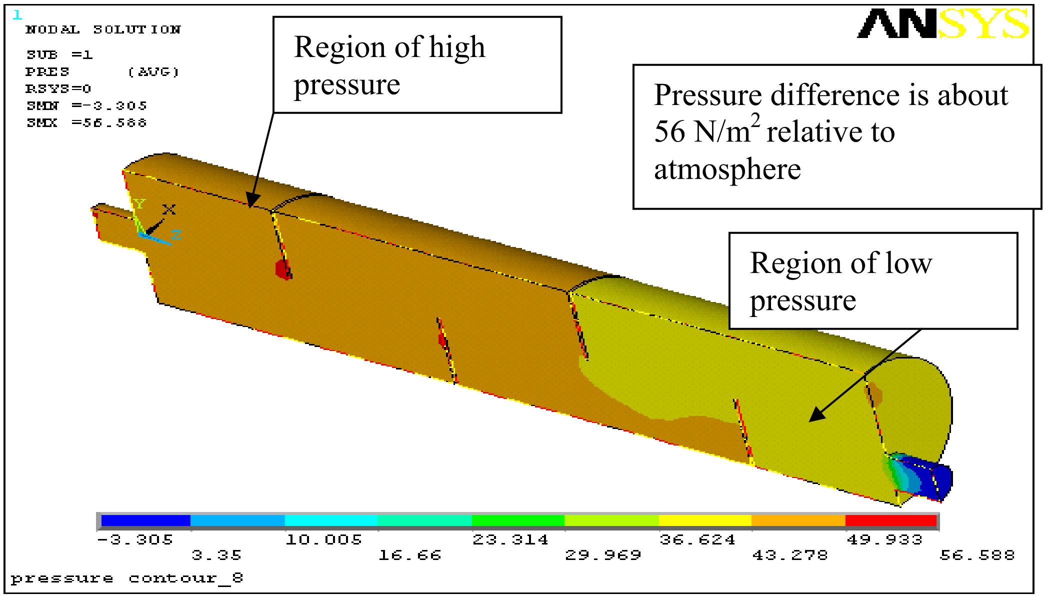


Figure 7.13 Relative pressure (with respect to atmosphere) distributions of the combustion products.

As seen from the above results, combustion product stays nearly stagnant in most of the regions. Velocity of the combustion products were drastically reduced by the solid obstacles in the flow.

7.8 Summary

- Muffler is a fluid flow device, one of the best example of “ Flow resistance devices”
- The main function of muffler is to reduce the pressure pulsations of the combustion products
- Muffler was simulated for a typical flow velocity of 8 m/s
- Solid models were utilized to carry out CFD analysis
- Velocity of the combustion products was reduced very rapidly by the solid obstacles in the muffler.
- Combustion products were nearly stagnant in most of the regions of the muffler
- At some regions in the muffler, velocity is more than the incoming flow velocity because of the fact that flow has to satisfy the mass conservation because effective flow area reduces due to boundary layers and no slip conditions.
- Velocity vector plot helps us to understand ‘how combustion products flow through the muffler’ and fluid flow tendencies at varies points along the downstream.
- Pressure difference (pressure difference between the inlet and outlet sections) was about 56 N/m^2 relative to the atmospheric conditions.

Conclusions

- Internal combustion engines have been one of the active areas of research for many academic researches since the inception.
- Almost all of the components in an internal combustion engine are subjected to heat and structural loads.
- The currant study emphasis on stress, strain, temperature, heat flux, thermal gradient, velocity and pressure distributions in the component materials.
- The study was carried out using the Finite Element Methods Approach
- The type of study is **peak moment simulation** which means that only the conditions prevailing at the point of combustion are simulated
- *Finite Element Analysis* is a way to simulate loading conditions on a design and determine the design's response to those conditions.
- Structural analysis is used to determine deformations, strains, stresses, and reaction forces
- Thermal analysis is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux
- CFD is used to determine the flow distributions and temperatures in a fluid
- All the analyses were carried out for different boundary conditions, different geometries and different materials properties.
- The peak surface temperature of the piston material when there is no cooling is about 1980^{o}C against 518 ^{o}C when cooling was provided in a aluminium piston.
- Peak stress in the piston due to combustion pressure= 118 N/mm^2
- The average temperature of the upper portion of the engine block when there is no sort of cooling mechanism was about 1400 ^{o}C against the 750 ^{o}C when it enjoys water and oil and, atmospheric cooling in a cast iron block.
- Peak stress in the engine block for a gas pressure of 120 bars = 118 N/mm^2

