CRANKSHAFT ANALYSIS AND MODELING USING ANSYS SOFTWARE

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR

THE AWARD OF THE DEGREE OF

MASTER OF TECHNOLOGY
IN
COMPUTATIONAL DESIGN

SUBMITTED BY

SIDDHARTH BOON
ROLL NO. -2K18/CDN/08

Under the supervision of

Dr. R.C SINGH
(PROFESSOR)

DEPARTMENT OF MECHANICAL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042
JULY, 2020

CANDIDATE'S DECLARATION

I, SIDDHARTH BOON, Roll No. 2K18/CDN/08 student of M.Tech (Computational Design), hereby declare that the project Dissertation titled "Crankshaft Analysis Modeling Using Ansys Software" which is submitted by me to the Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi **SIDDHARTH BOON**

Date:

CERTIFICATE

I hereby certify that the project Dissertation titled "CRANKSHAFT ANALYSIS MODELING USING ANSYS SOFTWARE" which is submitted by SIDDHARTH BOON, Roll No. 2K18/CDN/08 Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of degree of Master of Technology, is a record of the project work carried out by the student under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi

Date:

Dr. R.C SINGH

PROFESSOR

Department of Mechanical Engineering

Delhi Technology University

ACKNOWLEDGEMENTS

Any accomplishment is a result of positivity of thoughts and efforts. It is important here to appreciate contribution, encouragement and support from persons who stood as 'Light House' throughout the voyage.

I wish to express my sincere gratitude for my project supervisor and mentor, Prof. R.C Singh, Department of Mechanical Engineering, Delhi Technological University (Delhi). It was a golden opportunity to work under his kind supervision. His scholastic guidance and sagacious suggestions helped me to complete the project.

I wish to thank Dr. Vipin, Head, Department of Mechanical Engineering, Delhi Technological University (Delhi), for constantly motivating and providing able guidance.

I am also thankful to the staff of Department of Mechanical Engineering for their continual support and cooperation.

Finally, but importantly, I would like to express my heartfelt thanks to my beloved parents Mr. Shishir Kumar and Mrs. Prabha Kumari who have endured the long working hours and whose motivation kept me going.I would also like to thank my friends Abhishek,Shubham and Gopal for their continuous support.

SIDDHARTH BOON

(Roll No. 2K18/CDN/08)

M.Tech. (Computational Design)

Delhi Technological University, Delhi, India

TABLE OF CONTENTS

ABSTRACT

The reliability of any system, during which the linear displacement of a piston is regenerate into the rotation of a power transmission shaft, strongly depends on the reliability of the crankshaft. The crankshaft is that the vital element and any damage occurring to the crankshaft might put the system out of order. Crankshaft is massive volume production component with a complex geometry within the ICE. This converts the reciprocating displacement of the piston into a rotation of the crank. The crank shaft takes the power from piston that is generated because of combustion method within the combustion chamber of the cylinder. Throughout the power transmission method the load acts at a specific crank angle to the max and thus the connecting rod is analyzed for the stress developed, due to load conditions and the changes mentioned. The existing works design first the 3D model of the engine parts are built in the software ‘CATIA V5’ and are then transferred to ‘ANSYS’. The analysis of crank throw distortion and stress provides a conceptual support to enhance the design by weigh reduction. The proposed research works the 3D model of crankshaft system, obtained from CATIA V5 software is analyzed in ANSYS to assess the motion and loads acting on the crankshaft. Topology optimization is help to optimize the performance of any machine. The present model is test in three different loading conditions 22624N, 32624N and 42624N load is applied in crank shaft. After analysis ad comparison of all present models it is seen that the lowest stress, deformation and weight are found in model 3 in all loading condition. The topology optimization process is help to reduce the material of workpiece without affecting the performance.

Keywords: ANSYS, crankshaft, CATIA V5, Forget steel, Topology optimization, weight reduction

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

In various mechanical engineering applications the most widely used machine elements is Shaft. The crankshaft, impeller shaft, propellor shafts, camshafts etc. use shaft. Crankshaft is one among the most vital moving elements consisting of 2 web sections and one crankpin that convert the piston reciprocator displacement to a rotation with a four link mechanism. Crankshaft experiences massive forces from gas combustion. This force is applied to the highest of the piston and since the connecting rod connects the piston to the crank shaft, the forces are transmitted to the crankshaft. The magnitude of the forces depends on several factors that include crank radius, connecting rod dimensions, and weight of the rod, piston, piston rings, and pin. Since the crankshaft experiences an outsized range of load cycles throughout its service life, fatigue performance and durability of this element needs to be thought of within the style method. That the lifetime of internal combustion engine and reliability depend upon the strength of the crankshaft mostly and because the engine runs, the power impulses hit the crankshaft in one place so another. Failures of shafts not only end in cost, however additionally in method period. The one among the most common causes of shaft failure is break. Fatigue failures are vital issues in mechanical styles. to determine the stress, geometry and dimensions Analysis, evaluations and engineering principles are utilized within the style. to confirm the lifetime of engine, strength calculation of crankshaft becomes a key issue. historically beam and area frame model were used to calculate the stress of crankshaft however in these models the number of node is restricted. With the development of computer, additional and additional design of crankshaft has been used finite element methodology (FME) to calculate the stress of crankshaft.(Randhavan and Galhe, 2017).

The crankshaft is a moving part of the internal combustion engine (ICEIts main operate is to transform the linear motion of the piston into rotational motion. The pistons are connected to the crankshaft through the connecting rods. The crankshaft is mounted inside the engine block. The crankshaft is fitted into the engine block through its main journals. The connecting rods are fixed on the conrod journals of the crankshaft. On opposite sides of the conrod journals the crankshaft has counterweights that compensates outer moments, minimizes internal moments and therefore reduces vibration amplitudes and bearing stresses.

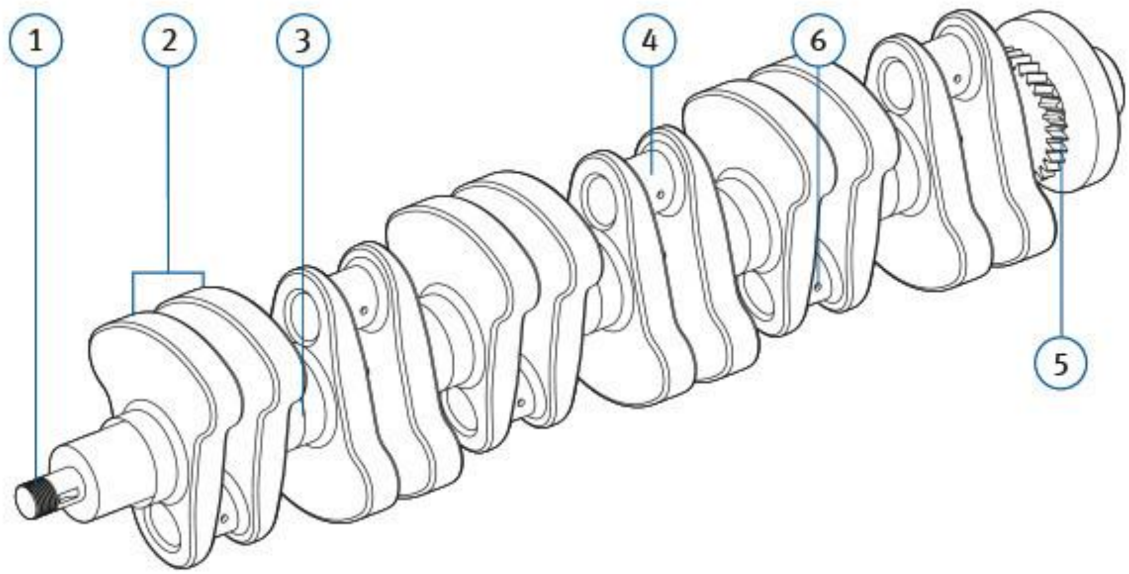


Fig.1.1 Crankshaft Nomenclature

Engine Crankshaft Description

1. Control side or drive end
2. Counterweights
3. Main bearing journal
4. Conrod journal
5. Flywheel side / force transfer
6. Oil bore

Crank shaft may be a giant element with a complex geometry within the I.C engine that converts the reciprocator displacement of the piston to a rotation with a four bar link mechanism. Crankshaft consisting of shaft components, 2 journal bearings and one crankpin bearing. The Shaft components that revolve within the main bearings, the crank pins to that the massive end of the connecting rod are connected, the crank arms or webs that connect the crank pins and shaft components. Additionally, the linear displacement of an engine isn't smooth; because the displacement is caused by the combustion chamber thus the displacement has sudden shocks.

The idea of using crankshaft is to vary these explosive displacements to as smooth rotary output that is the input to several devices like generators, pumps and compressors. It ought to even be expressed that the employment of a flywheel helps in smoothing the shocks. Crankshaft experiences large forces from gas combustion. This force is applied to the highest of the piston and since the rod connects the piston to the crank shaft, the forces are transmitted to the crankshaft. The magnitude of the forces depends on several factors that contain crank radius, connecting rod dimensions, and weight of the connecting rod, piston, piston rings, and pin. Combustion and inertia forces performing on the crankshaft.

1. Torsional load

2. Bending load

Crankshaft must be strong enough to take the downward force of the power stroke without excessive bending so the reliability and life of the internal combustion engine depend upon the strength of the crankshaft mostly. The crank pin is sort of a in-built beam with a distributed load on its length that varies with crank positions. Every net is sort of a cantilever beam subjected to bending and twisting.

1. Bending moment that causes tensile and compressive stresses.

2. Twisting moment causes shear stress.

1.2 INTRODUCTION OF TOPOLOGY OPTIMIZATION IN ANSYS

Unless it's been topologically optimized, each part in an assembly most likely weighs over it has to. additional weight means that excess materials are being employed, loads on moving elements are more than necessary, energy efficiency is being compromised and shipping the part prices additional. Now, with Topology improvement technology, ANSYS Mechanical offers you the tools you need to style durable, light-weight elements for any application. You can outline objectives simply and apply controls to confirm that producing necessities are met, minimum material thicknesses are set and exclusion areas are outlined.

Topology improvement in ANSYS Mechanical permits you to:

- Takings into account several static loads joined with optimizing for natural frequencies (modal analysis)
- Fulfil necessities for minimum material thickness
- Observe rules around feature direction (for machining operations for example)
- Have scope for both cyclic and planar symmetry
- Easily validate results

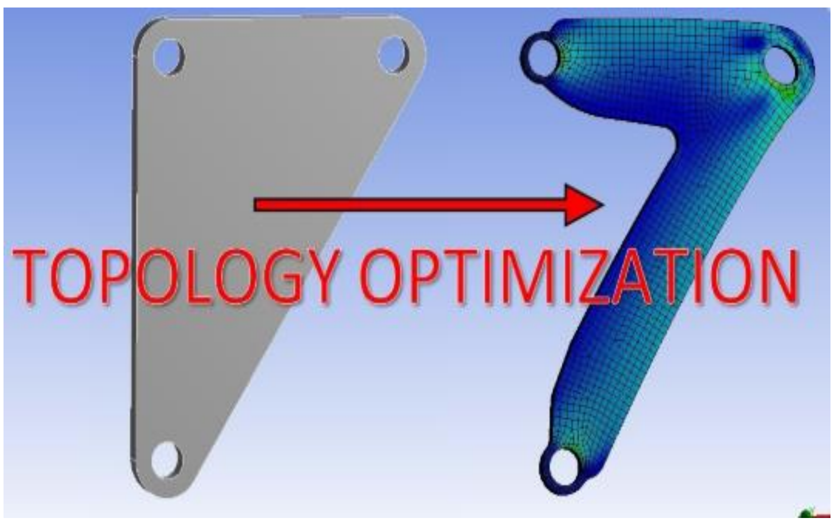


Fig.1.2 Topology optimization

1.3 METALS AND ALLOYS

1.3.1 Metals:

A metal could be a material that's usually hard, opaque, shiny, and has smart electrical and thermal conduction. Metals are usually malleable that's, they will be hammered or pressed for good out of form while not breaking or cracking also as melted and ductile. Metals normally have high electrical conduction, high thermal conduction and high density. Mechanical properties of metals include ductility, i.e. their capability for plastic deformation. Reversible elastic deformation in metals may be represented by Hooke's law for restoring forces, wherever the stress is linearly proportional to the strain. Forces larger than the elastic limit, or heat, might cause a permanent (irreversible) deformation of the thing, referred to as plastic deformation or plasticity.

This irreversible change in atomic arrangement may occur as a result of:

- The action of an applied force (or work). An applied force may be tensile (pulling) force, compressive (pushing) force, shear, and bending or torsion (twisting) forces.

1.3.2 Alloys:

An alloy may be a mixture of 2 or a lot of components during which the most part may be a metal. Most pure metals are either too soft, brittle or with chemicals reactive for practical use. Combining totally different ratios of metals as alloys modifies the properties of pure metals to provide desirable characteristics. The aim of making alloys is mostly to form them less brittle, harder, immune to corrosion, or have a lot of desirable color and luster of all the metallic alloys in use these days, the alloys of iron form up the biggest proportion each by amount and business price. Iron alloyed with varied proportions of carbon offers low, mid and high carbon steels, with increasing carbon levels reducing ductility and toughness. The addition of Si can produce cast irons, whereas the addition of Cr, nickel and metal to carbon steels leads to stainless steels. Different vital metallic alloys are those of aluminium, titanium, copper and Mg. Copper alloys are best-known since period of time bronze gave the Bronze Age its name and have several applications these days, most significantly in electrical wiring. The alloys of the opposite 3 metals are developed relatively recently; because of their chemical reactivity they need electrolytic extraction processes. The alloys of Al, Ti and Mg are valued for their high strength-to-weight ratios; Mg may also give electromagnetic shielding. These materials are ideal for things wherever high strength to weight ratio is a lot of necessary than material value, like in region and a few automotive applications. Alloys specially designed for highly demanding applications, like jet engines, could contain quite 10 components.

Iron Alloys and Materials: There are a number of different kinds of alloys containing iron. a number of the most necessary include carbon steels, alloy steels, stainless steels, tool steels, cast iron, and managing steel**.**

Carbon steels are steels within which the most alloying additive is carbon. Mild steel is the commonest because of its low value. it's neither brittle nor ductile, has comparatively low tensile strength, and is malleable. Surface hardness will be increased through carburizing. High carbon steels have a better carbon content that provides a far higher strength at the value of plasticity**.**

Alloy steels are steels (iron and carbon) alloyed with different metals to enhance properties. the most common metals in low alloyed steels are Mo, chromium, and nickel to enhance weld ability, formability, wear resistance, and corrosion resistance**.**

Stainless steels are steels that contain a minimum of 100% Cr. There are several grades of stainless-steel; however the most common grade used for typical corrosion resistant applications is sort 304, additionally referred to as 18-8. The term 18-8 refers to the number of Cr (18%) and nickel (8%) combined with iron and alternative parts in smaller quantities. The metal's end is pictured by variety, 3 to 8, with three being the roughest and eight being a mirror-like end. different specifications to consider include textures and coatings**.**

Tool steels are specific steels designed for being created into tools. they are known for toughness, resistance to abrasion, ability to carry a cutting edge, and/or their resistance to deformation at high temperatures. The 3 varieties of tool steel offered are cold work steels utilized in lower operational temperature environments, hot work steels used at elevated temperatures, and high speed steels able to withstand even higher temperatures giving them the power to cut at higher speeds**.**

Cast iron is AN iron alloy derived from pig iron, alloyed with carbon and Si. Carbon is added to the base melt in amounts that exceed the solubility limits in iron and precipitates out as C particles. Semiconductor is added to the melt to nucleate the C that optimizes the properties of cast iron. typically discharged as a cheap, dirty, brittle metal; cast iron is obtaining way more attention and use nowadays due to its physicist inability, lightweight weight, strength, wear resistance, and damping properties**.**

Merging steels are carbon free iron-nickel alloys with additions of Co, molybdenum, titanium, and metal. The term managing springs from the strengthening mechanism, that is transforming the alloy to marten website with subsequent age hardening. With yield strengths between 1400 and 2400 MPa, merging steels belong to the class of ultra-high-strength materials. The high strength is combined with excellent toughness properties and weld ability.

1.4 MOTIVATION OF RESEARCH

The current methods of manufacturing crankshafts include casting with iron or forging in steel. Cast iron crankshafts are a relatively low cost for high production, while forged steel crankshafts offer a greater fatigue life in high-performance engines. Due to the increasing need for high-performance, high-efficiency engines in both the automotive and industrial vehicles, cast iron crankshafts may not continue to meet the strength and fatigue criteria. At the same time, forged steel crankshafts, are able to meet strength and fatigue criteria, but are not as cheaply manufactured as cast crankshafts. The forging manufacturing process also limits the ability to reduce the weight of the crankshaft by hollowing out journals, which is easily done in metal casting. Reducing the weight of the crankshaft improves the fuel efficiency of the engine. A high-performance cast steel crankshaft combines the benefits of both aforementioned crankshaft-manufacturing processes. Steel is a stronger metal than iron, and the casting process allows for the use of cores to create hollow sections, which leads to a lighter more efficient crankshaft.

1.5 OBJECTIVE

- To model a crankshaft on software
- To perform static analysis of crankshaft
- To optimize the crankshaft design using topology optimization method.
- To reduce the overall weight of crankshaft implementing the new optimised design.
- To compare the existing crankshaft design with new optimised design in terms of obtained von-mises stress and deformation.
- To compare the weight of conventional crankshaft with new optimised design model.

1.6 PROBLEM STATEMENT

As crankshaft is generally made of ferrous material, it has high density the crankshaft become bulky. Because of bulky crankshaft weight of the engine is increases, hence selecting a new material to reduce the weight of the crankshaft without compromising on the properties of the material.

1.7 ORGANIZATION OF THESIS

Chapter 1 presents a brief description of the Design and Analysis of Crankshaft, i.e., general introduction, objective, research problems, research motivation, and introduction of materials and the structure of the thesis.

Chapter 2 reviews previous research related to the Design and Analysis of Crankshaft using topology optimization in ANSYS of the current investigations in more detail.

Chapter 3 describes the methodology of proposed design of crankshaft, flow chart of methodology, modelling and meshing.

Chapter 4 represents the simulation results of proposed research work. It includes different cases simulation results.

Chapter 5 summarizes the outcome of the proposed research work.

CHAPTER 2

LITERATURE SURVEY

(Thejasree, Dileep Kumar and Leela Prasanna Lakshmi, 2017) presented analysis the effect of gas forces was analyzed at crankpin and main journals of the crankshaft. The maximum load acting on the crankpin was found to be 22163 N for the bench mark model whereas for the developed concepts 1, 2 and 3, it was found to be 22624 N, 22066 N and 22303 N respectively. The maximum stress for the bench mark model was found to be 67 MPa and whereas for the developed concepts 1, 2 and 3, it was found to be 80MPa, 71 MPa and 79 MPa respectively. Structural static analysis shows that the stress concentration regions are located at the fillets of crank pin and main journals. Dynamic stress and strain analysis has been carried to determine the stress and deformation of the crankshaft in a working cycle. The weight of the crankshaft for concept-2 has been reduced by 1.6 kg which is a 12.8% reduction in weight without much increase in the stress.

(Ahmad and Gurlal, 2018) Crankshaft is one amongst the most vital moving elements in combustion engine. Crankshaft could be a massive part with a complex geometry within the engine that converts the mutual displacement of the piston into a rotation. It should be strong enough to require the downward force throughout power stroke while not excessive bending. That the reliability and lifetime of combustion engine depend upon the strength of the crankshaft mostly. Maximum deformation produced in the crankshaft before optimization 0.11265 and after optimization 0.11189. Maximum stress produced in the crankshaft before optimization 269.2 and after optimization 257.71. Maximum strain produced in the crankshaft before optimization 0.0014779 and after optimization 0.0013648. Max Deformation, Stress and Strain are produced in the old design of the crankshaft. The optimized design has less stress produced as compared to the old design. And also the weight decrease in the new optimized design (approximate 0.83%).Therefore we can prefer the new optimized design of the crankshaft.

(R.Raju, 2018) observed that there has been considerable increase in the von mises stress (max.) for Grey Cast Iron. We can also derive that the maximum deformation value is 5.572e-3mm for

Grey Cast Iron as compared to 5.821e-3mm with Steel. Hence, fatigue life will be more for Grey Cast Iron than that of Steel.

(Reddy, Prasad and Reddy, 2017) discussed the crankshaft model was created by Pro/ENGINEER software. Then the created model was imported to ANSYS software. After structural analysis of crankshaft the maximum stress value on forged steel is 150 MPA and on composite material is 146 MPA. So the composite materials have better performance than forged steel. But the manufacturing cost of composites is higher than the forged steel so due to the low cost of forged material, it's almost used for crankshaft.

(Karandikar et al., 2017) observed from the stress analysis using ANSYS software, maximum deformation or displacement in crankshaft happens at the top central portion of the crankpin. The maximum stresses befall in the regions where there is quite a sudden change in the geometry of the crankshaft. This area is the junction where the crank web connects the shaft. This is a natural because the stress lines suddenly change themselves. This leads to the accumulation of high stresses over there, making the material weak. These fillets should be designed such that the crankshaft is least affected by these high stresses. Also, high surface finish should be maintained to minimize these stresses. This may be the reason that the manufacturers nowadays prefer forged materials over cast materials. Since the crankshaft is a rotating element and is imperilled by cyclic loading, there are chances of fatigue cracks being developed in these locations over time. This finally leads to fatigue failure.

(Yashik, Ramprasanth and Manikandan, 2017) The conventional crank shaft used in the engines was replaced with a Composite crank shaft. The conventional crank shaft and the Composite crank shaft were analyzed by finite element methods. From the results, it is clear that the stress induced in the Composite crank shaft is found to be lower than that of the conventional crank shaft. Composite crank shaft material is replaced for good fatigue strength, minimizing weight and without violating the limiting constraint formed by induced stress. A reduction of 37.5% weight is achieved when a conventional crank shaft is replaced with Composite crank shaft under identical conditions of design parameters.

(Randhavan and Galhe, 2017) conclude that to reduce the weight of the crankshaft it should use aluminum alloy without compromising the properties of the existing material. By using FEA results we can analyses the von misses stresses, max principal stress and deformation when load is applied considering boundary conditions. Experimental performances will help to analyze the new material.

(Pandiyan et al., 2016) studied topology optimization in Crank case cover reduced a net mass of 0.737kg from the initial design. This could lead to a better performance of the engine. While achieving this reduction in mass, necessary precaution is to be taken to ensure the factor of safety is still in the industry standards. The advanced and High Performance Computing with ANSYS Topology improvement formed a stronger result than the conceptual designs. There are 2 issues to be mentioned and solutions ought to be suggested. The primary one high localized stress at the fastenings, by increasing space by providing fillets or offer much length for the fastening support this drawback will be resolved. The second drawback is throughout producing; standard manufacturing strategies like Casting, Molding, and Forming are not possible with the planning. Meanwhile Additive producing can provides a higher result. The goal of the paper is to get a 30 percentage in weight reduction of the initial style domain; however the result was a promising 50 percentage in weight reduction.