- Connecting rod was simulated for different gas loads as rod is subjected to different gas forces depending upon load on the engine.
- Maximum stresses induced in the connecting rod were about 140, 104, and 90 **N/mm^2** for the gas loads of 120, 70, and 60 bars respectively, which are less than 50 % strength of the of the **steel alloy-1040** taken for the study.
- Exhaust Manifold was simulated for typical flow velocity of 10 m/s,7 m/s and 5 m/s (the purpose is to simulate the different engine speeds)
- Velocity vector plot helps us to understand 'how combustion products flow through the manifold and fluid flow tendencies at varies points along the downstream.
- Geometry of the exhaust manifold can be optimized by observing that how combustion products flow through it.
- Muffler was simulated for a typical flow velocity of 8 m/s
- Velocity of the combustion products was reduced very rapidly by the solid obstacles in the muffler.
- Combustion products were nearly stagnant in most of the regions of the muffler

Scope for future work

Since analyses carried were of "peak moment simulation" type, only peak working conditions were simulated. But internal combustion engines components are subjected varying heat and pressure loads as pressure and temperature in the cylinder varies through out the cycle. Further that most of the components are subjected to fatigue loadings. Therefore peak moment simulation certainly not adequate enough to predict the real working conditions of the engine components. For instance, dynamic and transient heat transfer analysis can be performed on piston and engine block to predict the real working conditions of the these components. Buckling and fatigue analysis can be performed on connecting rod. Similarly in the exhaust manifold and exhaust muffler are subjected continuous flow and thermal loads. Hence transient heat and turbulent flow analysis can be conducted on these combustion product flow devices. This FEM study can be extended to engine valves, heads, bearing analysis, and fuel injection systems etc.

REFERENCES

1. Ganesan V, “Internal combustion Engines”, 2nd Ed., Tata McGraw-Hill, New Delhi, 2004.
2. Yunus A.Cengel, “Heat Transfer-A practical Approach”, 2nd Ed., Tata McGraw-Hill, New Delhi, 2003.
3. Hey Wood, J. “Internal Combustion Fundamentals” New York, 1988.
4. Ferguson, C.R. and Kirkpatrick, A.T. “ Internal combustion Engines-Applied Thermo sciences”, 2nd Ed., John Wiley and Sons(Asia) Pte.Ltd, Singapore, 2004.
5. Callister, W.D. “Materials science and engineering –An introduction,” 6th Ed., John Wiley and Sons (Asia) Pte.Ltd, Singapore, 2004.
6. Taylor, C., The Internal Combustion Engine in Theory and Practise, Vol.2, MIT Press, Cambridge, Massachusetts, 1985.
7. Obert, E., The Internal Combustion Engine, International Text book Co., Scranton, Pennsylvania, 1950.
8. Mathur, Sharma, “Internal combustion engines,”, Dhanbat Rai Publications,New Delhi,2003.
9. Reynolds, W, “A Computer Program for calculating Properties of Equilibrium Combustion Products with Some Applications to I.C Engines,” SAE paper 750468.
10. Budynas R.G, “Advanced strength and stress analysis” 2nd ed., McGraw-Hill, 1999.
11. Zienkiewicz, O.C., and R.L Taylor. The Finite Element Method, 4th ed., New York: McGraw-Hill, 1991.
12. Cook R.D., D.S.Malkus; and M.E.Plesha. Concepts and Applications of Finite Element Analysis, 3rd ed., New York: Wiley, 1989.
13. Bathe, K.J. Finite Element Procedures. Englewood Cliffs, NJ: Prentice Hall, 1996
14. Desai, S., F.Abel, “Introduction to Finite Element Analysis,” Prentice Hall, 1995
15. Saeed Moaveni, “The Finite Element Analysis-Theory and Application with ANSYS,” Prentice-Hall, Englewood Cliffs, NJ, 1999.
16. R.H. Ghallaghar, Finite Element Analysis: Fundamentals, Prentice-Hall, Englewood Cliffs, NJ, 1975
17. Lancaster, D.R., R.B Krigher, and J.H Lienesch, “Measurement and Analysis of Engine Pressure Data,” SAE paper 971013, 1975.
18. Bishop, I., “Effect of Design Variables on Friction and Economy,” SAE paper 640807, 1964.
19. Yagi, S., K.Fujiwara, N.Kuroki, and Y.Maeda, “Estimate of Total engine Loss and Engine Output in Four Stroke S.I Engines,” SAE paper 910347.
20. Thomas Schuszter, “Engine Design using State-of-the-art Tools,” Publication journal, MAN B&W.
21. AVL, “Simulation News”, Advanced Simulation Trend letter 2003, AVL.
22. Sagakuchi, K., “The Mitsubishi UEC Engine Development Program, ISME Tokya, 2000.
23. Sagakuchi, K., Y.Hesegawa “Development of New Series in Mitsubishi UE Large 2 – Bore Cycle Diesel Engine”, ISME Tokya, 2000.

24. Mike Bomfard, and Y.Lin “3D Piston Secondary Motion”, Ford Motor Company, Advanced Power Train Divison, Brett Harris.
25. Kolbanschimdt., Piston Technology for Passenger Cars”, Published Journal Kolbanschimdt Company, Neckersulm, 2004.
26. Shibata T., and TSubouchi, “Evaluation of CFD tools Applied to Engine Coolant Flow Analysis”, Technical Review, Mitshibushi Motors, 2004.
27. Assanis, N., and Min Chun. “Measurements and Predictions of Steady State and Transient Stress Distributions in a Diesel Engine Cylinder block, SAE paper 010973.
28. Fitzgeorge, D. and Pope, J. A. “An Investigation of the Factors Contributing to the Failure of Diesel Engine Pistons and Cylinder Covers,” Trans. N.E.Coast Institute of Engine. And Shipbuilders, 71, pp.163-236, 1955.
29. Shalev, M., Zvirin, Y. and Scotter, A., “Experimental and Analytical Investigation of the Heat Transfer and Thermal Stress in a Cylinder block of a Diesel Engine,” Int. J. Mech. Sci., Vol.25, No. 7, pp. 471-483, 1983.
30. Blech, J. J.,”Heat Transfer and Stress Analysis of Engine Heads and Evaluation of Methods of Preventing Head Cracks,” SAE Paper 820504, 1982.
31. Bertodo, R. and Carter, T. J.,”Stress Analysis of Diesel Engine Cylinder Heads,” J. of Strain Analysis, 6, 1, pp.1-12, 1971.
32. Garro, A. and Vullo, V., “Some Consideration on the Evaluation of Thermal Stresses in Combustion Engine,” SAE Paper 780664, 1978.
33. Nozue, Y., Satoh, H. and Umetani, S., “Thermal Stress and Strength Prediction of Diesel Engine Cylinder Head,” SAE Paper 830148, 1983.
34. Groeneweg, M. A., Ahuja, R., Pfeiffer, R. F. and Bezue, T. N., “Current Applications of Finite Element Analysis to Diesel Engine Component Design,” SAE Paper 870814, 1987.
35. Rosenberg, R.C, “General Friction Considerations for Engine Design,” SAE paper 821576.
36. Annand, W.J.D, “Heat Transfer in the cylinders of Reciprocating Internal Combustion Engines,” Proc.Inst.Mech.Engrs., 1977, p.973, 1963.
37. Bendersky, D. “A special Thermocouple for Measuring Transient Temperature,” Mech., 75, p.117, 1953.
38. Bohac, S., DE.Baker, and D.Assanis, “A Global Model for Steady State and Transient SI. Engine Heat Transfer Studies,”SAE paper 960073, 1996.
39. Dent, J.C. and S.J Sulaiman, “Convective and Radiative Heat Transfer in a High Swirl Direct Injection Diesel Engines,” SAE paper 770407.
40. Furuhama, S. and Y.Tateishi, “Gases in Piston Top- Land Space of Gasoline engine,” Trans.soc.Automotive Eng.of Japan, 4, p.30-39, 1992.
41. Woshini,G, “ A Universally Applicable Equation for the Insaneness Heat Transfer Co-efficient in the Internal Combustion Engines,” SAE paper 670931,1967.
42. Rado, G. E. and Seaberg, W. G., “Cylinder block Design Utilizing Rapid Development Tools,” Paper No. 98-ICE-124, ICE-Vol. 31-1, 1998 Fall Technical
43. Conference, ASME, 1998.
44. Keribar, R. and Morel, T., “Thermal Shock Calculations in I.C. Engines,” SAE Paper 870162, 1987.
45. Spalding, D. B. and Afgan, N. H., Heat & Mass Transfer in Gasoline and Diesel Engines, Hemisphere Publishing Co., 1989.