(Sowjanya V, et al., 2016) discussed the crankshaft model is created by Pro/ENGINEER software. Then the model created by pro/Engineer was imported to ANSYS software. The maximum deformation seems at the center of crankshaft surface. The most stress seems at the fillets between the crankshaft journal and crank cheeks, and close to the central point. The edge of main journal is high stress space. The crankshaft deformation was principally bending deformation under the lower frequency. and therefore the most deformation was set at the link between main bearing journal and crankpin and crank cheeks. Therefore this space prones to look the bending fatigue crack. Base on the results, it will forecast the chance of mutual interference between the crankshaft and different components. The resonance vibration of system may be avoided effectively by appropriate structure design. The results give a theoretical basis to optimize the planning and fatigue life calculation. Dynamic loading analysis of the crankshaft ends up in a lot of realistic stresses whereas static analysis provides overestimated results. Correct stresses are important input to fatigue analysis and improvement of the crankshaft.

(Shelke, Dhamejani and Gadhave, 2016) The comparison between baseline and optimized style of the crankshaft for von-Mises stress shows higher enhancements. the stress raised from 204 MPa to 241 MPa however, the stress values invariably stay among yield criteria of fabric. The deflection in crankshaft for baseline style and optimized style are 0.0182 millimetre and 0.018 millimetre severally. the overall mass of the baseline model was 3.5 kilogram that is small up to 2.95 Kg. the proportion reduction in mass of the crankshaft determined up to 19. The baseline analysis is compared with the testing of the baseline specimen. The maximum variation determined between simulation and testing is up to 18.64% that is appropriate.

(Sujata Satish Shenkar, 2015) The stress analyses of a single-cylinder crankshaft are discussed using finite element method in this paper. Three dimension models of single crankshaft and crank throw were created exploitation Pro/ENGINEER package. The finite element analysis (FEM) software system ANSYS was won't to analyze. Result shows that FEA Results matches with the theoretical calculation therefore we will say that FEA may be a good tool to reduce time overwhelming theoretical Work. The most deformation seems at the middle of crankpin neck surface. The most stress seems at the fillets between the crankshaft journal and crank cheeks and close to the central purpose Journal. The edge of main journal is high stress space. The value of Von-Misses Stresses that comes out from the analysis is much but material yield stress therefore our style is safe and that we ought to select improvement to reduce the material and value.

(Pawar *et al.*, 2015) offers a summary of RP technology in short-term and stresses on their capacity to reduce the product design and development process. Classification of RP processes and details of few important processes is given. The description of various stages of data preparation and model building has been presented. An attempt has been created to incorporate some necessary factors to be thought-about before beginning part deposition for correct utilization of potentials of RP processes. Speedy prototyping won't build machining obsolete, however rather complement it.

(Marchesi *et al.*, 2015) research is a first step towards an effective application of topology optimization and additive manufacturing for the diesel engine support. As future work, the confirmation that the additively manufactured and topologically optimized part can replace the original one is a must. This example shows that topological optimization and additive manufacturing have great potential to replace conventional design and manufacturing processes.

(Baragetti, 2015) Useful criteria for the design of high power engines crankshaft have been reported in this paper. Numerical models were developed. The experimental evaluation of some parameters, such as the damping coefficient, needed to be determined in order safely verifies the resistance of the crankshaft. The procedure can be extended to any other kind of crankshafts.

(Gill *et al.*, 2014) a dynamic simulation was showed on a crankshaft for a single cylinder four stroke camless Engine. Finite element analysis was achieved to gain the variation of stress amount at serious locations. The maximum load ensues at the crank angle of 352 degrees for this specific engine. At this angle only bending load is useful to the crankshaft.

(Prasad and Somasundar, 2014) studied a cast iron crankshaft of one cylinder four –stroke ICE was taken and a static analysis was conducted to induce variation of stress magnitude at essential locations of the crankshaft. A model was created in CATIA of crank shaft and foreign into ANSYS to carryout static analysis. Meshing of crankshaft was done; hundreds and boundary conditions were applied as per the mounting conditions of the crankshaft on Finite component model of crankshaft. Results obtained from the analysis were then employed in optimization of the cast iron shaft. Weight improvement is achieved by variable the crankpin diameter. this needs the stress zero in Fe analysis to not exceed the magnitude of the stress zero in the first crankshaft. The improvement method involves geometry ever-changing while not ever-changing block.

(Singh et al., 2014) said that Dynamic FEA could be a smart tool reduces pricey experimental work. By observant the static analysis results shows that stress assesses exploitation nickel stainless steel and steel crank shafts from one cylinder four stroke engine are at intervals the permissible stress price. Thus using nickel stainless steel and steel is good for crank shafts however as compared between nickel chrome and steel, nickel chrome is best suited material over the steel.

(‘Ravi Kumar Goel, 2014) On the idea of this studies performed, it will be all over that the planning parameter of rod will be changed in such the way in order that sufficient improvement within the existing results will be obtained. Throughout the planning improvement, weight of the crankshaft is additionally reduced by 193 gm which ends in reduction in inertia and centrifugal forces. It had been found that the utmost stress purpose region was at the knuckle of the center main journal shaft and crank arm.

(Kütük and Göv, 2013) a new algorithm is developed for topology optimization of 3D machine parts. Validity of the algorithm is proved by means of simple beam parts and two parts from industrial application. The results of the comparisons imply that the element removal algorithm can be safely used for 3D machine parts. Also solution times are compared with ANSYS and up to 75% time reduction is obtained by using the element removal algorithm. When the stress and deformation results are compared, up to 50% reduction can be obtained depending on the boundary conditions in the ERM results.

(Khatawate and Idris, 2013) presents results of strength analysis done on crankshaft of one cylinder 2 stroke petrol engine, to optimize its style, using PRO/E and ANSYS code. The 3 dimensional model of crankshaft was developed in PRO/E and imported to ANSYS for strength analysis. The crankshaft was found to be over dimensioned. so web thickness was reduced from 13 millimetre to 10 millimetre. The reduction in mass obtained by this style modification is: Mass Reduction = 1.9725 kilo - 1.6375 kilo = 0.335 kilo percentage Mass Reduction = 16.98%.

(Thriveni K., 2013) Crankshaft is large volume production element with a complex geometry within the internal combustion (I.C) Engine. This converts the mutual displacement of the piston in to a rotary motion of the crank. An effort is formed during this paper to study the Static analysis on a crankshaft from one cylinder 4-stroke I.C Engine. The modelling of the crankshaft is formed exploitation CATIA-V5 software system. Finite element analysis (FEA) is performed to get the variation of stress at important locations of the crank shaft exploitation the ANSYS package and applying the boundary conditions. Then the results are drawn Von-misses stress induced within the crankshaft is 15.83Mpa and shear stress is induced within the crankshaft is 8.271Mpa. The Theoretical results are obtained von-misses stress is 19.6Mpa, shear stress is 9.28Mpa. The validation of model is compared with the Theoretical and FEA results of Von-misses stress and shear stress are among the boundaries. Additional it is often extended for the various materials and dynamic analysis, optimization of crank shaft.

(Deshbhratar and Suple, 2012) the crankshaft model is created by Pro/ENGINEER software. Then the model created by pro/Engineer was imported to ANSYS software. The maximum deformation seems at the centre of crankshaft surface. The most stress seems at the fillets between the crankshaft journal and crank cheeks, and close to the central purpose. The edge of main journal is high stress space. The crankshaft deformation was in the main bending deformation under the lower frequency. And also the most deformation was set at the link between main bearing journal and crankpin and crank cheeks. Therefore this space prones to appear the bending fatigue crack. Base on the results, we will forecast the likelihood of mutual interference between the crankshaft and alternative elements. The resonance vibration of system is often avoided effectively by acceptable structure style. The results offer a theoretical basis to optimize the planning and fatigue life calculation.

(R.M. Metkar, 2011) Crankshaft is a crucial part of engine, failure even creating engine useless additionally needs costly procurement and replacement. An intensive analysis within the past clearly indicates that the matter has not however been overcome fully and designers are facing lot of issues specially connected with multiaxial loading (Bending and Torsion), stress concentration and stress gradient and impact of variable amplitude loading. The finite element technique is the preferred approach and located usually used for analyzing fracture mechanics issues. the strategy may be applied to linear and non-linear issues. There are several commercial packages are obtainable for use in fracture mechanics applications, such as, Ansys Fatigue, Abaqus, Nastran, MSC. Fatigue and msc.Marc. These are currently equipped with many smart techniques and methodologies to see vital terms of fracture mechanics parameters like the J-integral, Fatigue Life Estimation by using Finite element Analysis that additionally yields considerably an improved result. However it's been determined that none of them have an in-built crack propagation capability. There are few additional software system tools recently been developed for fatigue analysis includes FESAFE, FEMFAT and nCODE. These tools not only helpful to predict location of crack and its propagation, but also, together with finite component software system like Ansys, Abaqus, Nastran, MSC. Fatigue and msc. Marc extremely builds wonders in crucial fatigue lifetime of any engineering elements.

(Gu and Zhou, 2011) Through the finite part analysis on a four-cylinder ICE crankshaft, it shows that the high stress region principally concentrates within the knuckles of the crank arm journal, and therefore the crank arm journal, that is the space most simply broken. Provided a theoretical basis and information support for the more improvement style.

(Shaari et al., 2010) The modeling of connecting rod and Fe Analysis has been given. Topology improvement were analyzed to the connecting rod and in keeping with the results, it is over that the load of optimized style is 11.7% lighter and most stress conjointly predicted not up to the initial style of rod. The results clearly indicate that the new style a lot of lighter and have a lot of strength than initial style of rod.

(Montazersadgh and Fatemi, 2007) Dynamic loading analysis of the shaft leads to additional realistic stresses whereas static analysis provides overestimated results. Correct stresses are crucial input to fatigue analysis and improvement of the crankshaft. There are 2 totally different load sources in an engine; inertia and combustion. These 2 load supply cause each bending and torsional load on the crankshaft. the utmost load happens at the crank angle of 355 degrees for this specific engine. At this angle only bending load is applied to the crankshaft. Considering torsional load within the overall dynamic loading conditions has no result on von Mises stress at the critically stressed location. The result of torsion on the stress vary is additionally comparatively small at different locations undergoing torsional load. Therefore, the shaft analysis can be simplified to applying only bending load. Superposition of FEM analysis results from 2 perpendicular masses is an efficient and easy technique of achieving stresses for various loading conditions consistent with forces applied to the crankshaft from the dynamic analysis. Experimental stress and FEA results showed shut agreement, at intervals seven-membered difference. These results indicate non-symmetric bending stresses on the crankpin bearing, whereas using analytical technique predicts bending stresses to be symmetrical at this location. the lack of symmetry could be a pure mathematics deformation result, indicating the requirement for FEA modeling because of the comparatively advanced geometry of the crankshaft. crucial (i.e. failure) locations on the crankshaft geometry are all situated on the fillet areas owing to high stress gradients in these locations, that lead to high stress concentration factors.

CHAPTER 3

METHODOLOGY

3.1 STEP OF WORKING

Proposed methodology to be adopted while performing the design, optimization and analysis on crankshaft design:

1. Design parameters in the present design were obtained by study of previous paper.
2. After obtaining the desired dimensions further the model in CATIA V5 will be created.
3. After the modelling part the design of crankshaft model will be further imported in Ansys static structural analysis workbench to perform the static structural analysis on it.
4. After the static analysis areas having higher and lower stress will be evaluated to perform the topology optimisation on it.
5. After the evaluation part further the model is imported in topology optimization work bench to have the areas which can be removed from the model having less stress values to have the weight reduction in overall crankshaft.
6. After the topology optimization again the model will be recreated in CATIA V5 and further static analysis will be performed.
7. After the analysis both the conventional and optimized model will be compared and evaluated to suggest the overall improvement in the crankshaft design.

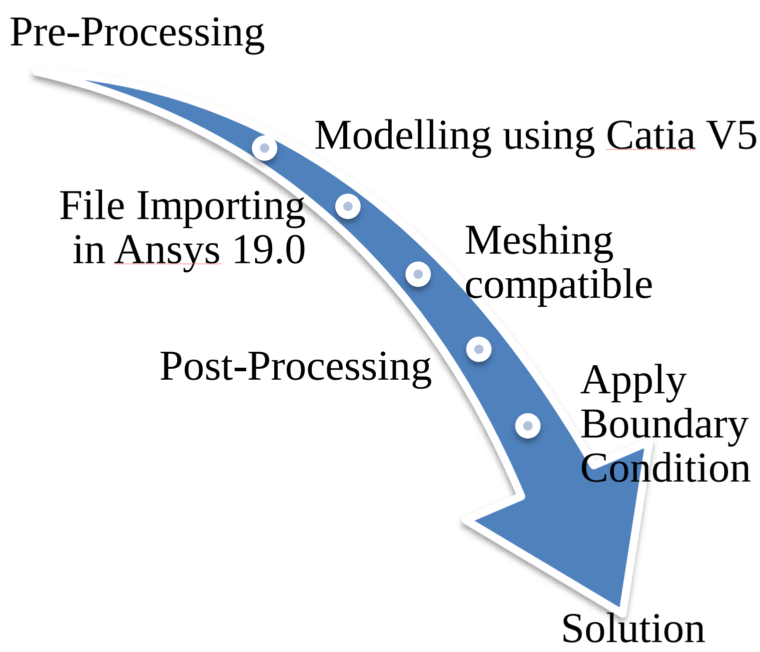


Fig.3.1 Flow chart of methodology

3.2 DESIGN ANALYSIS

Existing load condition and stress generation

The force acting on the crankpin for concept-1 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 22624 N. Similarly the maximum force acting on the crankpin due to gas loads at 4500 is 22066 N for concept-2 and 22303 N for concept-3.

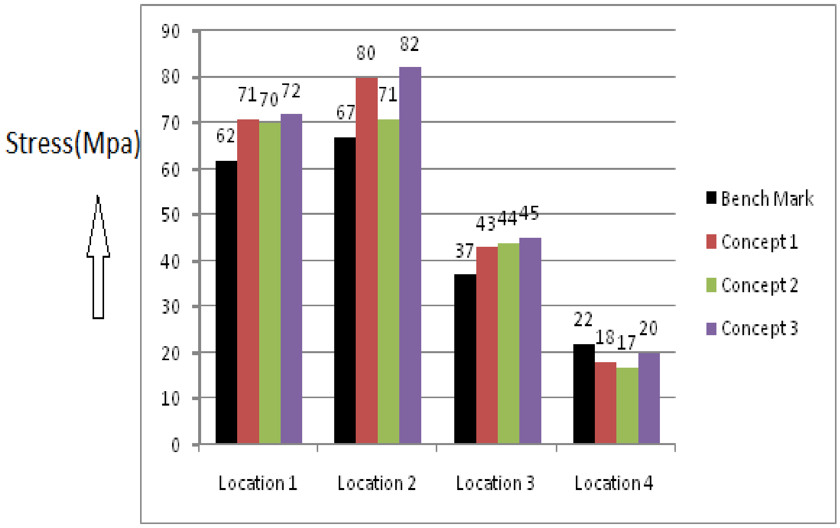


Fig 3.2(a)

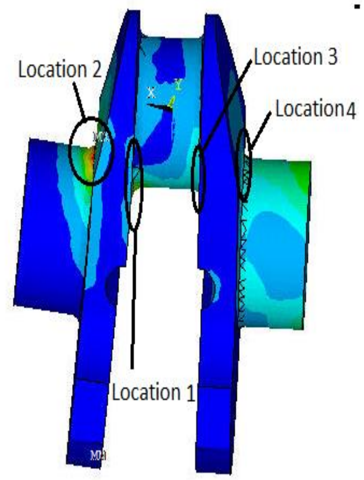


Fig 3.2(b)

It was found to be 22624 N, 22066 N and 22303 N respectively. The maximum stress for the bench mark model was found to be 67 MPa and whereas for the developed concepts 1, 2 and 3, it was found to be 80MPa, 71 MPa and 79 MPa respectively.

3.3 PROPOSED METHODOLOGY

Computer Aided Design gives point by point data on Computer Aided Design, Computer Aided Design Software, Computer Aided Design and Manufacturing, Computer Aided Design and Engineering and that's just the beginning. Computer Aided Design is associated with Cam and Computer Aided Design.

3.4 SOFTWARES IN USE

In light of the idea of CAD innovation, numerous CAD programming have been created by programming goliaths like Auto-work area Inc, Bentley, Dassult Systemes, Some of the main programming in the business are Auto-CAD, SOLIDWORKS, CATIA, Pro-Engineer, Uni-designs, Solid-Edge, STAAD Pro, Auto-Civil, Auto-work area Inventor and the rundown continues forever.

Because of CAD offices, the reiteration of work are limited, precise exactness can be accomplished; propagation isn't an issue now days. After change into computerized position it can likewise be sent through electronic mail to any piece of the world as an editable document. Because of accessibility of a great deal of record designs, a similar document can be opened and utilized in an assortment of CAD programming.

3.5 INTRODUCTION TO CATIA

CATIA is an abbreviation for Computer Aided Three-dimensional Interactive Application. It is one of the main 3D programming utilized by associations in various businesses extending from aviation, vehicle to customer items.

CATIA gives the ability to picture designs in 3D. At the point when it was presented, this idea was imaginative. Since Dassault Systems didn't have a skill in advertising, they had income sharing tie-up with IBM which demonstrated incredibly productive to both the organizations to showcase CATIA. In the beginning phases, CATIA was broadly utilized in the design of the Mirage airplanes; anyway the capability of the product before long settled on it a mainstream decision in the car division also. As CATIA was acknowledged by increasingly fabricating organizations, Dassault changed the item grouping from CAD/CAM programming to Project Lifecycle Management. The organization likewise extended the extent of the product.

CATIA can be utilized at various phases of the design - ideate, draw, test and repeat. The product accompanies various workbenches ("modules") that permit CATIA to be utilized across changed businesses – from parts design, surface design and get together to sheet metal design. CATIA can likewise be utilized for CNC.

CATIA offers numerous workbenches that can be inexactly named as modules. A couple of the significant workbenches and their concise usefulness depiction is given underneath:

Part Design: The most basic workbench required for strong demonstrating. This CATIA module makes it conceivable to design exact 3D mechanical parts with a natural and adaptable UI, from portraying in a gathering setting to iterative nitty gritty design.

Generative Shape Design: permits you to rapidly show both straightforward and complex shapes utilizing wireframe and surface highlights. It gives an enormous arrangement of instruments for making and altering shape designs. Despite the fact that not fundamental, information on Part Design will be extremely convenient in better usage of this module.

Get together: The essentials of item structure, imperatives, and moving congregations and parts can be adapted rapidly. This is the workbench that permits associating all the parts to frame a machine or a segment.

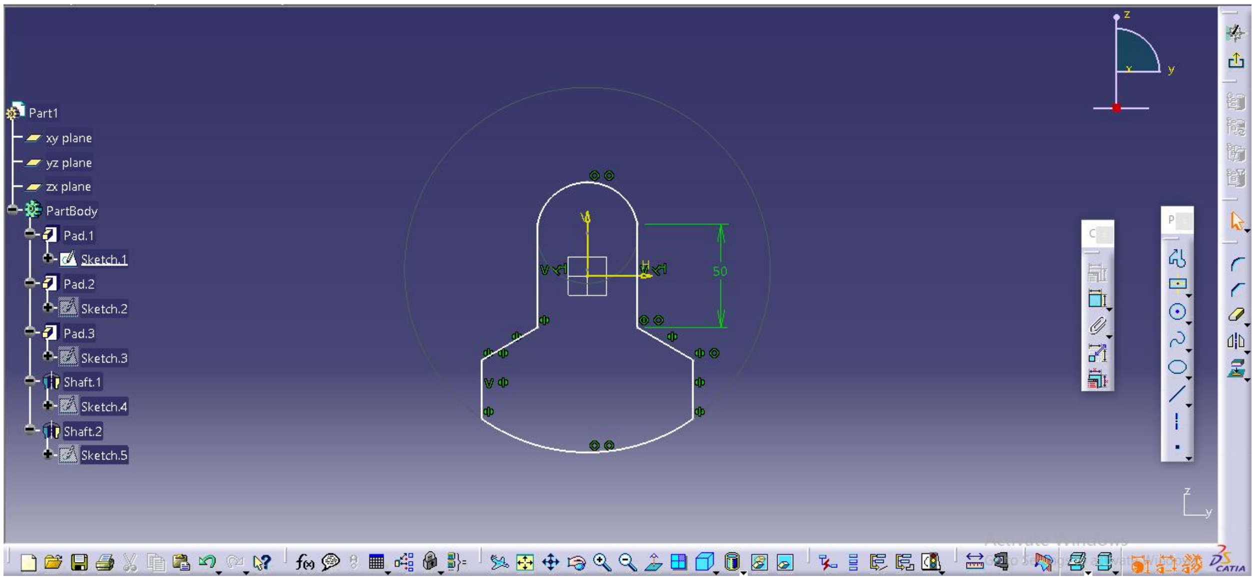


Figure 3.3: Design View

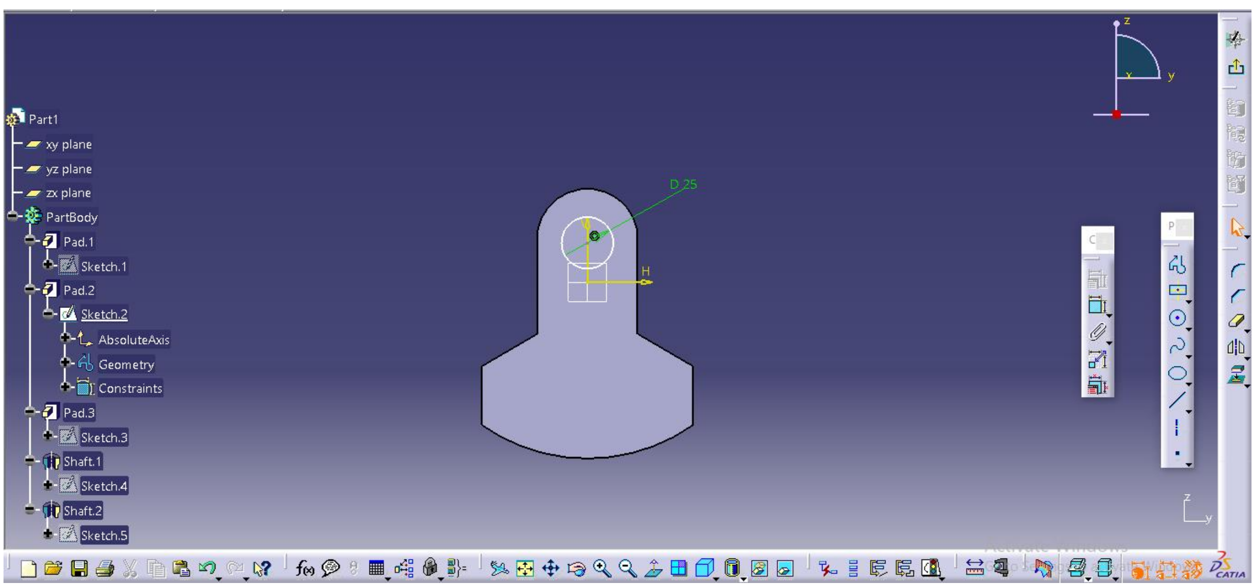


Figure 3.4: Design View part 2

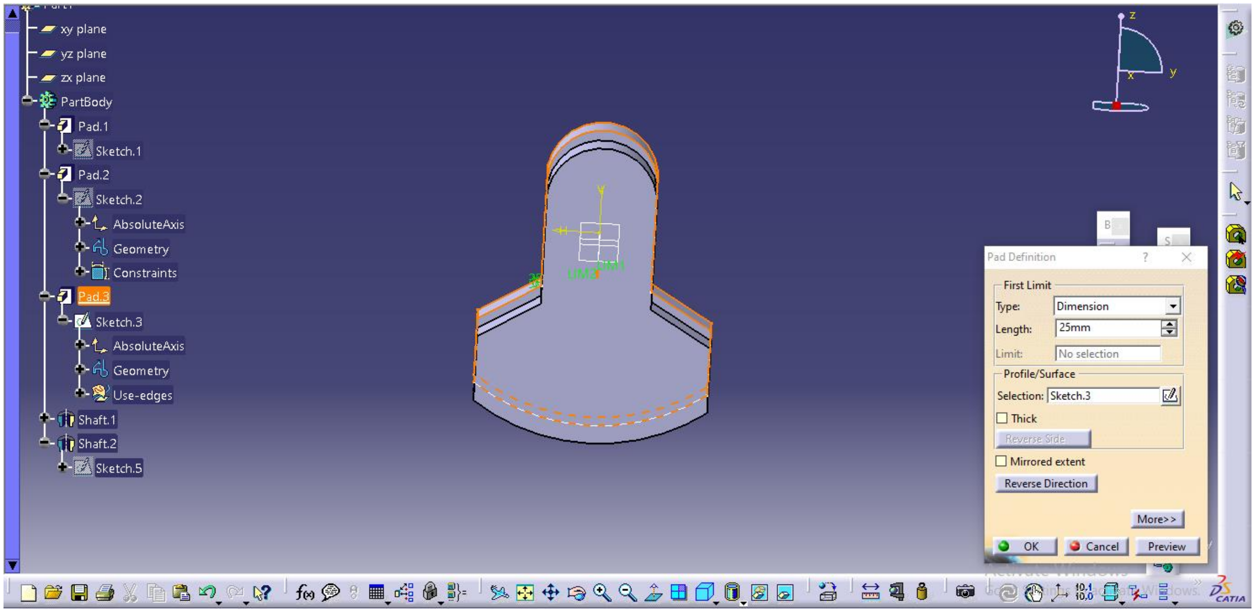


Figure 3.5: Design View 3

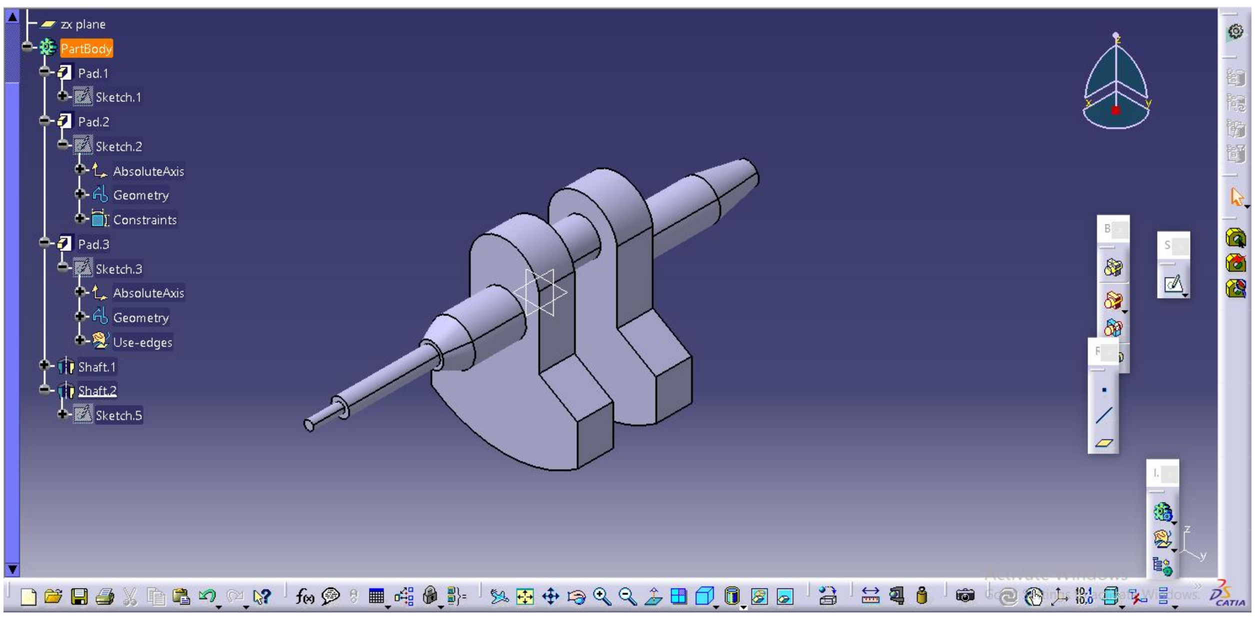


Figure 3.6: Design View

Figure 3.3 to 3.6, represents the modeling of the crankshaft.

3D MODEL

In present research 1 basic model and 3 implement model is proposed. The crankshaft model 3D geometry is created by using CATIA V5 and then it is import in ANSYS 19.0. And check all the result in three different load cases 22624N, 32624N and 42624N.

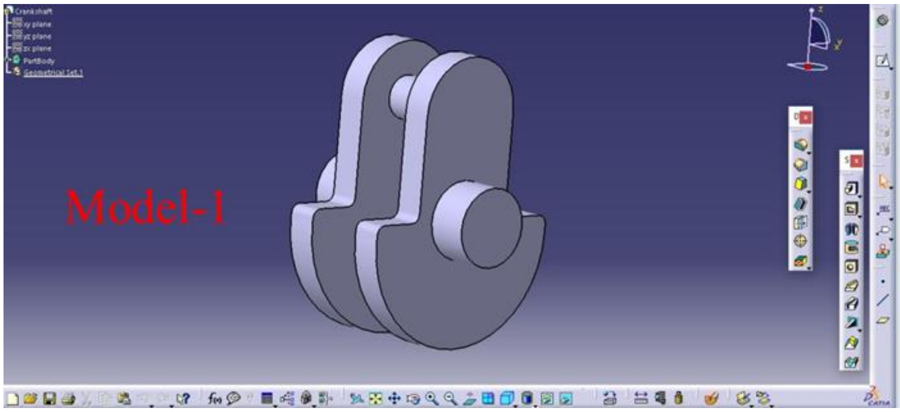


Fig. 3.7 Model 1

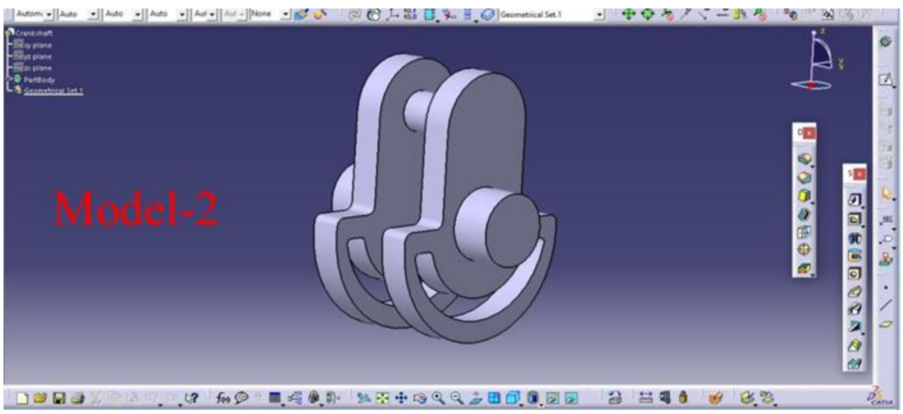


Fig. 3.8 Model 2

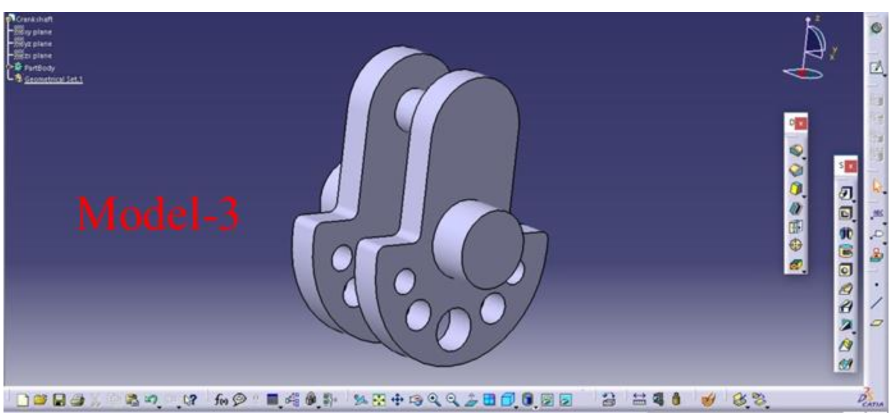


Fig. 3.9 Model 3

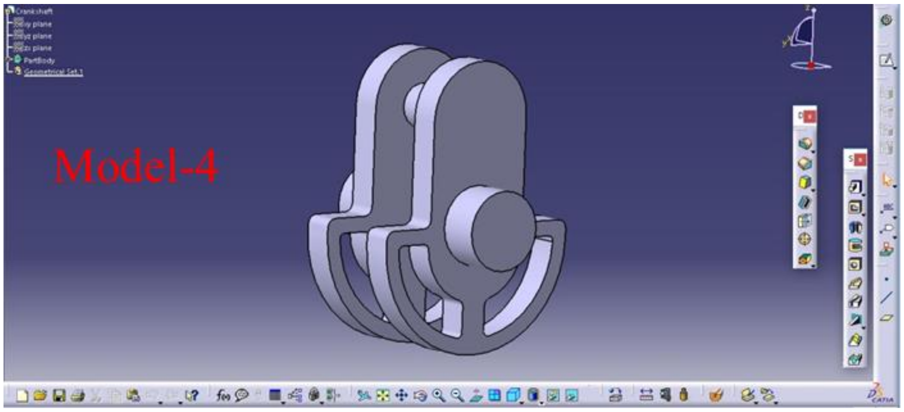


Fig. 3.10 Model 4

3.3.1 Material property

The forget steel is used to above analysis the mechanical property are used in ANSYS are shown in table.

Table 3.1 Material property

Material	Forget steel
Density (Kg/m3)	7833
Young’s Modulus (MPa)	2.21E+05

3.6 MESHING

After applying meshing 26641 nodes and 19234 element is generated in crankshaft.

Implementation:

Ansys Simulation is a design investigation framework completely coordinated with Solid works. Strong works Simulation gives reenactment answers for direct and nonlinear static, recurrence, clasping, warm, weakness, pressure vessel, drop test, straight and nonlinear dynamic, and streamlining examinations.

Figure 3.11 : simulation example

Benefits of Simulation:

In the wake of building your model, you have to ensure that it performs productively in the field. Without investigation devices, this errand must be replied by performing costly and tedious item advancement cycles. “An item improvement cycle normally incorporates the accompanying advances:

1. Building your model.
2. Building a model of the design.
3. Testing the model in the field.
4. Evaluating the aftereffects of the field tests.
5. Modifying the design dependent on the field test results.

This procedure proceeds until an agreeable arrangement is reached. Investigation can assist you with achieving the accompanying assignments:

•	Reduce cost by reenacting the testing of your model on the computer rather than costly field tests.

•	Reduce time to showcase by decreasing the quantity of item advancement cycles.

•	Improve items by rapidly testing numerous ideas and situations before settling on an official conclusion, giving you more opportunity to consider new designs".

Basic Concepts of Analysis:

The product utilizes the Finite Element Method (FEM). FEM is a numerical method for breaking down engineering designs. FEM is acknowledged as the standard examination technique because of its all inclusive statement and appropriateness for computer usage. FEM isolates the model into numerous little bits of basic shapes called components viably supplanting an unpredictable issue by numerous basic issues that should be understood at the same time.

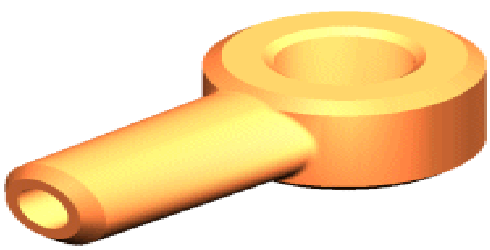


Fig 3.12(a)-CAD model of a part

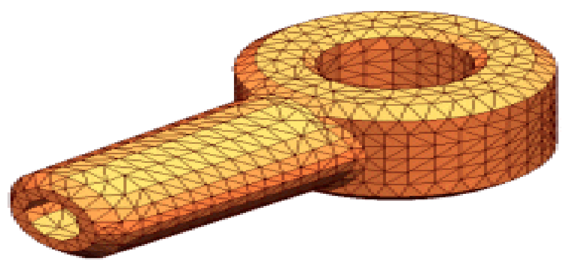


Fig 3.12(b) Model subdivided into small pieces (elements)

Components share regular focuses called hubs. The way toward partitioning the model into little pieces is called coinciding. The conduct of every component is notable under all conceivable help and burden situations. The limited component strategy utilizes components with various shapes. The reaction anytime in a component is introduced from the reaction at the component hubs. Every hub is completely portrayed by various boundaries relying upon the examination type and the component utilized. For instance, the temperature of a hub completely depicts its reaction in warm examination. For auxiliary examinations, the reaction of a hub is portrayed, as a rule, by three interpretations and three turns. These are called degrees of opportunity (DOFs). Investigation utilizing FEM is called Finite Element Analysis (FEA).

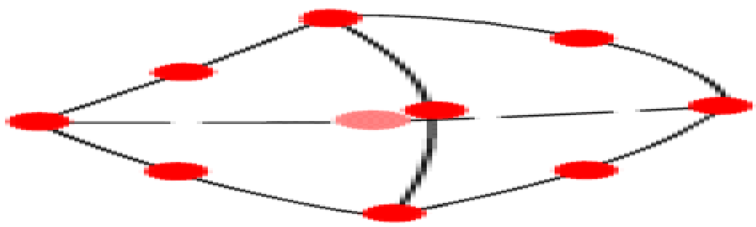


Figure 3.13: Nodes

A tetrahedral component. Red dabs speak to hubs. Edges of a component can be bended or straight.

The product defines the conditions overseeing the conduct of every component contemplating its availability to different components. These conditions relate the reaction to known material properties, restrictions, and burdens.

Next, the program composes the conditions into a huge arrangement of synchronous logarithmic conditions and illuminates for the questions.

In stress examination, for instance, the solver finds the relocations at every hub and afterward the program figures strains lastly stresses.

The product offers the accompanying sorts of studies:

Table 3.2: Software Icon Info

Study type	Study icon		
Static		Modal Time History	
Frequency		Harmonic	
Buckling		Random Vibration	
Thermal		Response Spectrum	
Design Study		Drop Test	
Nonlinear Static		Fatigue	
Nonlinear Dynamic		Pressure Vessel Design	

3.8 Analysis Steps:

The means expected to play out an investigation rely upon the examination type. You complete an examination by playing out the accompanying advances:

- Create an investigation characterizing its examination type and alternatives.
- If required, characterize boundaries of your examination. A boundary can be a model measurement, material property, power esteem, or some other information.
- Define material properties.
- Specify restrictions and burdens.
- The program consequently makes a blended work when various geometries (strong, shell, auxiliary individuals and so forth.) exist in the model.
- Define part contact and contact sets.
- Mesh the model to separate the model into numerous little pieces called components. Weakness and advancement considers utilize the cross sections in referenced investigations.

- Run the investigation.
- View results.

Specific capabilities of ansys Simulation:

1. Static Analysis:

At the point when burdens are applied to a body, the body disfigures and the impact of burdens is transmitted all through the body. The outer burdens initiate interior powers and responses to render the body into a condition of harmony. Straight Static investigation figures relocations, strains, stresses, and response powers under the impact of applied burdens.

2. Thermal Stress Analysis:

Changes in temperature can incite considerable disfigurements, strains, and stresses. Warm stress investigation alludes to static examination that incorporates the impact of temperature.

Perform warm stress examination utilizing one of the accompanying alternatives:

- Using a uniform ascent or drop in temperature for the entire model.
- Using a temperature profile coming about because of a consistent state or transient warm examination.
- Using a temperature profile from Flow Simulation.

3. Frequency examination:

“In the event that the design is exposed to dynamic situations, static investigations can't be utilized to assess the reaction. Recurrence studies can assist you with staying away from reverberation and design vibration seclusion frameworks. They likewise structure the reason for assessing the reaction of direct powerful frameworks where the reaction of a framework to a unique domain is thought to be equivalent to the summation of the commitments of the modes considered in the examination.

4. Dynamic investigation:

Dynamic investigation include:

Design auxiliary and mechanical frameworks to perform without disappointment in powerful situations. Adjust framework's attributes (i.e., geometry, damping systems, material properties, and so forth.) to diminish vibration impacts.

5. Buckling investigation:

Used to ascertain the clasping loads and decide the clasping mode shape. Both direct (Eigen esteem) clasping and nonlinear clasping examinations are conceivable.

6. Non-straight static investigation:

Every genuine structure carry on nonlinearly somehow at some degree of stacking. Now and again, direct investigation might be satisfactory. In numerous different cases, the direct arrangement can deliver wrong outcomes on the grounds that the suspicions whereupon it is based are disregarded. Nonlinearity can be brought about by the material conduct, enormous relocations, and contact conditions. We can utilize a nonlinear report to take care of a straight issue. The outcomes can be marginally extraordinary because of various strategies. In the nonlinear static investigation, dynamic impacts like inertial and damping powers are not thought of.

7. Drop test examines:

Drop test considers assess the impact of the effect of a section or a gathering with an unbending or adaptable planar surface. Dropping an article on the floor is a normal application and consequently the name. The program ascertains effect and gravity stacks naturally. No different burdens or limitations are permitted.

8. Fatigue Analysis :

Weariness is the prime reason for the disappointment of numerous items, particularly those made of metals. Instances of disappointment because of weakness incorporate, turning hardware, jolts, plane wings, purchaser items, seaward stages, ships, vehicle axles, extensions, and bones.

Direct and nonlinear auxiliary examinations don't anticipate disappointment because of weakness. They compute the reaction of a design exposed to a predefined domain of restrictions and burdens. On the off chance that the examination suppositions are watched and the determined stresses are inside as far as possible, they infer that the design is sheltered in this condition paying little heed to how frequently the heap is applied.

Consequences of static, nonlinear, or time history straight powerful examinations can be utilized as the reason for characterizing an exhaustion study. The quantity of cycles required for exhaustion inability to happen at an area relies upon the material and the stress variances. This data, for a specific material, is given by a bend called the SN bend.

9. Pressure vessel Design study :

In a Pressure Vessel Design study, you consolidate the consequences of static investigations with the ideal elements. Every static investigation has an alternate arrangement of burdens that produce relating results. These heaps can be dead loads, live loads (approximated by static burdens), warm loads, seismic burdens, etc. The Pressure Vessel Design study joins the aftereffects of the static investigations mathematically utilizing a straight blend or the square base of the aggregate of the squares” (SRSS).

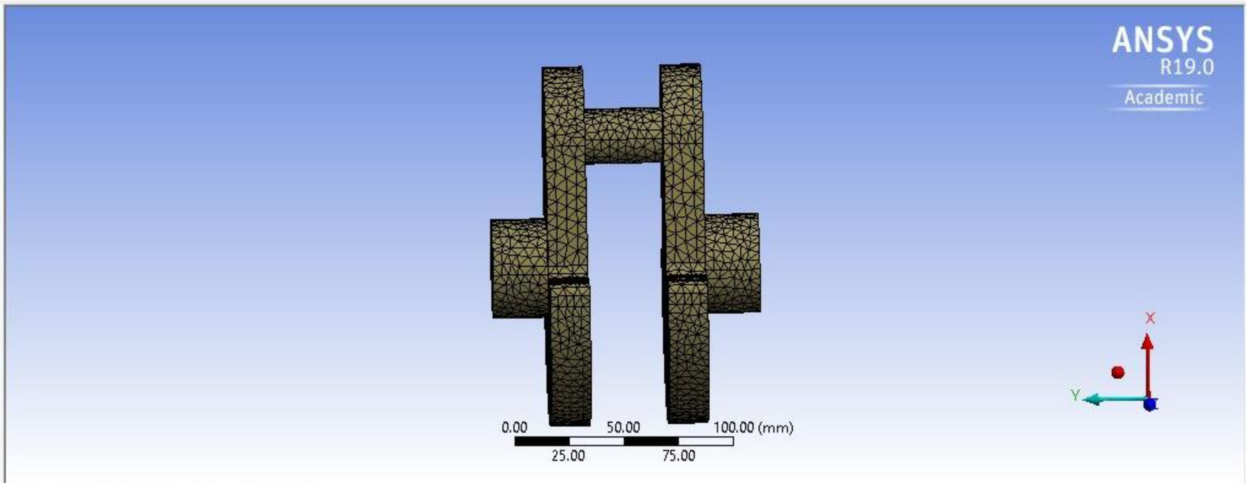


Fig.3.14 Meshing of crankshaft

3.9 Boundary condition

The FE model created was subjected to static structural analysis after assigning suitable material properties and boundary conditions.

The force acting on the crankpin for case-1 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 22624 N.

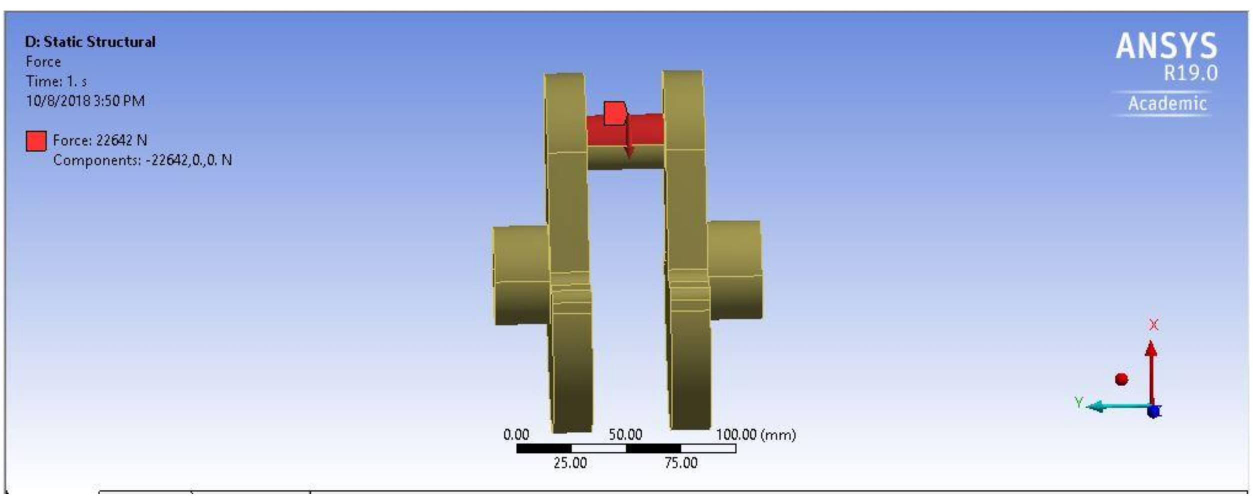


Fig.3.15 Applying force for case 1

The force acting on the crankpin for case-2 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 32624 N.