46. Lee, K. S., "Study on Heat Transfer and Thermal Behavior Characteristics of Naturally Aspirated Diesel Engine," Ph.D. Thesis, Yonsei University, Seoul, Korea, 1997.
47. Assanis, D. N. and Heywood, J. B., "Development and Use of a Computer Simulation of the Turbo compounded Diesel System for Engine Performance and Component Heat Transfer Studies," SAE Paper 860329, 1986.
48. Annand, W. J. D, "Experiments on a Model Simulating Heat Transfer between the Inlet Valve of a Reciprocating Engine and the Entering Stream," Int. Mech. Engr., Thermodynamics and Fluid Mechanics Convection, Bristol, 1968.
49. Hires, S. D. and Pochmara, G. L., "An Analytical Study of Exhaust Gas Heat Loss in an Engine Exhaust Port," SAE Trans., Vol. 85, 1976.
50. Kim, B. T., "Thermal and Mechanical Behavior Analysis of a Small Gasoline Block," J. of KSAE, Vol. 15, No. 3, pp. 55-67, 1993.Hoyt, S. L., Metals Properties, Mc-Graw Hill, 1954.
51. Rothmann, M. F., High Temperature Property Data; Ferrows Alloys, ASM International, 1988.
52. Marc ZELLAT , Stefano DURANTI , Yongjun LIAN , Cedomir KRALJ., "Advanced modeling of GDI and DI-DIESEL Engines: Investigations on Combustion,14th International Multidimensional Engine User's Meeting at the SAE Congress 2004,March, 8,2004 Detroit, MI.
53. Ciesla, C., Keribar, R., and Morel, T., "Engine/ Powertrain/Vehicle Modeling Tool Applicable to All Stages of the Design Process", SAE 2000 Congress, Paper 2000-01-0934, 2000
54. Hiroyasu, H., Kadota, T., "Models for Combustion and Formation of Nitrid Oxide and Soot in Direct Injection Diesel Engines", SAE 1976 Congress, Paper 76029, 1976.
55. Morel, T., and Wahiduzzaman, S., "Modeling of Diesel Combustion and Emissions", XXVI FISITA Congress, Praha, Czech Republic, June 16-23, 1996.
56. Weiss, J., "Using Engine Cycle Simulation in Truck Engine Development", Diesel Progress, North American Edition, August 2003, pp. 42-45, 2003.
57. Hires, S. D. and Pochmara, G. L., "An Analytical Study of Exhaust Gas Heat Loss and flow through Exhaust Port," SAE Trans., Vol. 55, 1979.
58. Bianchi, G.M., Falfari, S., and Parotto, M., "Advanced Modeling of Common Rail Injector Dynamics and Comparison with Experiments", SAE 2003-01-0006, 2003.
59. Kong, S.C. and Reitz, R.D., "Multidimensional Modeling of Diesel Ignition and Combustion Using a Multistep Kinetics Model", J. Eng Gas Turbines Power, Vol.101, pp.781-789, 1993.
60. Kong, S.C., Han, Z., and Reitz, R.D., "The Development and Application of a Diesel Ignition and Combustion Model for Multidimensional Engine Simulations", SAE Paper 950278, 1995.
61. Patterson, M.A. and Reitz, R.D., "Modeling the Effects of Fuel Spray Characteristics on Diesel Engine Combustion and Emission", SAE 980131, 1998.
62. Senecal, P.K. and Reitz, R.D., "Simultaneous Reduction of Engine Emissions and Fuel Consumption Using Genetic Algorithms and Multi-Dimensional Spray and Combustion Modeling", SAE 2000-01-1890, 2000.