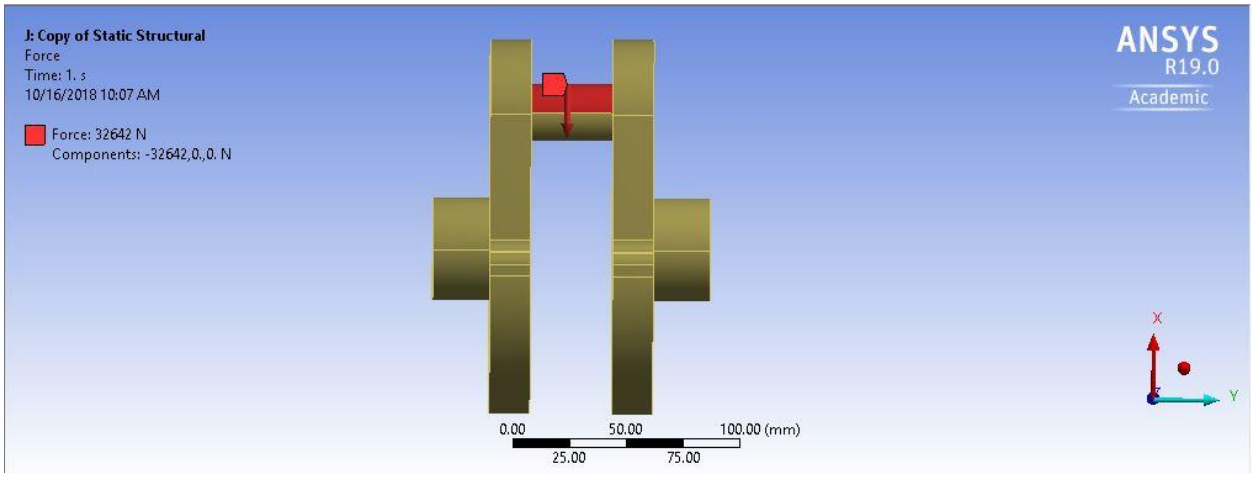


Fig.3.16 Applying force for case 2

The force acting on the crankpin for case-3 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 42624 N.

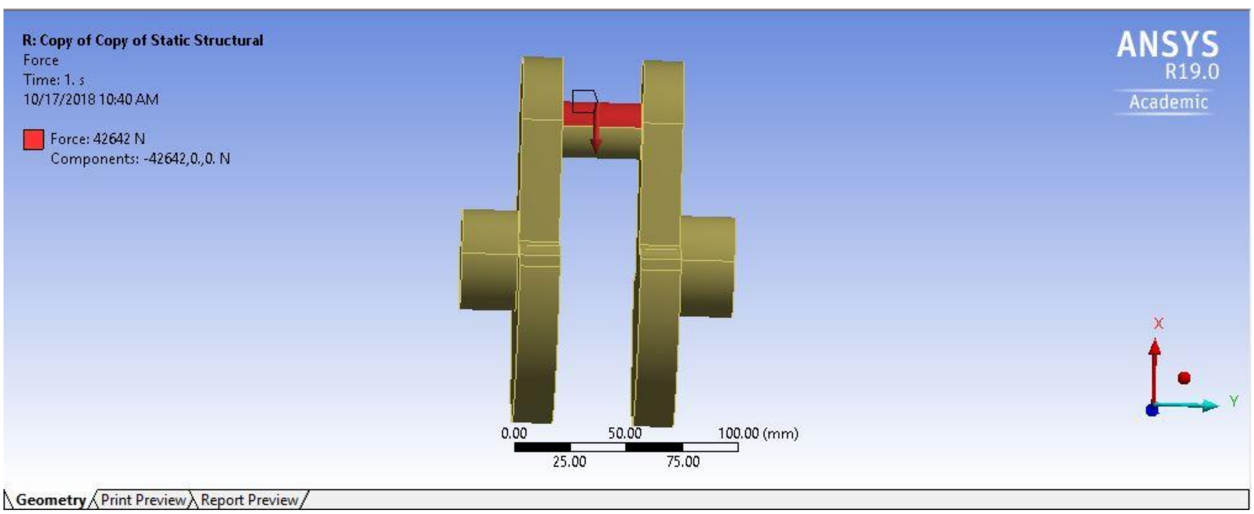


Fig.3.17 Applying force for case 3

Fixed support

The crankshaft is fixed in both the end of shaft.

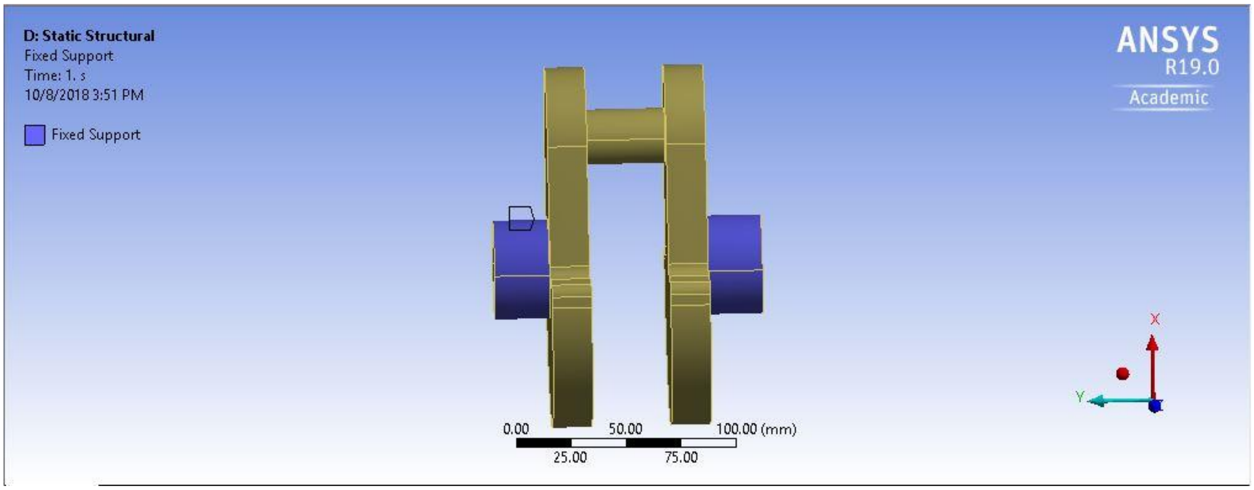


Fig.3.18 Fixed support of crankshaft

3.10 TOPOLOGY OPTIMIZATION

The ANSYS 19.0 provide the feature of topology optimization. After applying 22624N force the new model is proposed. The red region show that this material is does not affect the all over stress value. So all the new design is made only change in red region.

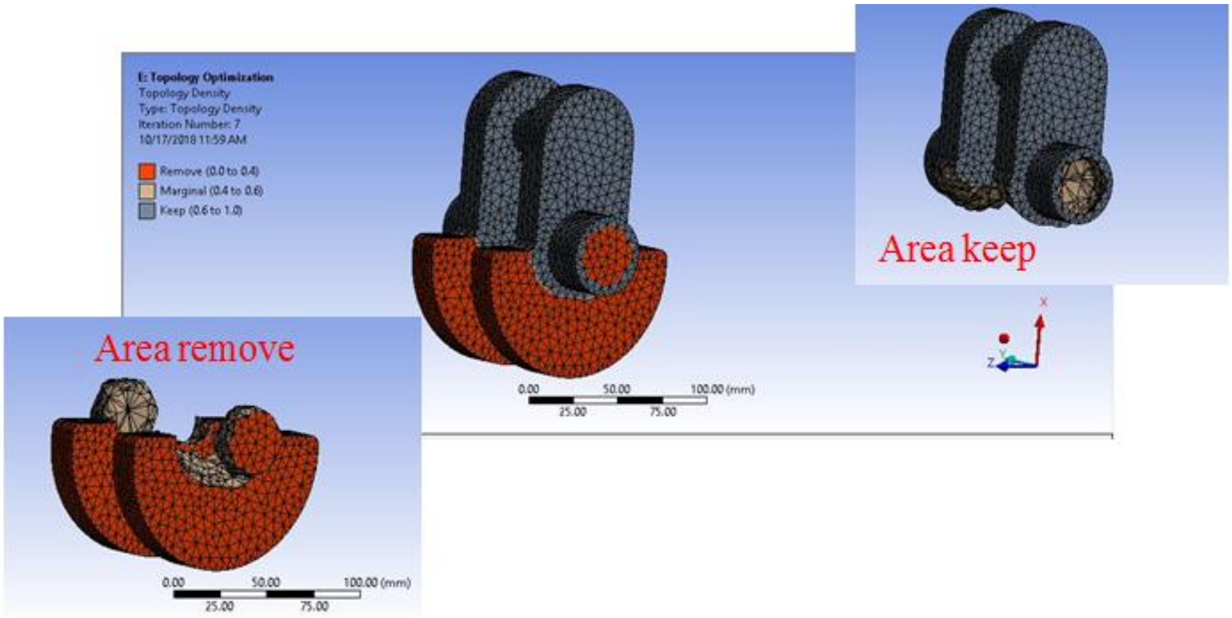


Fig.3.19 Topology optimization in ANSYS

CHAPTER 4

SIMULATION RESULT

The topology optimization in Crankshaft is help to reduce the net mass of crankshaft in initial design. This could lead to a better performance of the engine. While achieving this reduction in mass, necessary precaution is to be taken to ensure the factor of safety is still in the industry standards. The innovative and High Performance Calculating with ANSYS Topology Optimization designed an improved result than the conceptual designs. In the simulation result there are three parameters are measured and three cases are consider for the simulation. The following parameters where measured during the experiment:-

- Stress
- Deformation
- Weight

The primary target of the analysis is to reduce the maximum amount of weight by considering stress and deformation value. In new design it is necessary the stress and deformation of crankshaft is not increases in basic design.

Case-1 In this case the maximum force acting on the crankpin is 22624 N.

(Model -1) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is shown in figure 4.1. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case the maximum deformation is 0.011156mm.

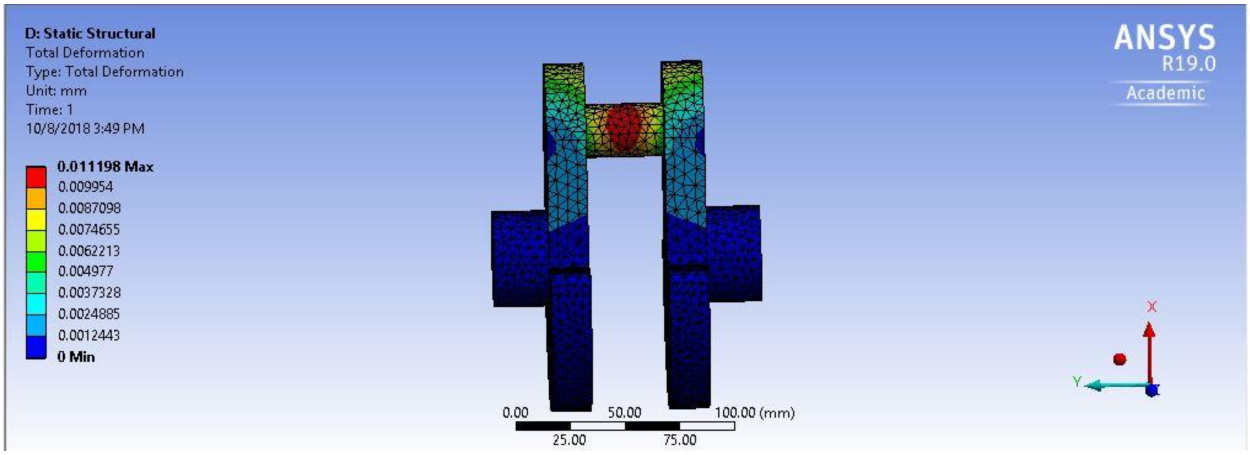


Fig.4.1 Total deformation in Case-1 for model 1

(Model-1) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress is present in top of journal which is presented in figure 4.2. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 68.42 Mpa maximum stress is obtained.

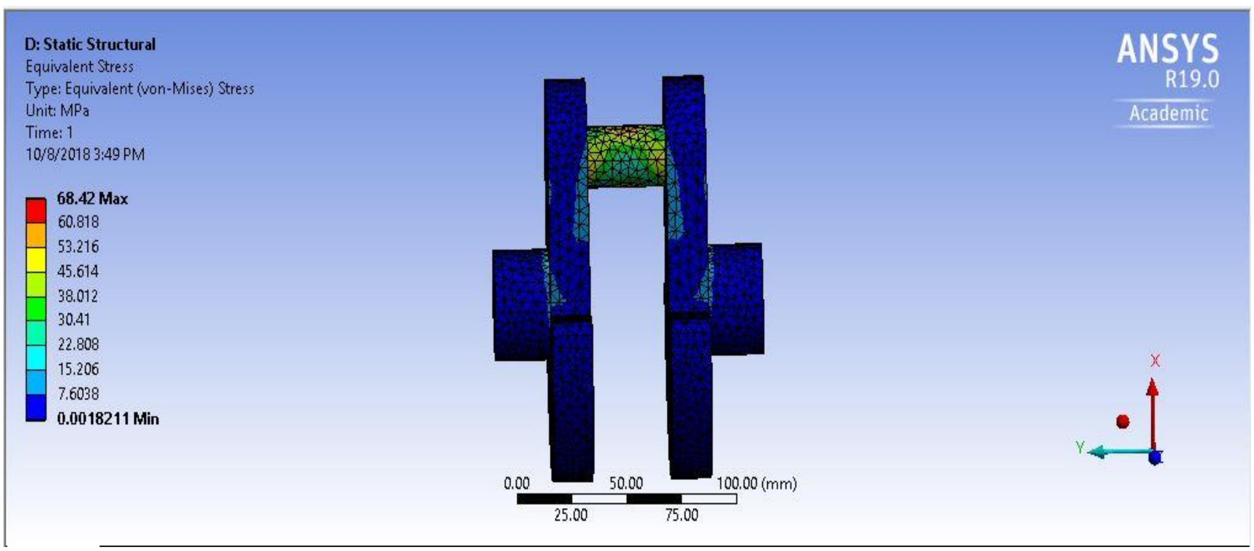


Fig.4.2 Equivalent stress in Case-1 for model 1

(Model-2) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is displayed in figure 4.3. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.011182mm deformation is found.

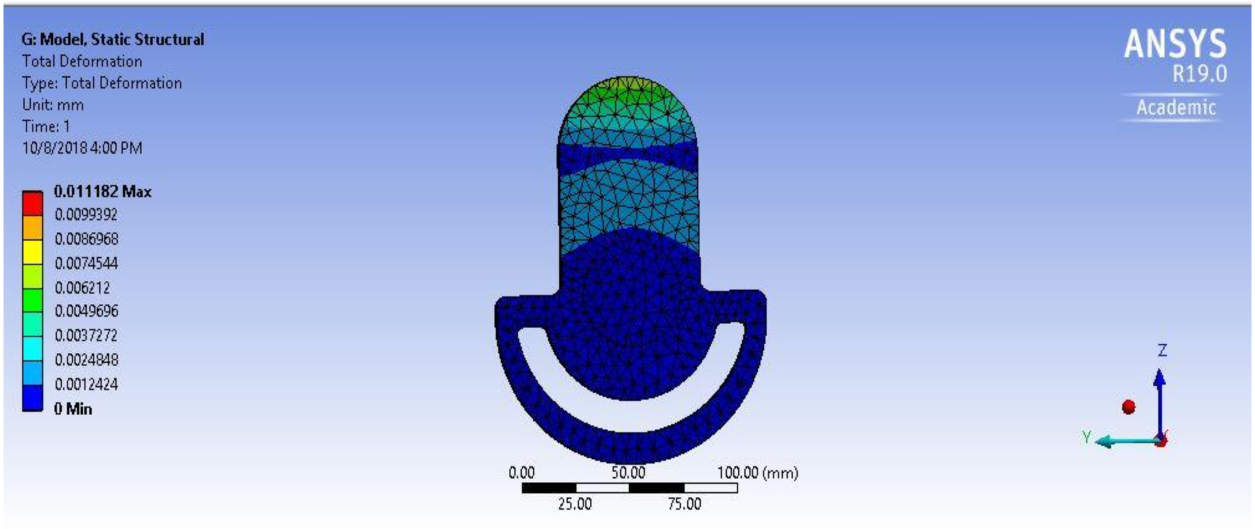


Fig.4.3 Total deformation in Case-1 for model 2

(Model-2) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress is present in top of journal which is displayed in figure 4.4. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 68.70 Mpa maximum stress is found.

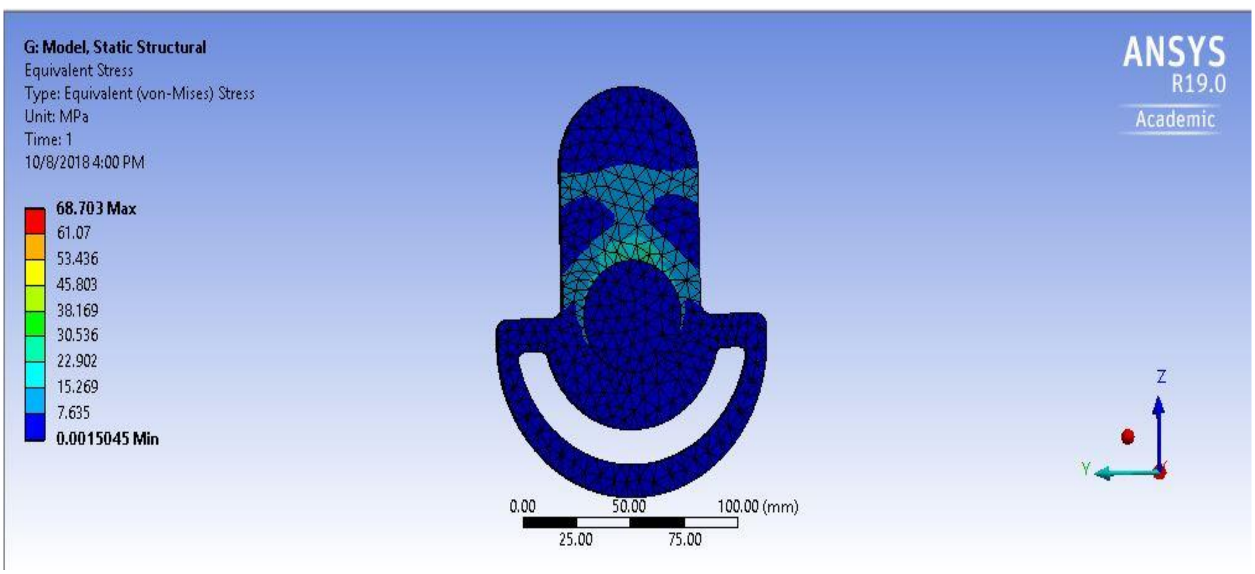


Fig.4.4 Equivalent stress in Case-1 for model 2

(Model-3) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is demonstrated in figure 4.5. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.011202mm deformation is found.

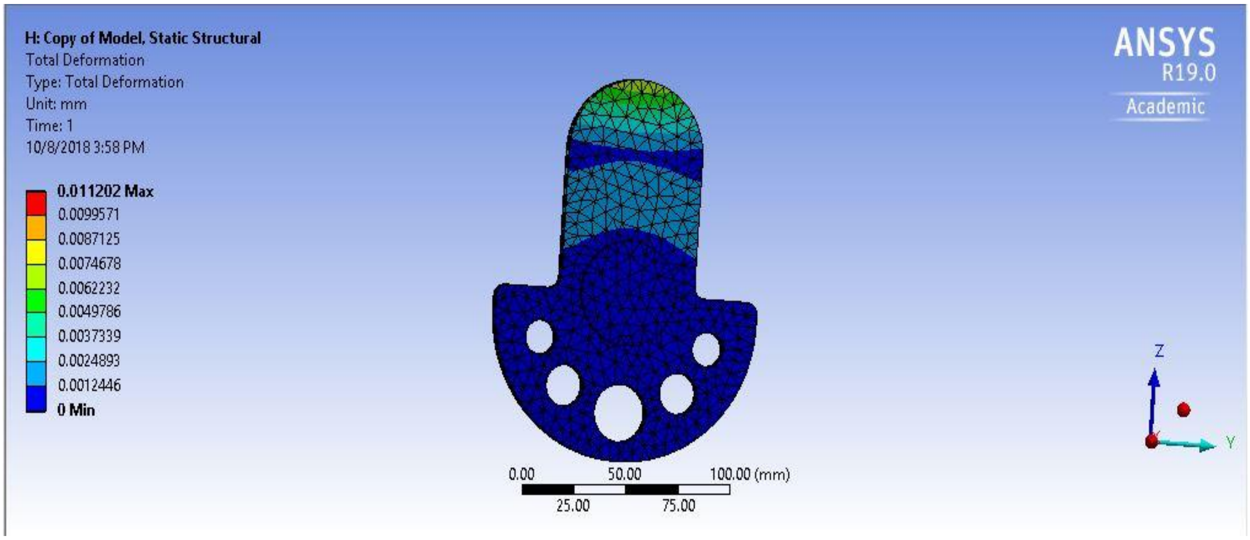


Fig.4.5 Total deformation in Case-1 for model 3

(Model-3) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress is present in top of journal which is elaborated in figure 4.6. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 63.49 Mpa maximum stress is found.

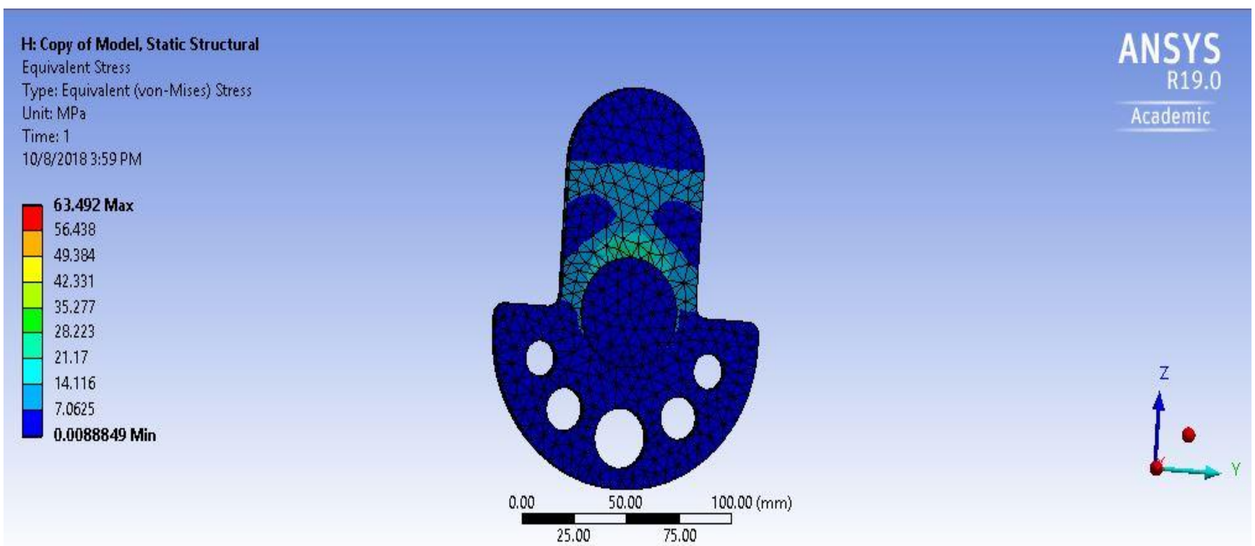


Fig.4.6 Equivalent stress in Case-1 for model 3

(Model-4) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is illustrated in figure 4.7. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.011207mm deformation is found.

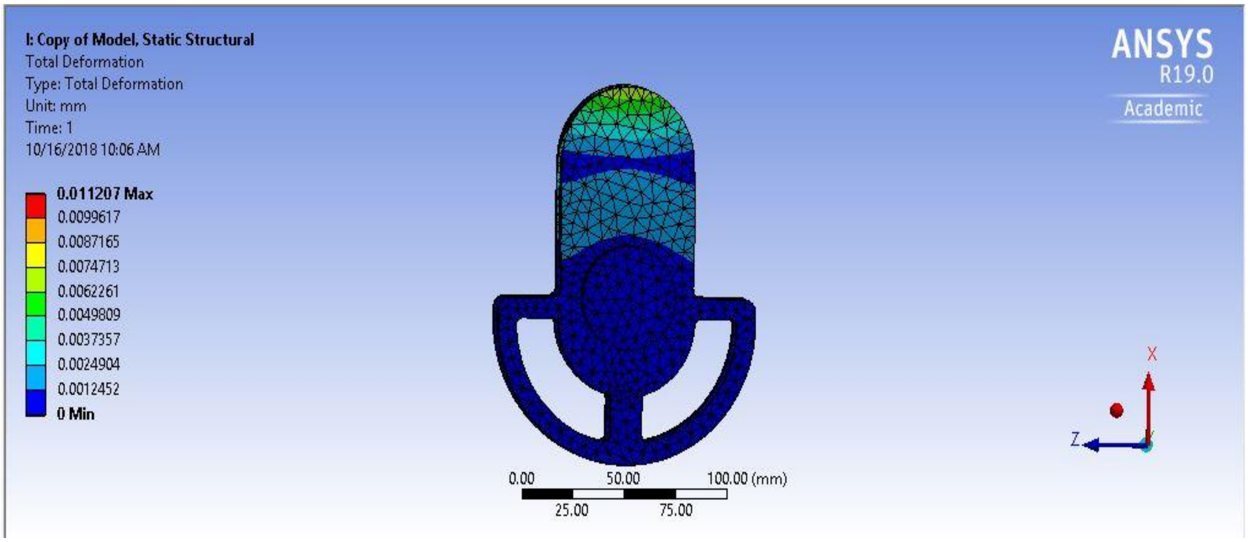


Fig.4.7 Total deformation in Case-1 for model 4

(Model-4) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress is present in top of journal which is displayed in figure 4.8. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 4 74.156 Mpa maximum stress is found.

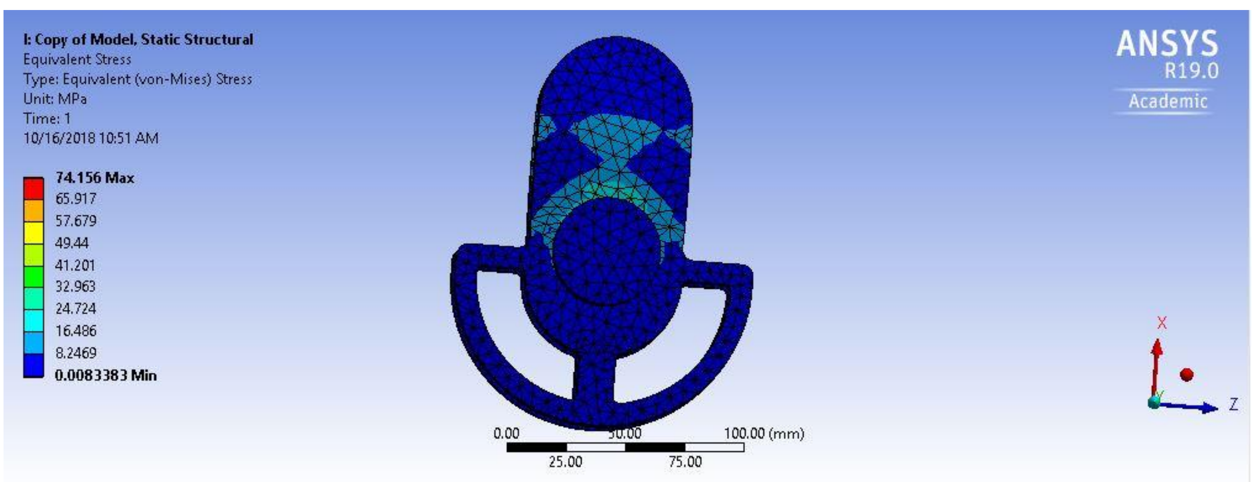


Fig.4.8 Equivalent stress in Case-1 for model 4

Case-2 In this case the maximum force acting on the crankpin is 32624 N.

(Model-1) Deformation: After applying 32624N force in crankshaft the maximum crankshaft is deformed in top of journal which is shown in figure 4.9. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.016144mm deformation is found.

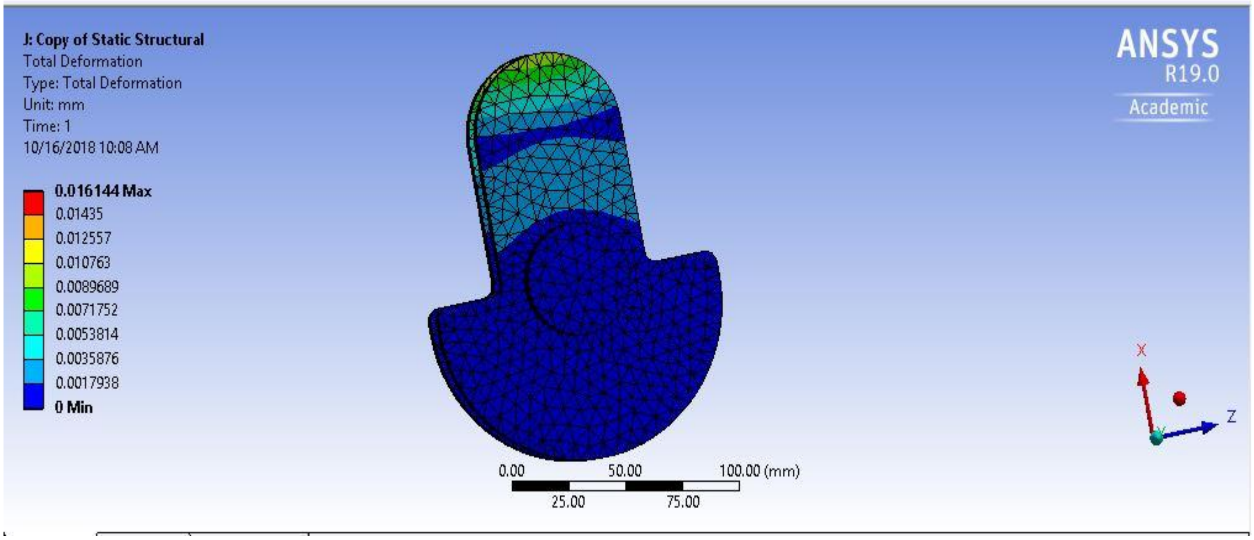


Fig.4.9 Total deformation in Case-2 for model 1

(Model-1) Equivalent Stress: After applying 32624N force in crankshaft the maximum stress is present in top of journal which is prsented in figure 4.10. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 98.68 Mpa maximum stress is found.

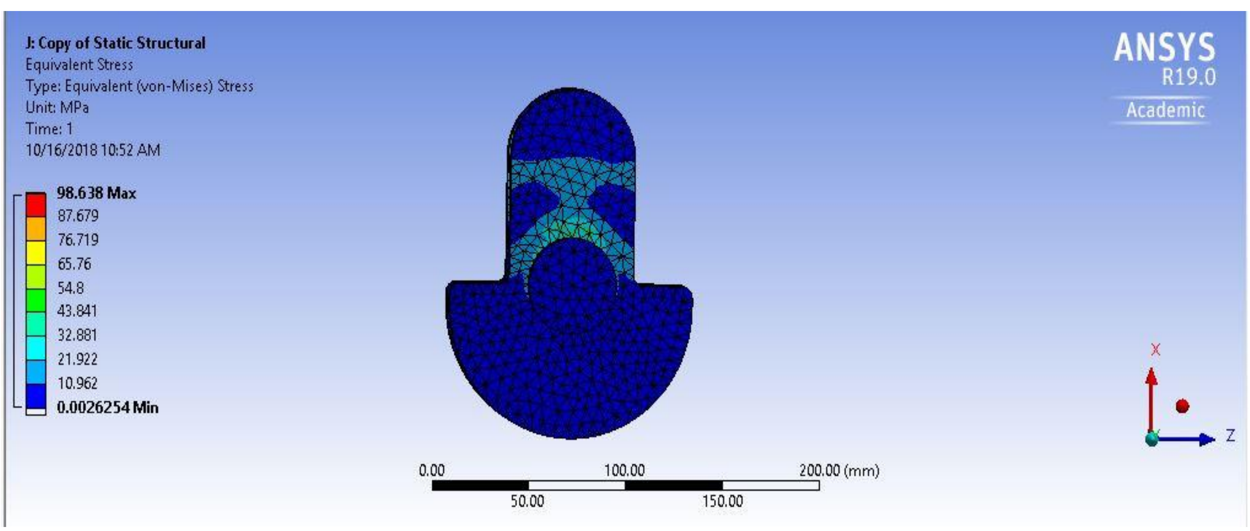


Fig.4.10 Equivalent stress in Case-2 for model 1

(Model-2) Deformation: After applying 32624N force in crankshaft the maximum crankshaft is deformed in top of journal which is elaborated in figure 4.11. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.016136mm deformation is found.

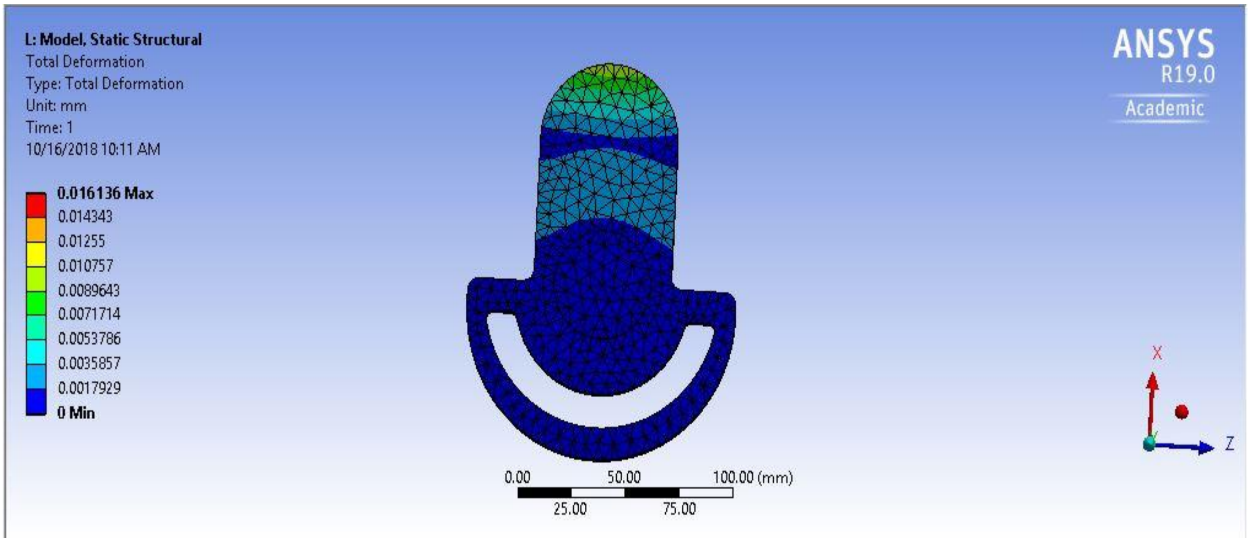


Fig.4.11 Total deformation in Case-2 for model 2

(Model-2) Equivalent Stress: After applying 32624N force in crankshaft the maximum stress is present in top of journal which is displayed in figure 4.12. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 108.57 Mpa maximum stress is found.

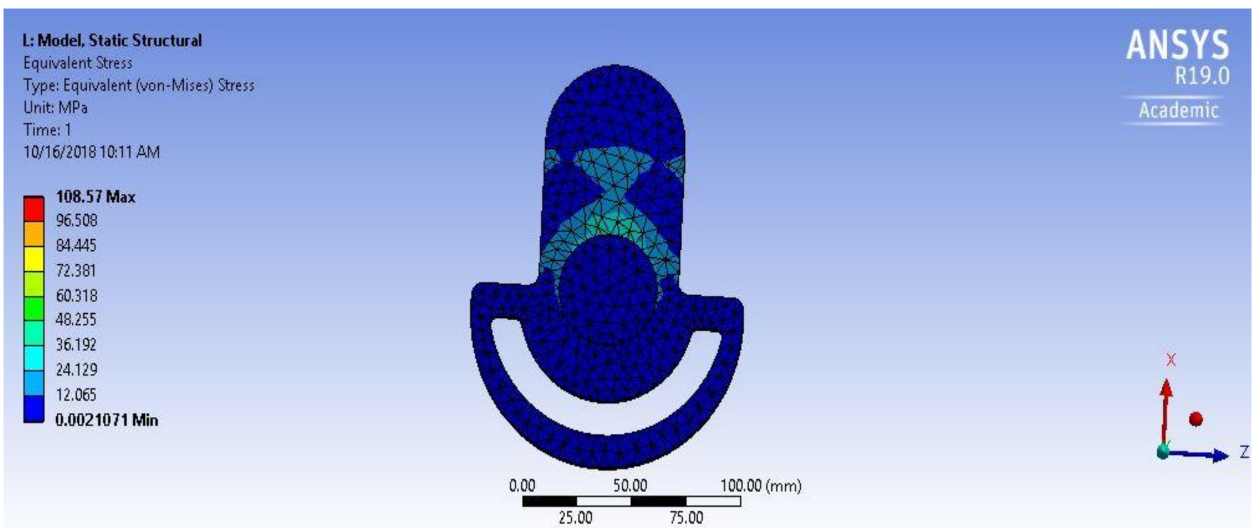


Fig.4.12 Equivalent stress in Case-2 for model 2

(Model-3) Deformation: After applying 32624N force in crankshaft the maximum crankshaft is deformed in top of journal which is exposed in figure 4.13. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.016149mm deformation is found.

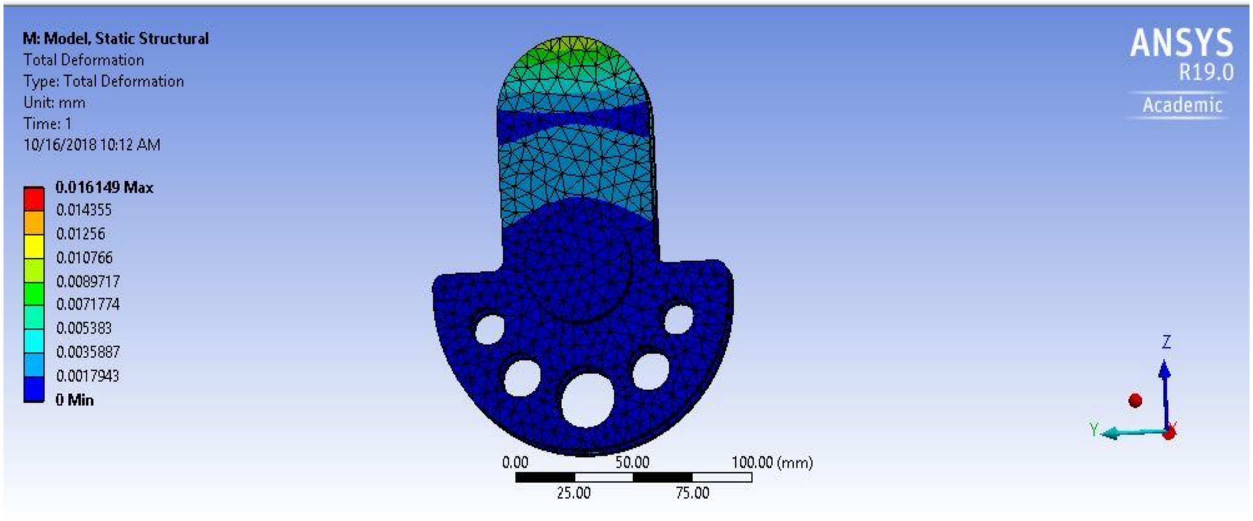


Fig.4.13 Total deformation in Case-2 for model 3

(Model-3) Equivalent Stress: After applying 32624N force in crankshaft the maximum stress is present in top of journal which is elaborated in figure 4.14. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 91.533 Mpa maximum stress is found.

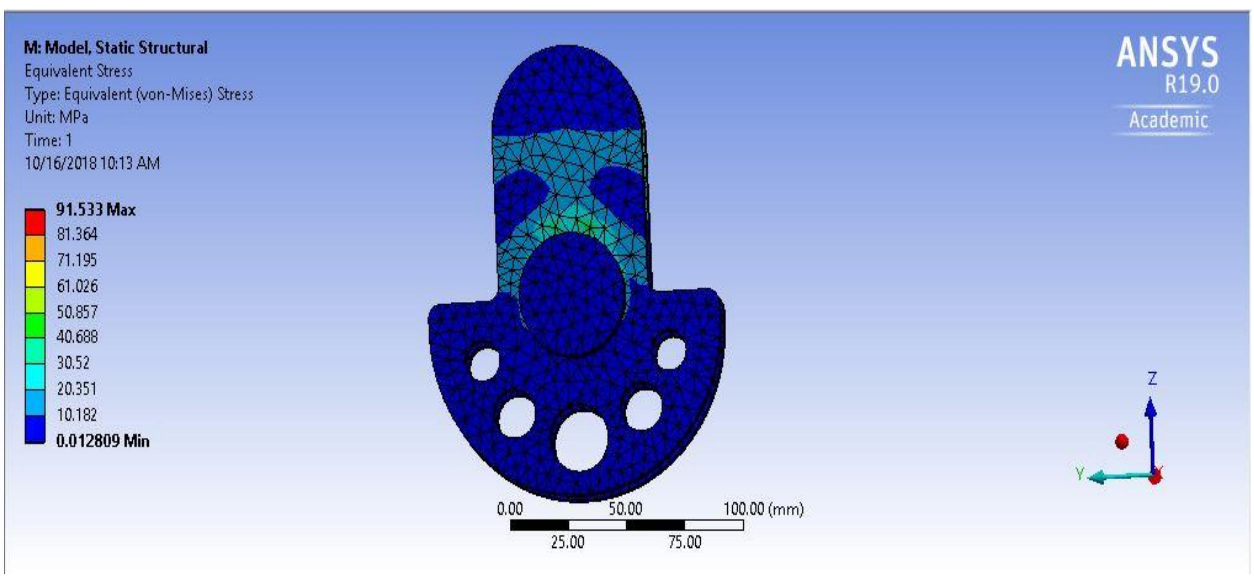


Fig.4.14 Equivalent stress in Case-2 for model 3

(Model-4) Deformation: After applying 32624N force in crankshaft the maximum crankshaft is deformed in top of journal which is demonstrated in figure 4.15. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.057078mm deformation is found.

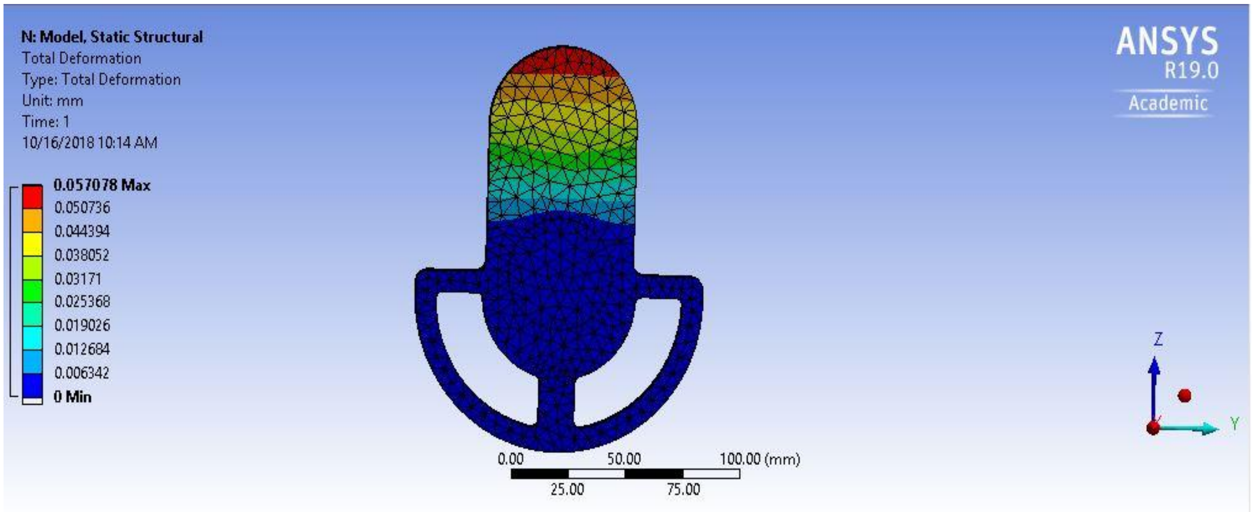


Fig.4.15 Total deformation in Case-2 for model 4

(Model-4) Equivalent Stress: After applying 32624N force in crankshaft the maximum stress is present in top of journal which is illustrated in figure 4.16. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 176.84 Mpa maximum stress is found.