63. Wickman, D.D., Senecal, P.K., and Reitz, R.D., “Diesel Engine Combustion Chamber Geometry Optimization Using Genetic Algorithms and Multi-Dimensional Spray and Combustion Modeling”, SAE 2001-01-0547, 2001.
64. Borghi, M., Mattarelli, E., and Montosi, L., “Integration of 3D-CFD and Engine Cycle Simulations: Application to an Intake Plenum”, SAE 2001-01-2512, 2001.
65. Patel, S.N.D.H. (AVL UK); Bogensperger, M., Tatschl, R.; Ibrahim. S.S., Hargrave, G.K. (Loughborough Univ.): „Coherent Flame Modeling of Turbulent Combustion – A Validation Study“, 2nd M.I.T. Conf. on Computational Fluid and Solid Mechanics, June 17-20, 2003
66. Peters, B.: „Numerical Simulation of a Diesel Particulate Filter During Loading and Regeneration“, ASME 2003,11.-14.05.2003, Schloß Hellbrunn/Salzburg.
    Wurzenberger, J., Peters, B.: „Design and Optimization of Catalytic Converters taking into Account 3D and Transient Phenomena as an Integral Part in Engine Cycle Simulations“, ASME 2003, 11.-14.05.2003, Schloß Hellbrunn/Salzburg.
68. Xiao Hu., and Lisa Mesaros, “In Cylinder Power-CFD analysis of Cylinder Mass Flow”, FLUENT News Spring, 2004.
69. H.O. Hardenburg and F.W. Hase, An Empirical Formula for Computing the Pressure Rise Delay of a Fuel from its Cetane Number and from the Relevant Parameters of Direct-Injection Diesel Engines, SAE 790493, 1979.
70. www.ptc.com (Parametric Technology Corporation (PTC) web resources on solid modeling)
71. www.purduetech.edu (University of Purdue web resources on solid modeling and Finite Element Analysis)
72. www.catiacomminity.com (CATIA tutorials and tips)
73. www.femweb.com (website for FEM)
74. www.ansys.com (ANSYS website)
75. ANSYS reference manual, ANSYS Inc.
76. www.andru.cmu.edu (Carnegie Mellon university-ANSYS tutorial)
77. www.swri.com (South West Research Institute (SWRI) , Engine research)
78. www.ualberta.com (University of Alberta web resources on ANSYS tutorials)
79. Segerlind L.S. “Applied Finite Element Analysis,” John Wiley and Sons, Canada, 1995.
80. www.cfd.com
81. www.ansyscfx.com

Appendix – A

Mechanical Properties of Some Materials

Mechanical Properties of Some Materials			
Material	Modulus of Elasticity in GPa	Poisson's ratio	Yield strength in MPa
Steel (alloy steel -1040)	207	0.3	375
Cast iron (Grade 80-55-06)	168	0.30	379
Aluminium (Alloy 6061)	69	0.33	276
Copper (Alloy C71500)	150	0.34	140

Reference: Callister, W.D. "Materials science and engineering –An introduction," 6th Ed., John Wiley and Sons (Asia) Pte.Ltd, Singapore, 2004.

Appendix – B

Thermo Physical Properties of Some Materials

Thermo Physical Properties of Some Materials			
Material	Density in Kg/m^3	Thermal Conductivity in W/m. K	Specific heat in J/kg. K
Steel (alloy steel -1040)	207	52	486
Cast iron (Grade 80-55-06)	168	46	544
Aluminium (Alloy 6061)	69	220	896
Copper (Alloy C71500)	150	388	380

Reference: Callister, W.D. “Materials science and engineering –An introduction,” 6th Ed., John Wiley and Sons (Asia) Pte.Ltd, Singapore, 2004.