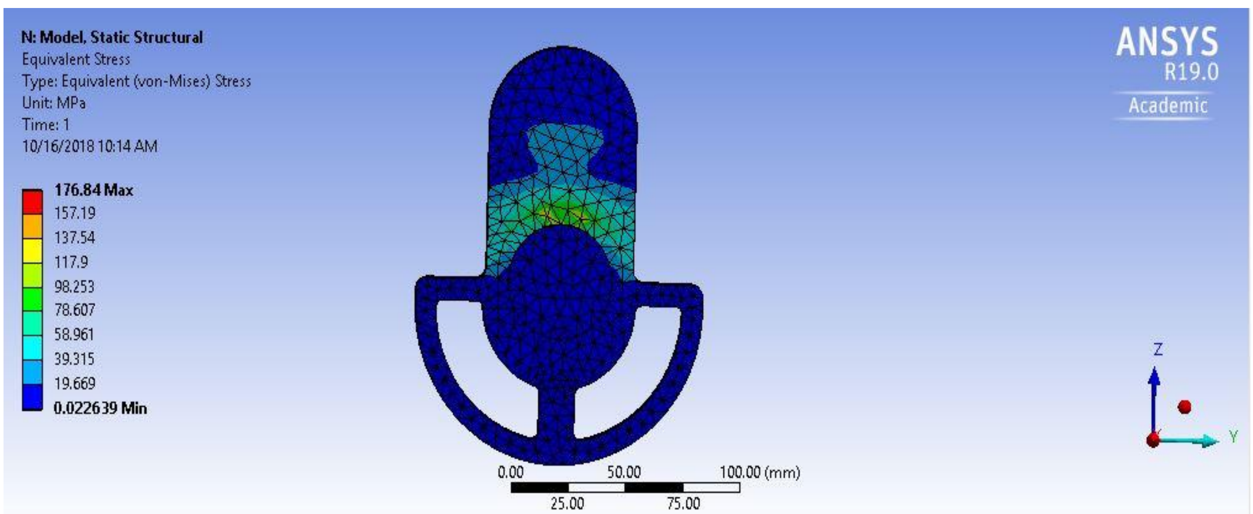


Fig.4.16 Equivalent stress in Case-2 for model 4

Case-3 In this case the maximum force acting on the crankpin is 42624 N.

(Model-1) Deformation: After applying 42624N force in crankshaft the maximum crankshaft is deformed in top of journal which is displayed in figure 4.17. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.02109mm deformation is found.

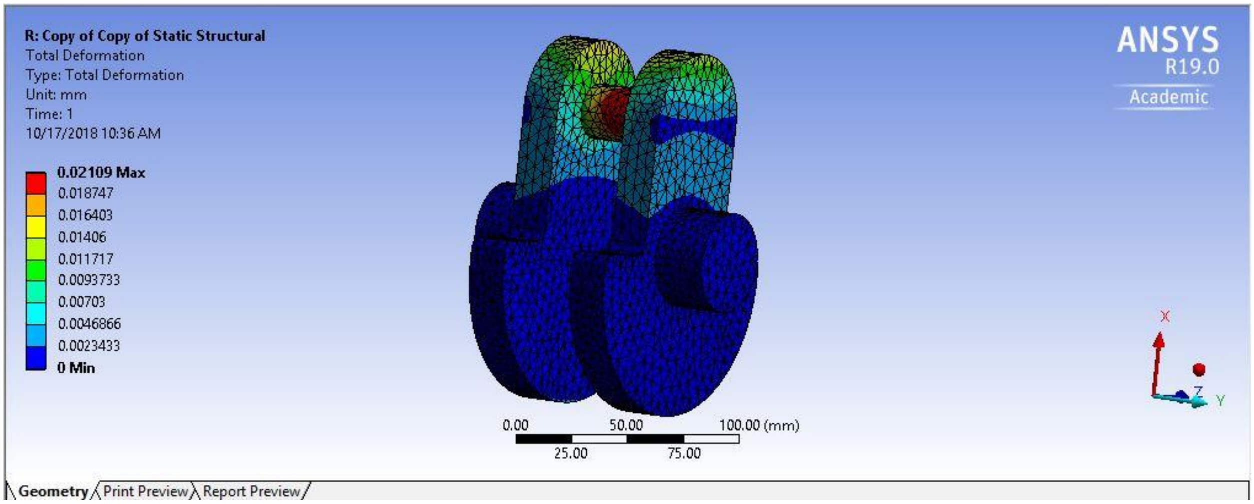


Fig.4.17 Total deformation in Case-3 for model 1

(Model-1) Equivalent Stress: After applying 42624N force in crankshaft the maximum stress is present in top of journal which is presented in figure 4.18. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 128.86 Mpa maximum stress is found.

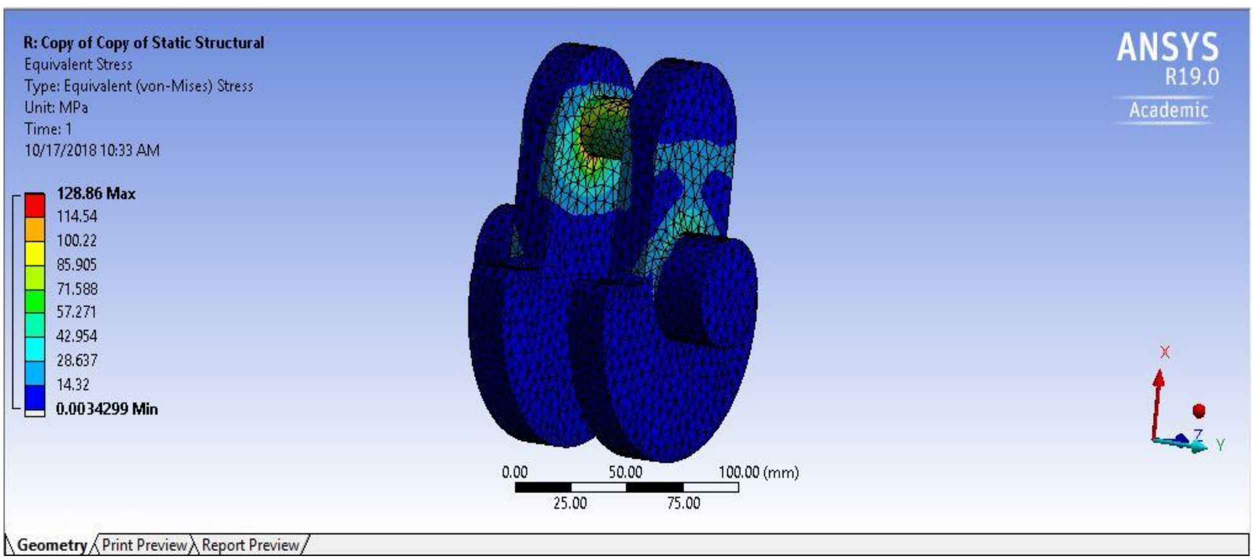


Fig.4.18 Equivalent stress in Case-3 for model 1

(Model-2) Deformation: After applying 42624N force in crankshaft the maximum crankshaft is deformed in top of journal which is demonstrate in figure 4.19. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.021079mm deformation is found.

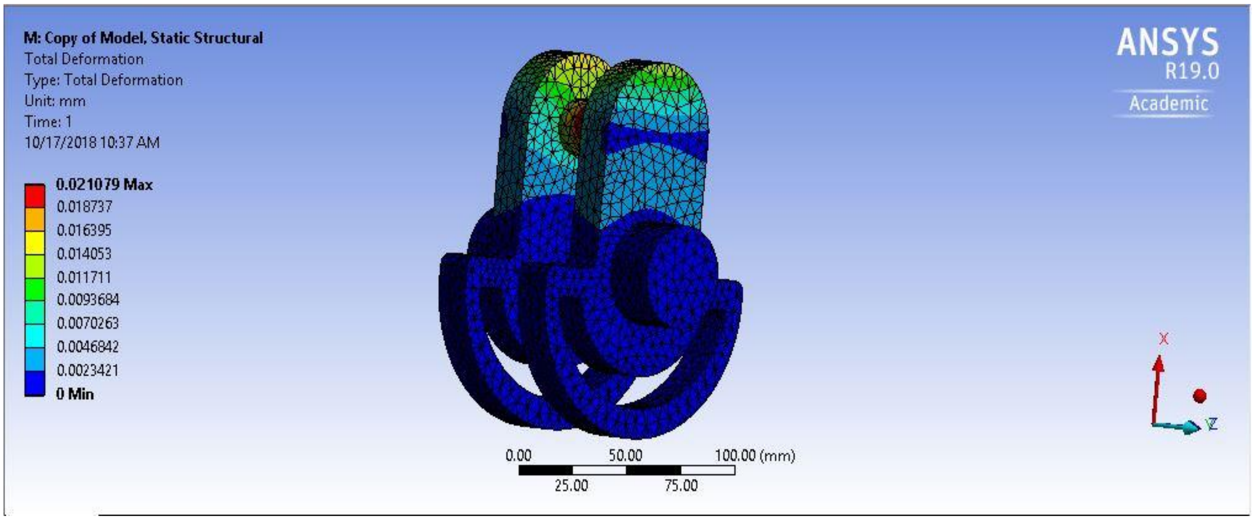


Fig.4.19 Total deformation in Case-3 for model 2

(Model-2) Equivalent Stress: After applying 42624N force in crankshaft the maximum stress is present in top of journal which is exposed in figure 4.20. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 148.83 Mpa maximum stress is found.

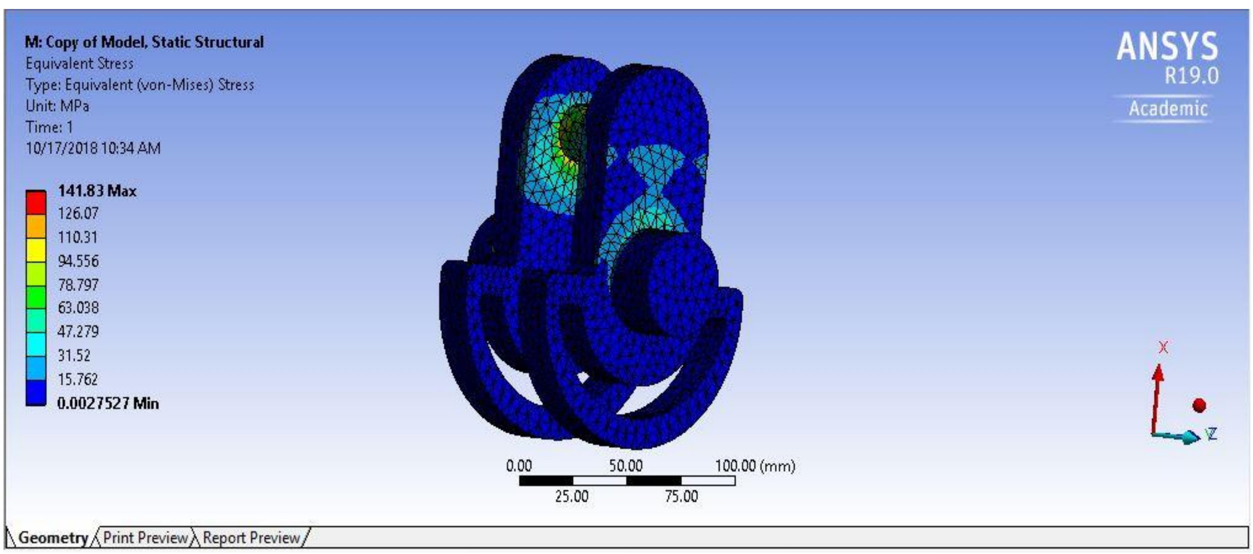


Fig.4.20 Equivalent stress in Case-3 for model 2

(Model-3) Deformation: After applying 42624N force in crankshaft the maximum crankshaft is deformed in top of journal which is elaborate in figure 4.21. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.021096mm deformation is found.

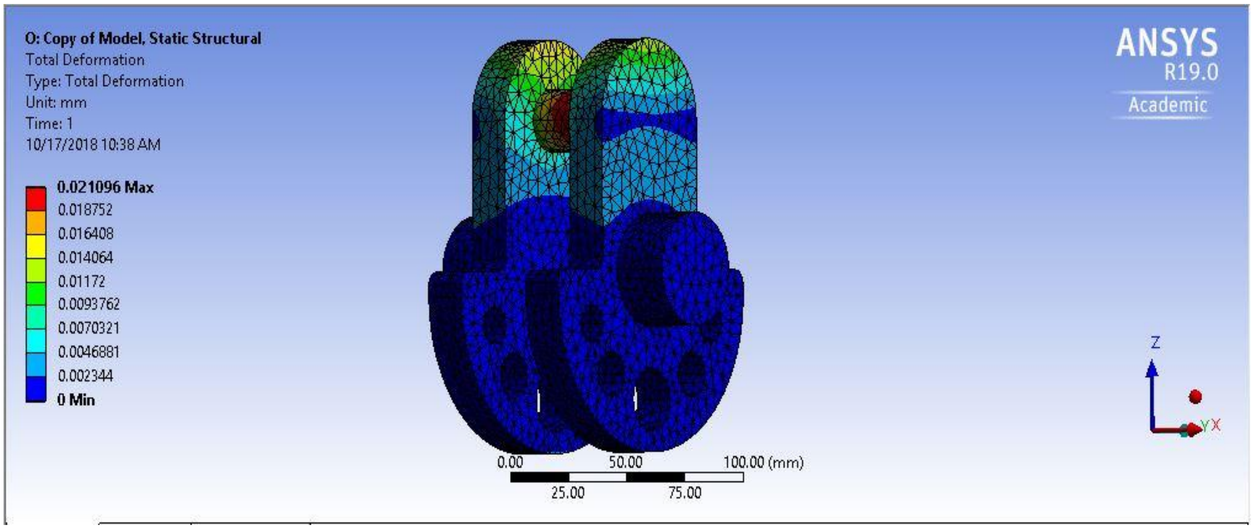


Fig.4.21 Total deformation in Case-3 for model 3

(Model-3) Equivalent Stress: After applying 42624N force in crankshaft the maximum stress is present in top of journal which is illustration in figure 4.22. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 119.57 Mpa maximum stress is found.

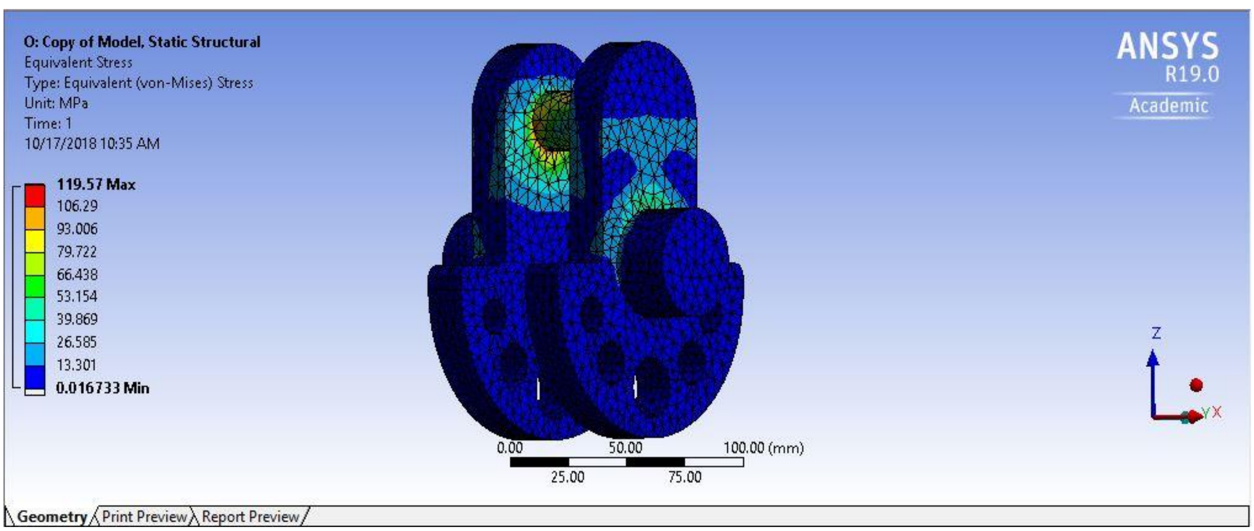


Fig.4.22 Equivalent stress in Case-3 for model 3

(Model-4) Deformation: After applying 42624N force in crankshaft the maximum crankshaft is deformed in top of journal which is displayed in figure 4.23. In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.074564mm deformation is found.

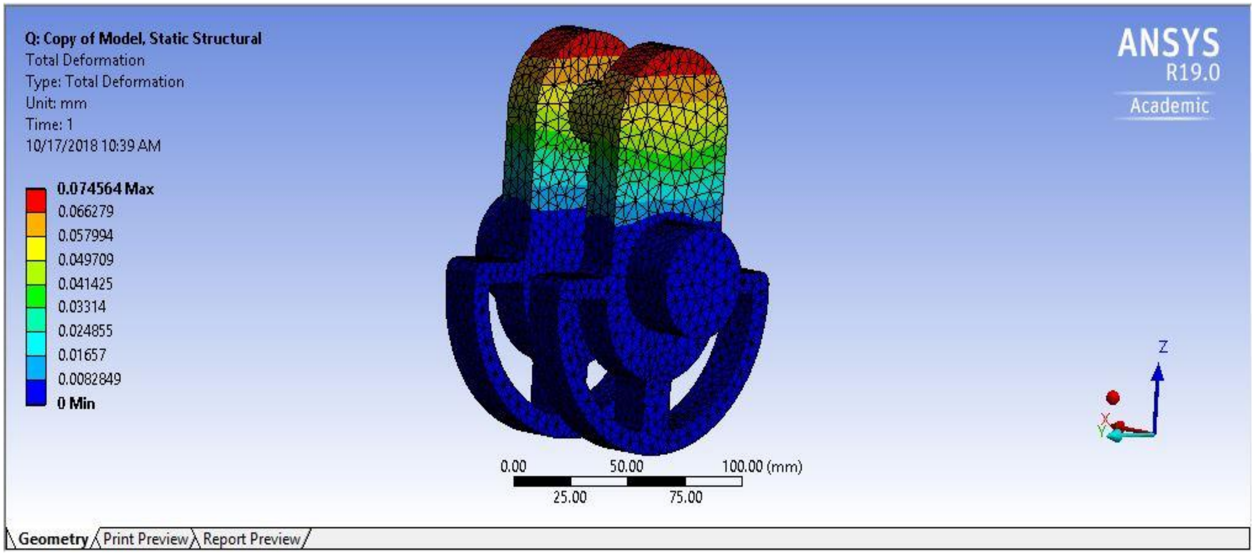


Fig.4.23 Total deformation in Case-3 for model 4

(Model-4) Equivalent Stress: After applying 42624N force in crankshaft the maximum stress is present in top of journal which is presented in figure 4.24. In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 231.01 Mpa maximum stress is found.

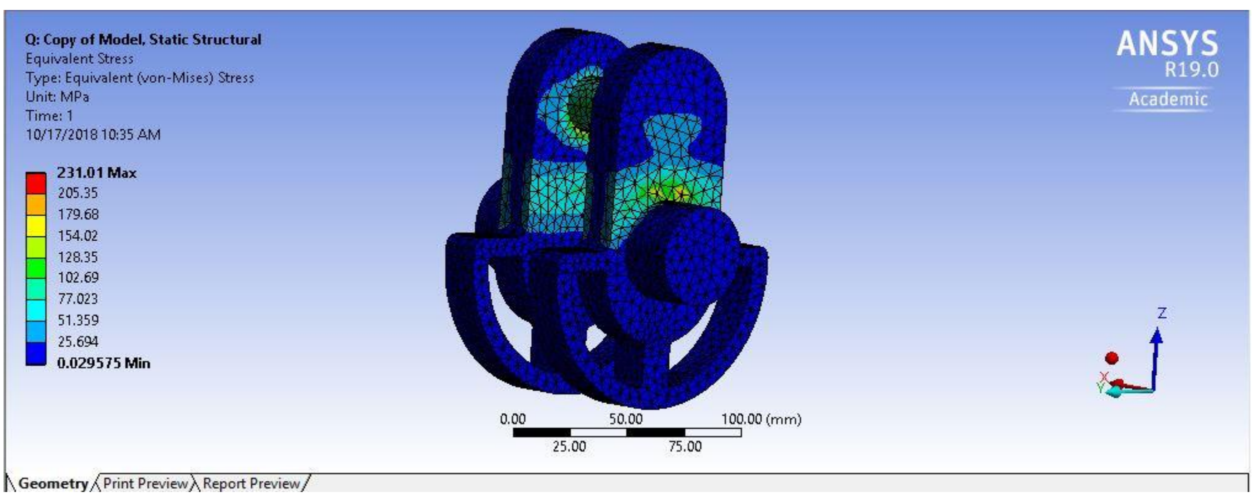


Fig.4.24 Equivalent stress in Case-3 for model 4

CHAPTER 5

STRUCTURAL ANALYSIS OF CRANK SHAFT

5.1 Analysis Results:

The analysis parameter is shown in table 5.1.

Table 5.1: Properties of material

Material	Density(g/cm^3)	Youngs modulus(Gpa)	Poissions ratio
Aluminum alloy	2.6898	68.3	0.34
Titanium alloy	4.62	96	0.36
Magnesium alloy	1.8	45	0.35

In figure 5.1 to 5.13 analysis of model screenshots are shown

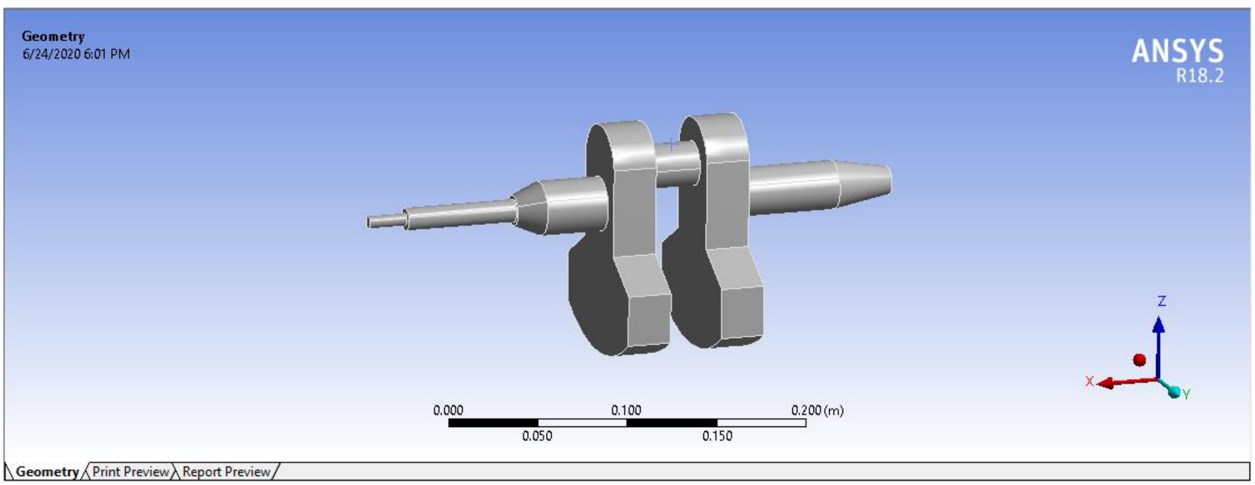


Figure 5.1: Model

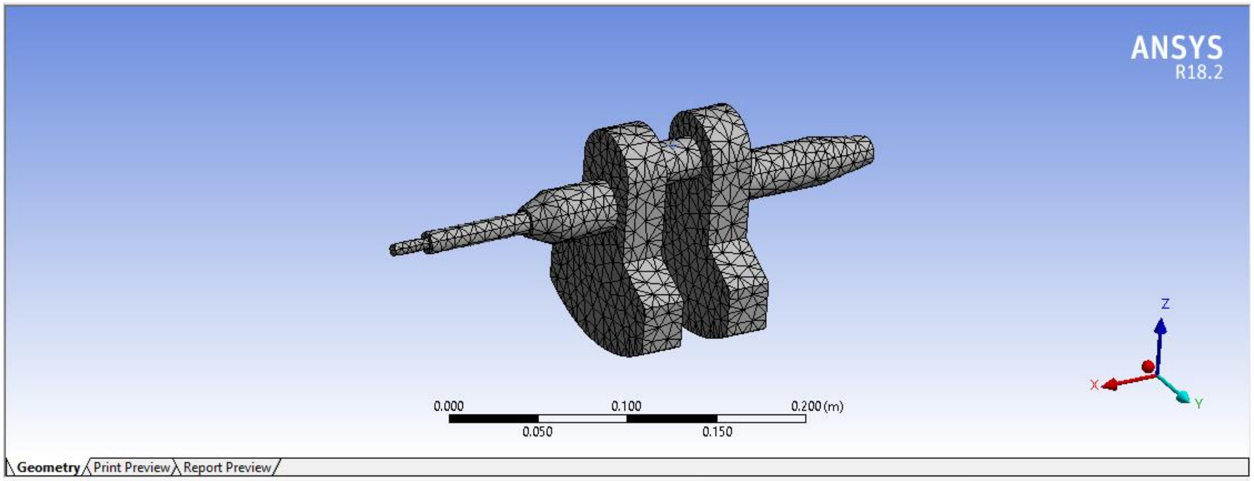


Figure 5.2: Mesh

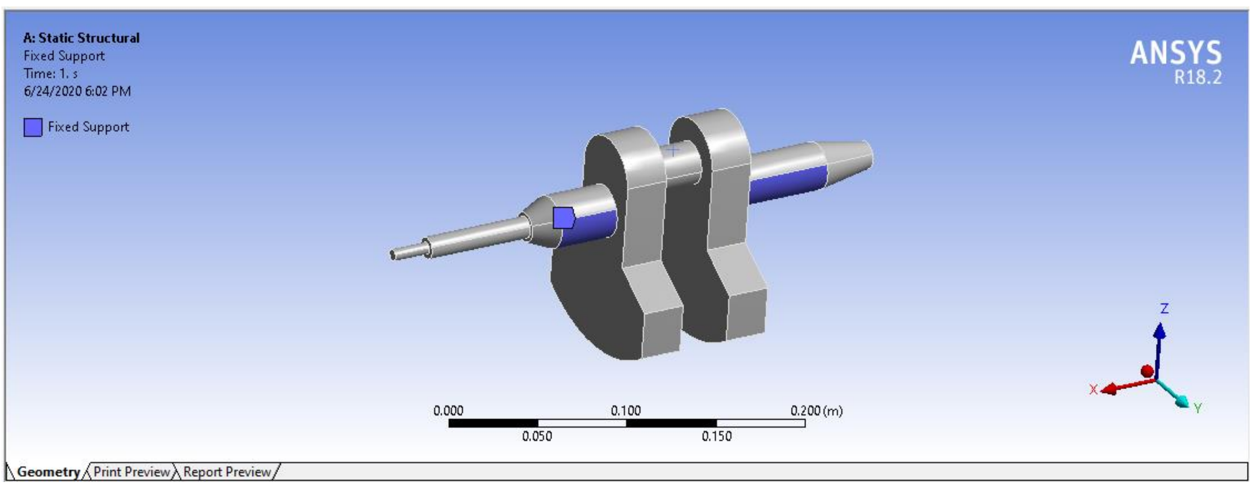


Figure 5.3: Fixed support

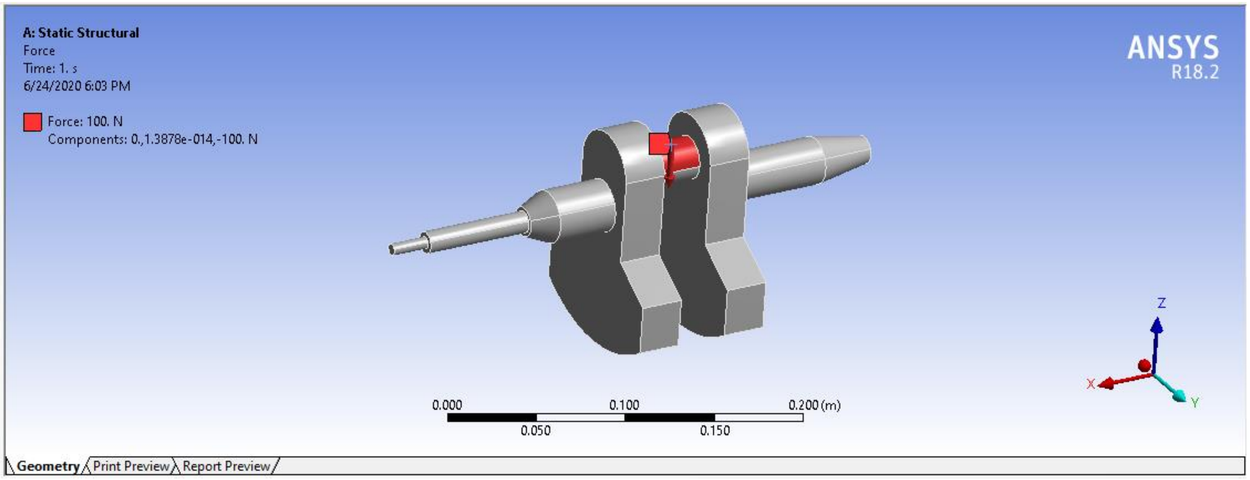


Figure 5.4: Load –force 100 N

5.2 For Aluminum alloy

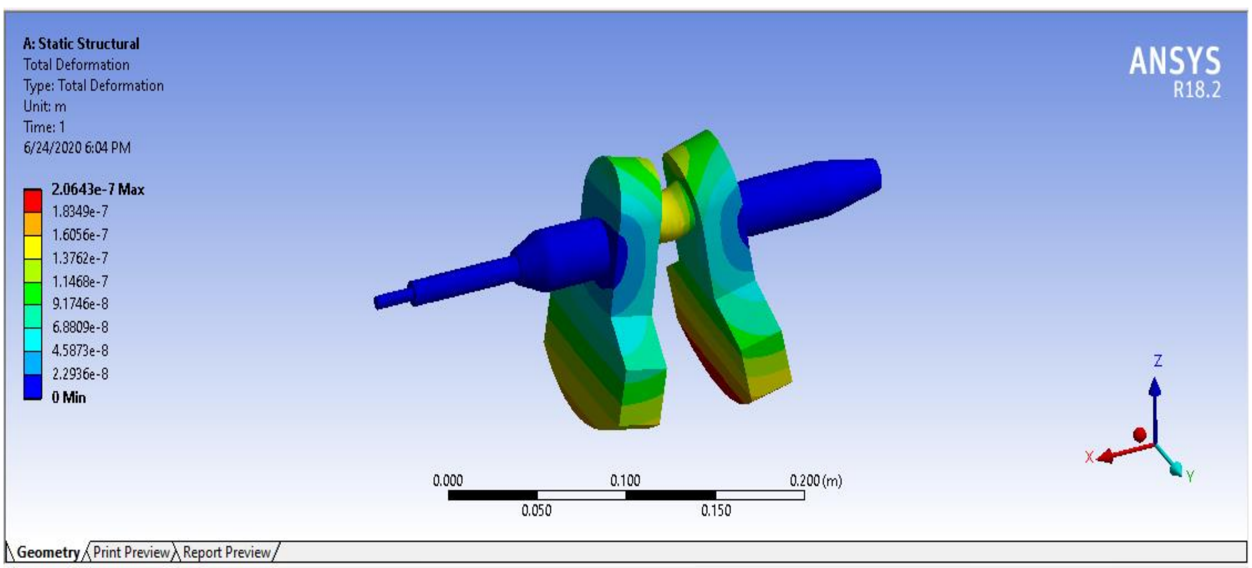


Figure 5.5: Total deformation

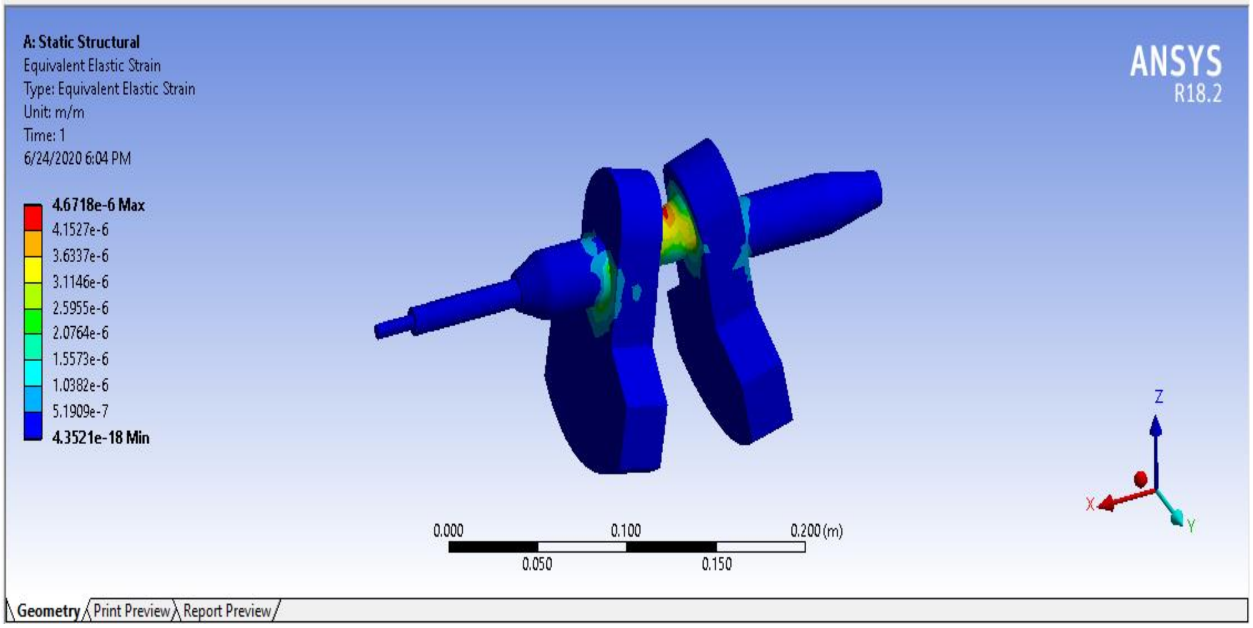


Figure 5.6: Strain

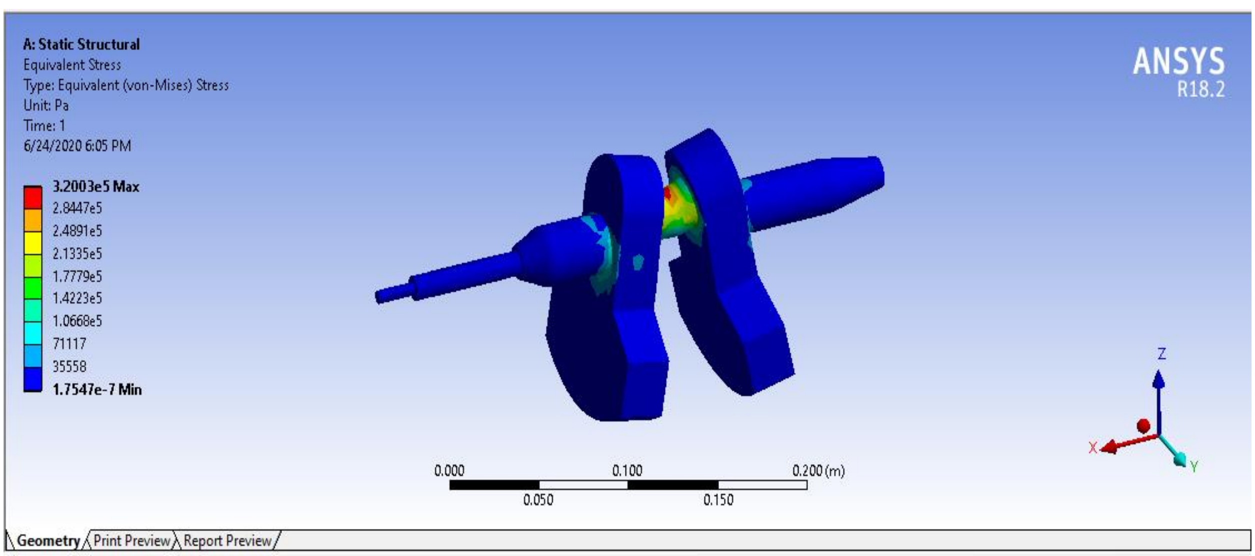


Figure 5.7: Stress

5.3 for Titanium alloy

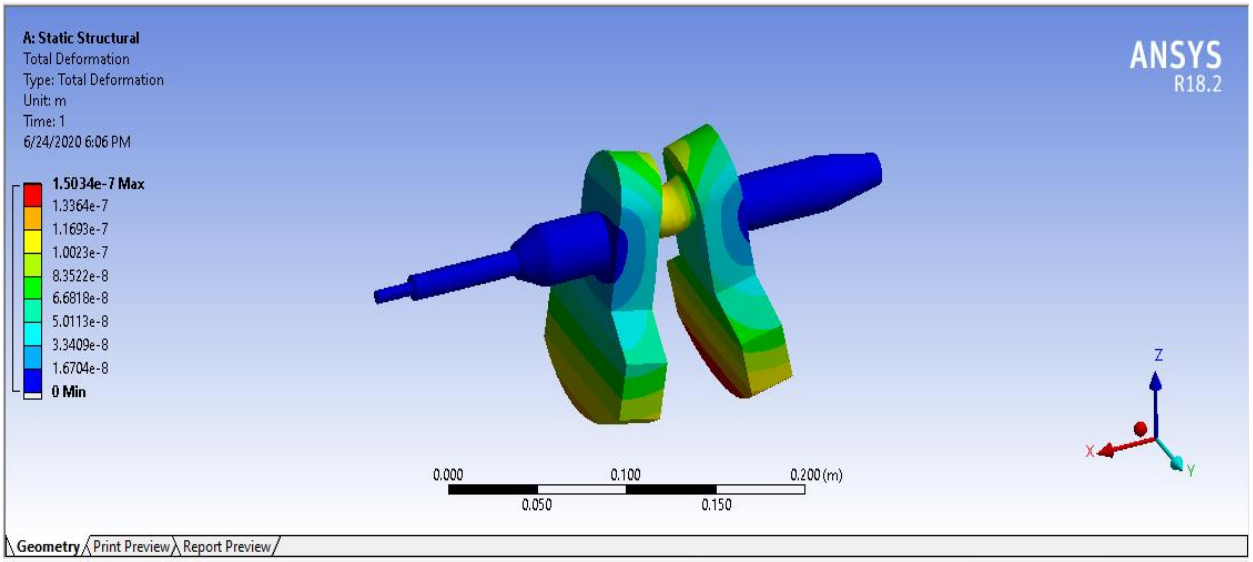


Figure 5.8: Total deformation

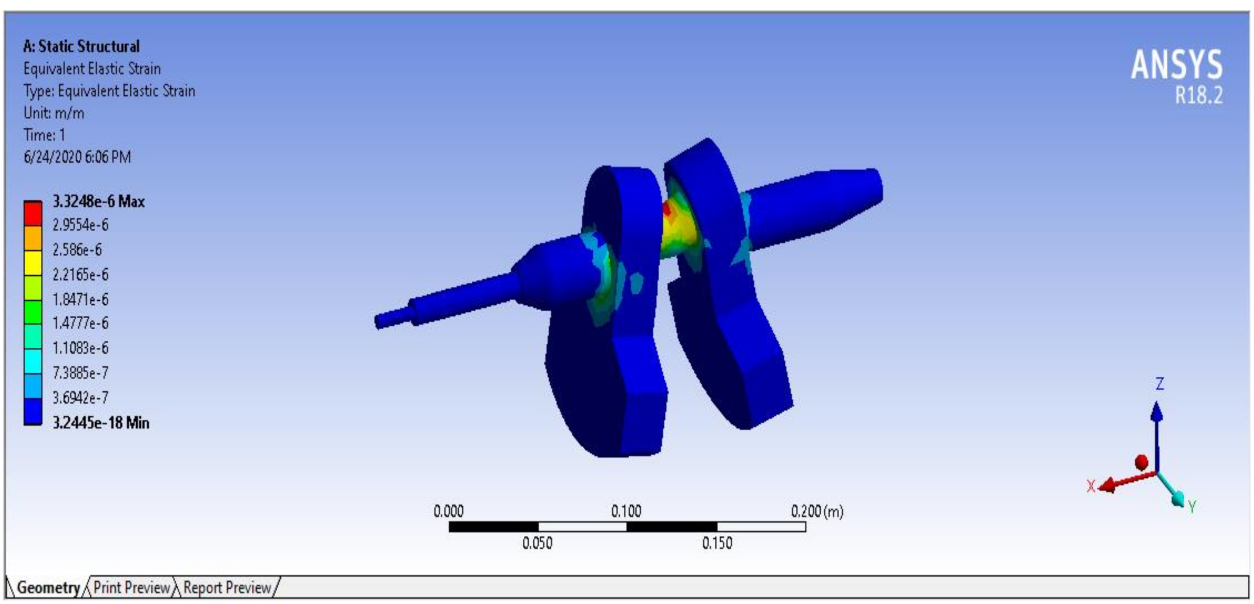


Figure 5.9: Strain

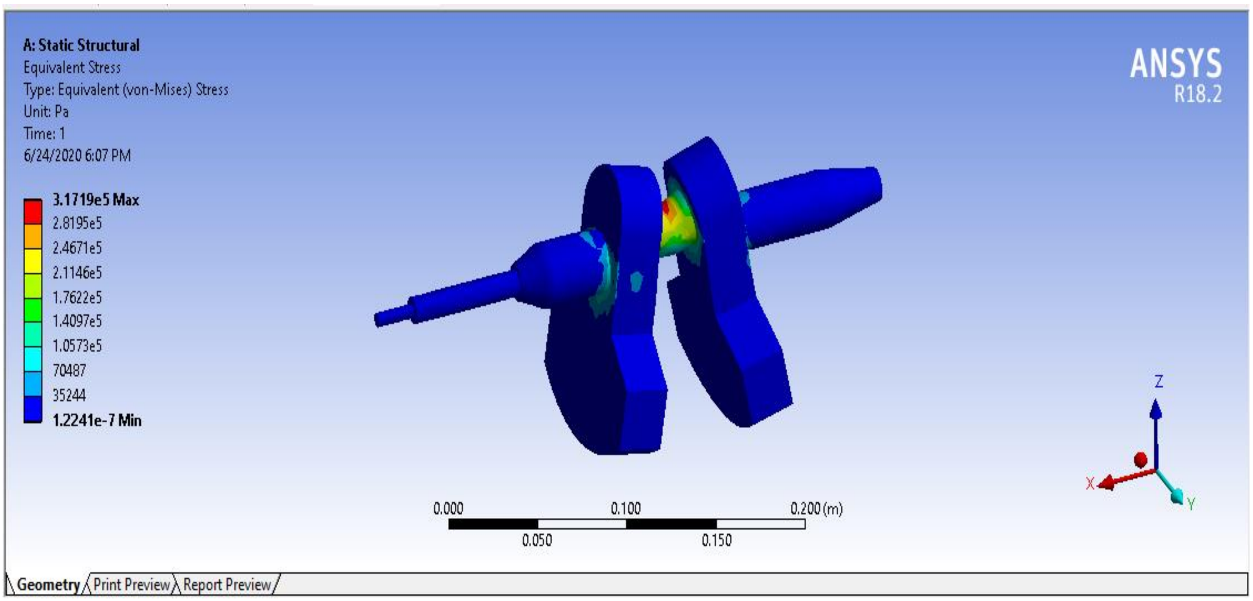


Figure 5.10: Stress

5.4 For Magnesium alloy

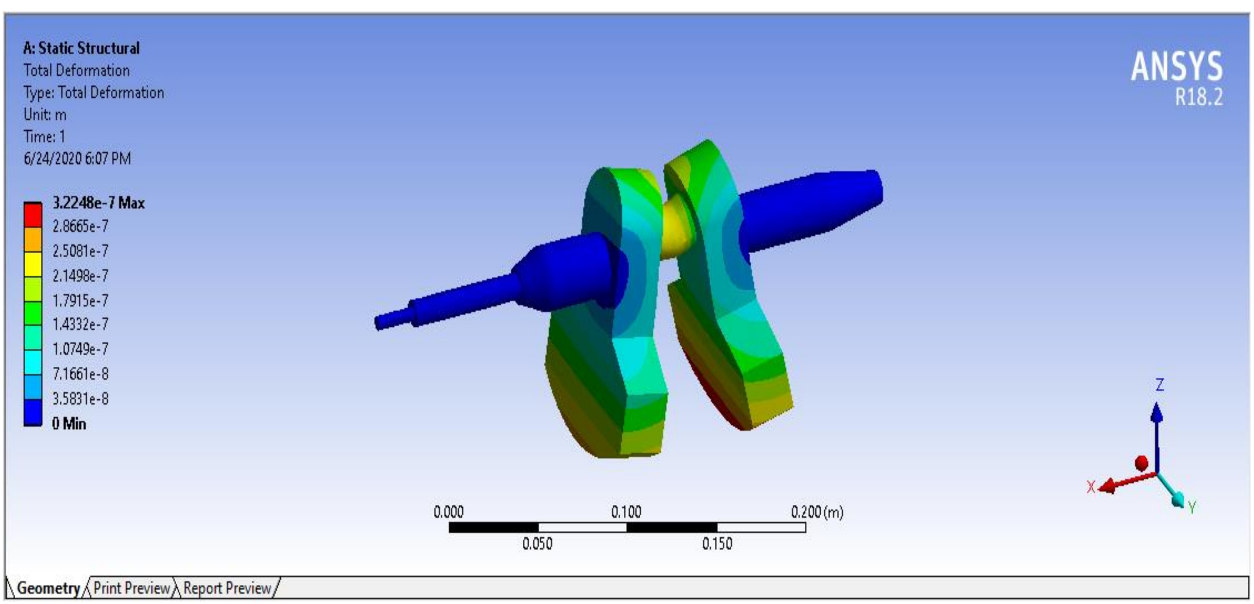


Figure 5.11: Total deformation

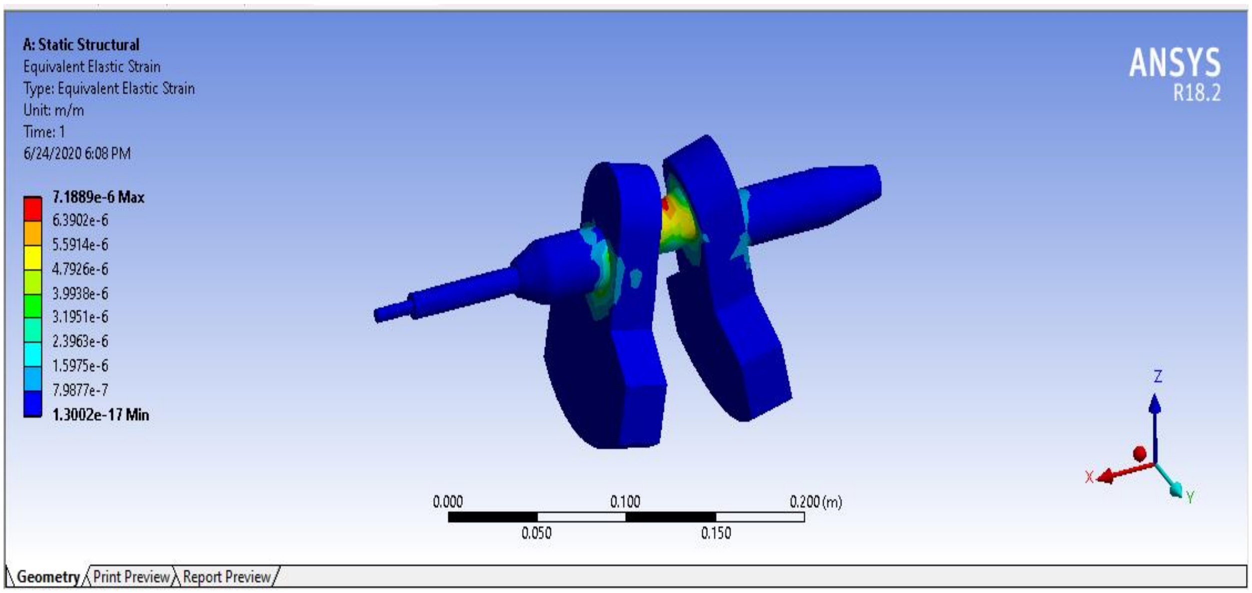


Figure 5.12: Strain

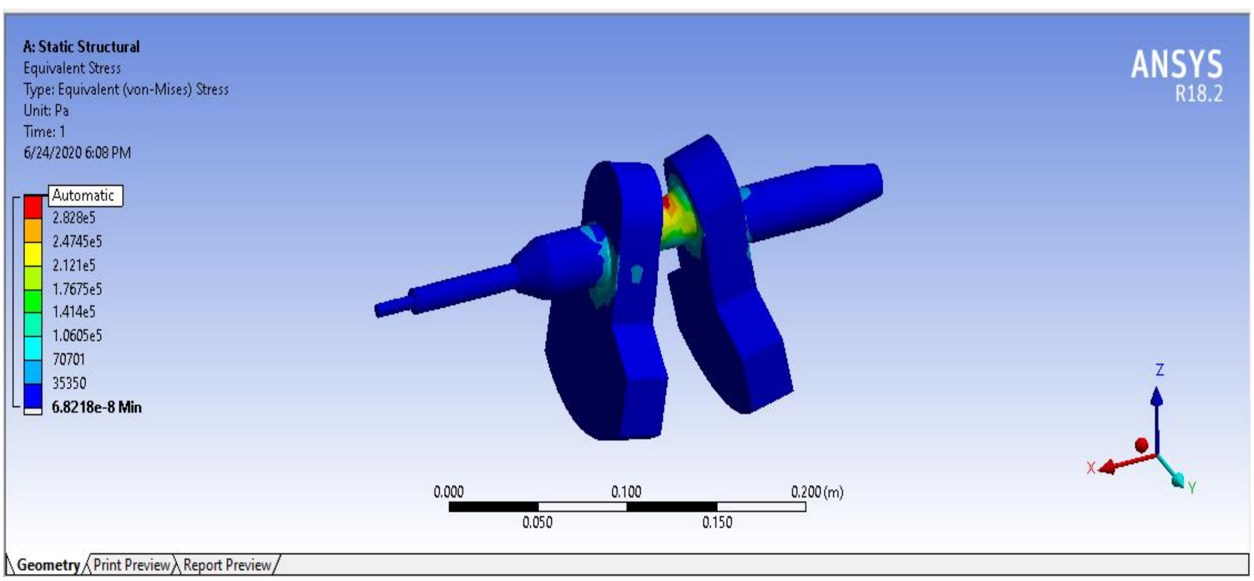


Figure 5.13: Stress

Chapter 6

Results

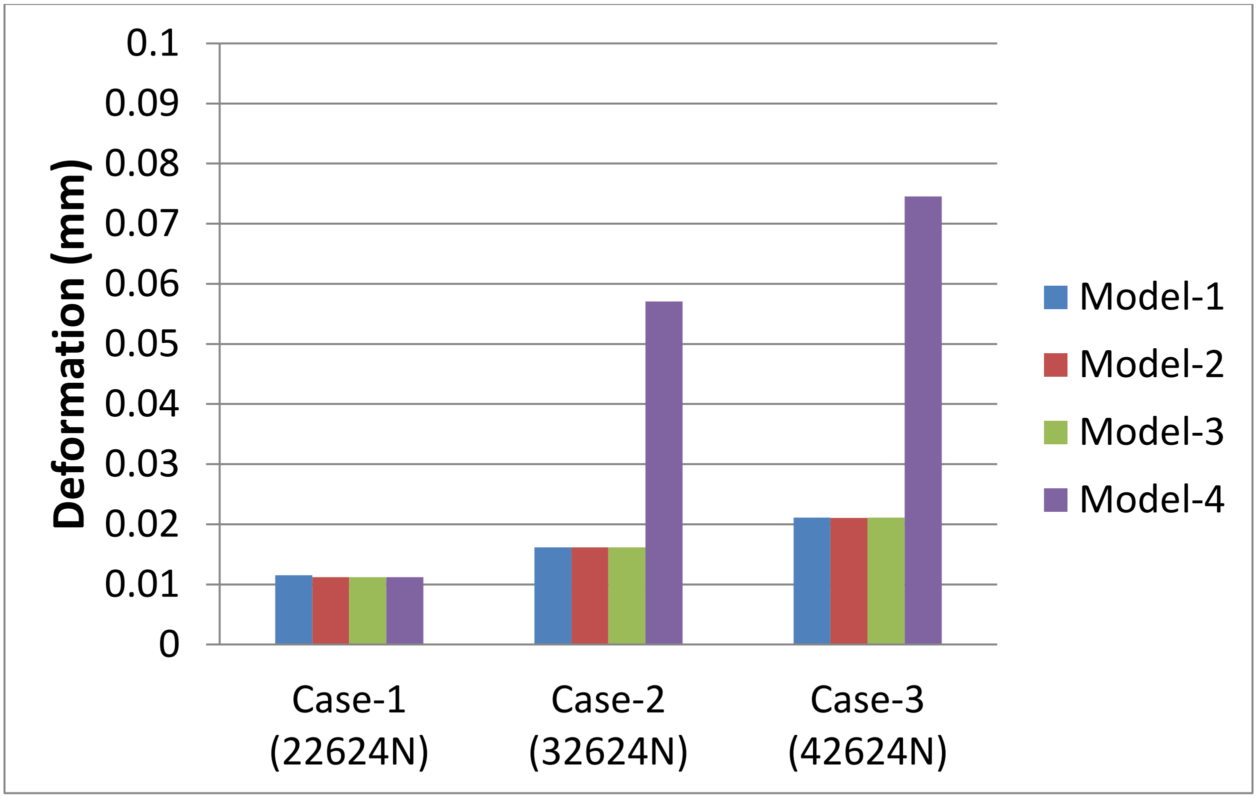
Comparison Study

Deformation comparison

After performing analysis in ansys following deformation are found as shown in table. The graph can be representing deformations in all 3 cases.

Table 6.1 Deformation comparison in all cases

Model	Case-1	Case-2	Case-3
1	0.0115	0.016144	0.02109
2	0.011182	0.016144	0.021079
3	0.011202	0.016149	0.021096
4	0.011207	0.057078	0.074564



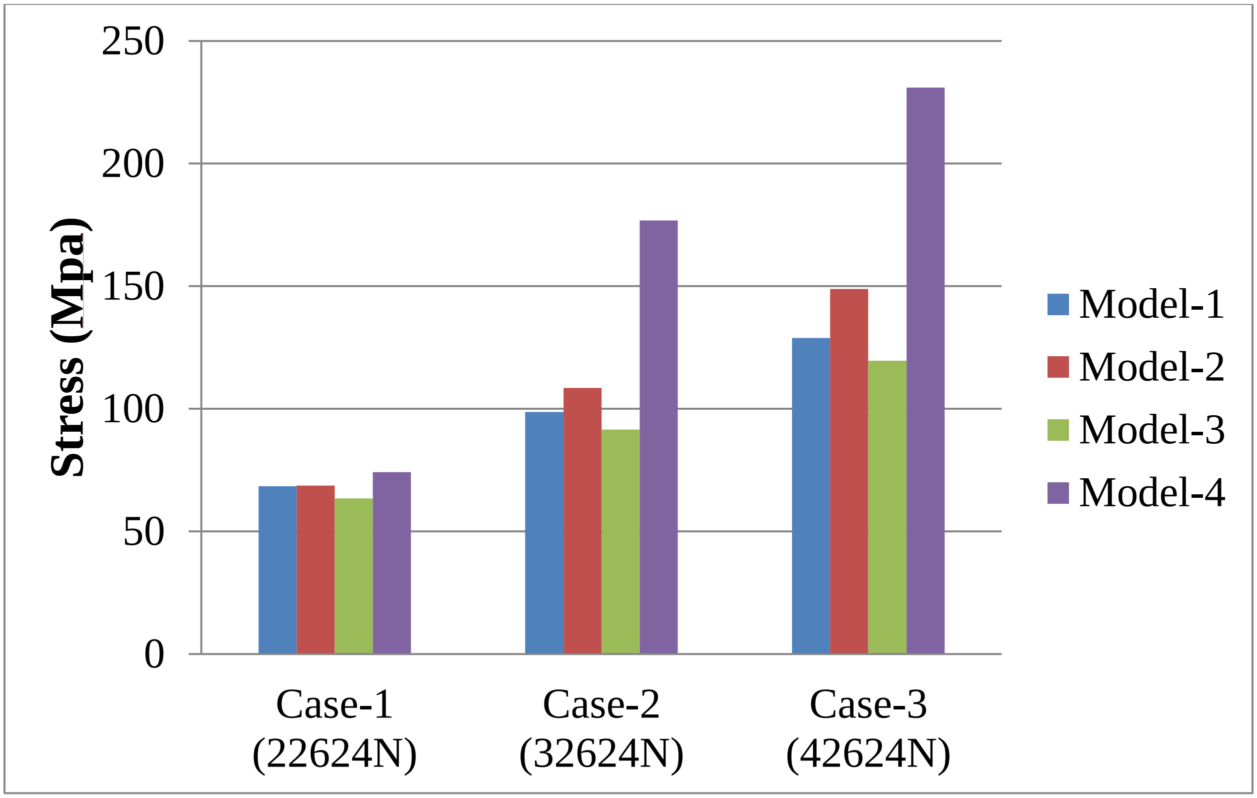
Graph 1 Deformation Graph

Stresses comparison

After performing analysis in ansys following Stresses are found as shown in table. The graph can be representing stress in all 3 cases.

Table 6.2 Stresses comparison in all cases

Model	Case-1	Case-2	Case-3
1	68.42	98.68	128.86
2	68.70	108.57	148.83
3	63.49	91.533	119.57
4	74.156	176.84	231.01



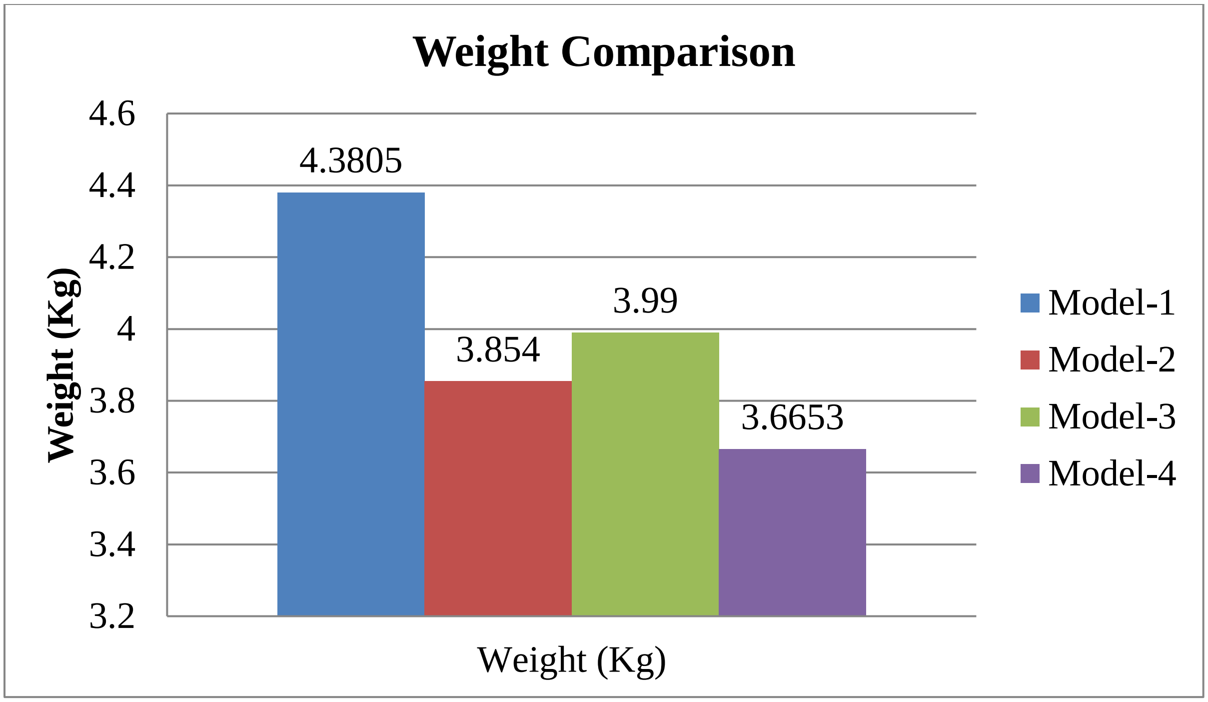
Graph 2 Stress Graph

Weight comparison

After performing analysis in ansys following Weight are found as shown in table. The graph can be representing weight in all 3 cases.

Table 6.3 Weight comparison in all 3 cases

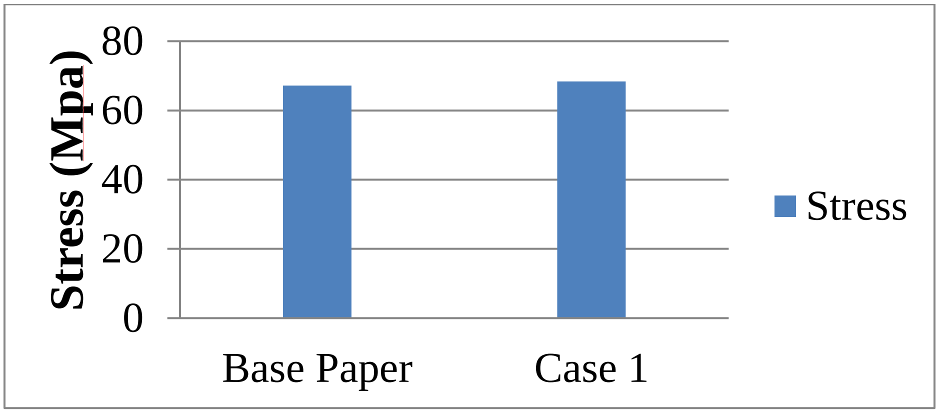
Case	Weight (Kg)
1	4.3805
2	3.854
3	3.990
4	3.6653



Graph 3 Weight comparison

Validation of Work

The maximum stress in previous is 67.14 Mpa and total deformation is 0.0433 mm which is vary closed to over model at loading condition 22624N force on crankshaft. The maximum stress in proposed case 1 is 68.42 Mpa and total deformation is 0.0115 mm this value is very closed to previous work model.



Graph 4 validation graph of Stress

Table 6.4: results for the analysis

Material	Total deformation (M)	Strain	Stress (pascals)
Aluminum alloy	2.0643e-7	4.6718e-6	3.2003e5
Titanium alloy	1.5034e-7	3.3248e-6	3.1719e5
Magnesium alloy	3.2248e-7	7.1889e-6	3.1815e5

CHAPTER 7

CONCLUSION

Crankshaft is a vital element of engine, failure even making engine useless conjointly needs expensive procurance and replacement. An intensive analysis within the past clearly indicates that the matter has not however been overcome fully and designers are facing lot of issues specially connected with multiaxial loading (Bending and Torsion), stress concentration and stress gradient and result of variable amplitude loading. Topology improvement was analyzed to the connecting rod. Based on these analysis results, concepts have been developed which reduce the weight of the crankshaft to a possible extent, without affecting the performance of the engine. The 3D model of crankshaft system, obtained from CATIA V5 software is analyzed in ANSYS to assess the motion and loads acting on the crankshaft. Topology optimization is help to optimize the performance of any machine. Some conclusions following are:

1. Topology optimization helps to optimize the performance of any machine. In these research three new models is proposed.

2. In previous research it is seen that The maximum stress for the bench mark model was found to be 67 MPa and whereas for the developed concepts 1, 2 and 3, it was found to be 80MPa, 71 MPa and 79 MPa respectively.

3. The present model is test in three different loading conditions 22624N, 32624N and 42624N load is applied in crank shaft.

4. After analysis ad comparison of all present models it is seen that the lowest stress, deformation and weight are found in model 3 in all loading condition.

5. The stress generate in three loading condition which are 22624, 32624, and 42624 N is 63.49, 91.53 and 119.57 Mpa respectively.

6. The weight of model 3 is 3.99Kg. The reduction of weight in present model is 9% less in overall weight of crank shaft.

7. It is conclude that the topology optimization process is help to reduce the material of workpiece without affecting the performance.

8. Modeling of crank shaft is done in catia v5 design programming by utilizing different orders
9. The catia part record is changed over into IGS document and imported to ansys workbench.
10. First Static auxiliary investigation is done on spike gear at 100 N with three unique materials, for example, aluminum compound , titanium amalgam and magnesium composite in ansys workbench.
11. Maximum stress, twisting and most extreme strain are noted and arranged

REFERENCES

[1.] Rincle Garg, Sunil Baghla, "Finite element analysis and optimization of crankshaft", International Journal of Engineering and Management Reaserch, vol-2,Issue-6,ISSN: 2250-0758, Pages:26-31, December 2012.

[2.] C.M Balamurugan, R. Krishnaraj, Dr.M.sakhivel, K.kanthavel, Deepan Marudachalam M.G, R.Palani, "Computer Aided modelling and optimization of Crankshaft", International Journal of scientific and Engineering Reaserach, Vol-2, issue-8, ISSN:2229- 5518, August-2011.

[3.] Gu Yingkui, Zhou Zhibo, "Strength Analysis of Diesel Engine Crankshaft Based on PRO/E and ANSYS", Third International Conference on Measuring Technology and Mechatronics Automation, 2011.

[4.] Abhishek choubey, Jamin Brahmbhatt, "Design and Analysis of Crankshaft for single cylinder 4- stroke engine", International Journal of Advanced Engineering Reaserch and studies, vol-1, issue-4, ISSN:2249-8974, pages: 88-90, July-sept 2012.

[5.] R.J Deshbhratar, Y.R Suple, " Analysis and optimization of Crankshaft using FEM", International Journal of Modern Engineering Reasearch, vol-2, issue-5, ISSN:2249-6645, pages:3086-3088, Sept-Oct 2012.

[6.] Farzin H. Montazersadgh and Ali Fatemi " Stress Analysis and Optimization of Crankshafts Subjected to Dynamic Loading", AISI, August 2007.

[7]Xiaorong Zhou., Ganwei Cai., Zhuan Zhang. Zhongqing Cheng., 2009, "Analysis on Dynamic Characteristics of Internal Combustion Engine INTERNATIONAL JOURNAL OF PROFESSIONAL ENGINEERING STUDIES Volume VIII /Issue 5 / AUG 2017 IJPRES Crankshaft System," International Conference on Measuring Technology and Mechatronics Automation.

[8]. Farzin H. Montazersadgh and Ali Fatemi., 2007,"Dynamic Load and Stress Analysis of a Crankshaft," SAE Technical Paper No. 010258, Society of Automotive Engineers

[9]. Jonathan Williams, Farzin Montazersadgh, and Alifatemi.,2007, "FatiguePerformance Comparison And Life Prediction Of Forged Steel And Ductile Cast Iron Crankshafts," Published in Proceeding of the 27th Forging Industry Technical Conference in Ft.Worth,Texas

[10]. Shenoy, P. S. and Fatemi, A., 2006, “Dynamic analysis of loads and stresses in connecting rods,”IMechE, Journal of Mechanical Engineering Science, Vol. 220, No. 5, pp. 615- 624

[11]. Zoroufi, M. and Fatemi, A.,2005, "A Literature Review on Durability Evaluation of Crankshafts Including Comparisons of Competing Manufacturing Processes and Cost Analysis", 26th Forging Industry Technical Conference, Chicago,

[12]. Prakash, V., Aprameyan, K., and Shrinivasa, U.,1998, “An FEM Based Approach to Crankshaft Dynamics and Life Estimation,” SAE Technical Paper No. 980565, Society of Automotive Engineers

[13]. Payar, E., Kainz, A., and Fiedler, G. A., 1995,“Fatigue Analysis of Crankshafts Using Nonlinear Transient Simulation Techniques,” SAE Technical Paper No. 950709, Society of Automotive Engineers [14]. Guagliano, M., Terranova, A., and Vergani, L., 1993,“Theoretical and Experimental Study of the Stress Concentration Factor in Diesel Engine Crankshafts, “Journal of Mechanical Design, Vol. 115, pp. 47-52

[15]. Henry, J., Topolsky, J., and Abramczuk, M., 1992,“Crankshaft Durability Prediction – A New 3-D Approach,” SAE Technical Paper No. 920087, Society of Automotive Engineers

[16]. Stephens, R. I., Fatemi, A., Stephens, R. R., and Fuchs, H. O., 2001, “Metal Fatigue in Engineering,”2nd edition, John Wiley and Sons, New York, NY,USA co.