



COMPUTER AIDED DESIGN & ANALYSIS OF CRANKSHAFT FOR 4S-S.I ENGINE USING FINITE ELEMENT METHOD

SHAIK NASEERUDDIN,
14NK1D1514, Velaga Nageswara Rao
College Of Engineering (Vnrc),
Chintalapudi, Ponnur, Guntur, Andhra
Pradesh -522124.
Email: naseer3.shaik@gmail.Com

P. MARIYA KUMAR,
Assistant Professor, Velaga Nageswara Rao
College Of Engineering (Vnrc),
Chintalapudi, Ponnur, Guntur, Andhra
Pradesh -522124.

ABSTRACT

In this project we designed and analyzed a crankshaft of 4-stroke S.I engine using "Ansys", which is software that works on the basis of finite element method.

Firstly, we created a solid model of the crankshaft using designing software "CATIA". Then the model is imported into Ansys and analyzed by applying necessary conditions, which were considered in designing it and then checked for the strength and life. The specifications required for the design are taken from the drafted design.

The results were found in the analysis of the crankshaft, the design is found out to produce more stresses and some modifications were done to the design and again it is analyzed and the stresses developed were lesser when compared to the previous design.

The engine, which is used, is a four-stroke S.I engine. It is a twin cylinder multi utility engine. It is a horizontal shaft engine. The cylinder volume of the engine is 196cc and it is used in cold countries for snow cutting purpose. It is also used for grass cutting.

Keywords: CAD, Analysis, FEM, Stress Development.

INTRODUCTION

An engine is something that produces an effect from a given input. The origin of engineering was the working of engines. There is an overlap in english between two meanings of the word "engineer": 'those who operate engines' and 'those who design and Internal combustion engine.

There is a wide range of internal combustion engines corresponding to their many varied applications. Likewise there is a wide range of ways to classify internal-combustion engines, some of which are listed below.

Although the terms sometimes cause confusion, there is no real difference between an "engine" and a "motor." At one time, the word "engine" (from Latin, via old french, ingenium, "ability") meant any piece of machinery. A "motor" (from Latin motor, "mover") is any machine that produces mechanical power. Traditionally, electric motors are not referred to as "engines," but combustion engines are often referred to as "motors." (An electric engine refers to locomotive operated by electricity). With that said, one must understand that common usage does often dictate definitions. Many individuals consider engines as those things which generate their power from within, and motors as requiring an outside source of energy to perform their work. Evidently, the roots of the words seem to actually indicate a real difference. Further, as in many definitions, the root word only explains the beginnings of the word, rather than the current usage. It can certainly be argued that such is the case with the words motor and engine.

INTERNAL COMBUSTION ENGINE:

The internal combustion engine is an engine in which the burning of a fuel occurs in a confined space called a combustion chamber. This exothermic reaction of a fuel with an oxidizer creates gases of high temperature and pressure, which are permitted to expand. The defining feature of an internal combustion engine is that useful work is performed by the expanding hot gases acting directly to cause movement, for example by acting on pistons, rotors, or even by pressing on and moving the entire engine itself.

The term Internal Combustion Engine (ICE) is almost always used to refer specifically to reciprocating engines, Wankel engines and similar designs in which combustion is intermittent. However, continuous combustion engines, such as Jet engines, most rockets and many gas turbines are also internal combustion engines.

CRANKSHAFT

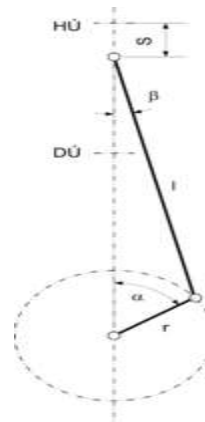
The crankshaft, sometimes casually abbreviated to crank, is the part of an engine which translates reciprocating linear piston motion into rotation. It typically connects to a flywheel, to reduce the pulsation characteristic of the four-stroke cycle, and sometimes a torsional or vibrational damper at the opposite end, to reduce the torsion vibrations often caused along the length of the crankshaft by the cylinders farthest from the output end acting on the torsional elasticity of the metal. The crankshaft was invented by the inventor Al-Jazari in the 12th century.

CRANK (MECHANISM):

A **crank** is a bent portion of an axle, or shaft, or an arm keyed at right angles to the end of a shaft, by which motion is imparted

to or received from it; also used to change circular into reciprocating motion, or reciprocating into circular motion. Familiar examples of a crank for manual use include the crank on a manual pencil sharpener and the crank set that drives a bicycle via the pedals.

Cranks were formerly common on some machines in the early 20th century; for example almost all phonographs before the 1930s were powered by clockwork motors wound with cranks, and internal combustion engines of automobiles were usually started with cranks before electric starters came into general use.



DESIGNING OF CRANK SHAFT USING CATIA

Overview of Solid Modeling

The CATIA modeling application provides a solid modeling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design. Design engineers can quickly perform conceptual and detailed designs using the Modeling feature and constraint based solid modeler. They can create and edit complex, realistic, solid models



interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modeling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Advantages of Solid Modeling

Solid Modeling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position.

Features

- Powerful built-in engineering-oriented form features-shaft, holes, extrudes, rib, sweep, swept blend, cuts-capture design intent and increase productivity.
- Patterns of feature instances-rectangular and circular arrays-with displacement of individual features; all features in the pattern are associated with the master feature

Blending and Chamfering

- Zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modeling Operations

- Profiles can be swept, extrude or shaft to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary

- Fixed and variable radius blends may overlap surrounding faces and extend to a Tapering for modeling manufactured near-net shape parts
- User-defined features for common design elements

FINITE ELEMENT METHOD

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called “Finite Elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement with in the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an aircraft. In the recent years, FEA has been universally used to solve structural engineering problems. The



departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to rely on this technique to stay competitive.

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a hand brake design fault, they would end up having to replace up to few millions of hand brakes. This will cause a heavier loss to the company.

The finite element method is a very important tool for those involved in engineering design; it is now used routinely to solve problems in the following areas.

- Structural analysis
- Thermal analysis
- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method for design evaluation. This is because contemporary design problems usually can not be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in the years gone by, but now it is simply too expensive and time consuming also.

Basic Concepts: The Finite Element Method is based on the idea of building a

complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as engineering. The philosophy of FEA can be explained with a small example such as measuring the area of a circle. Area of one Triangle: $S_i = \frac{1}{2} * R^2 * \sin \theta_i$

Area of the Circle: $S_N = \frac{1}{2} * R^2 * N * \sin (2\pi / N) \rightarrow \pi R^2$ as $N \rightarrow \infty$

Where N = total number of triangles (elements)

If one needs to evaluate the area of the circle without using the conventional formula, one of the approaches could be to divide the above area into a number of equal segments. the area of each triangle multiplied by the number of such segments gives the total area of the circle.

ANSYS

The ANSYS program is self-contained general purpose finite element program developed and maintained by Swanson Analysis Systems Inc. The program contain many routines, all inter related, and all for main purpose of achieving a solution to an engineering problem by finite element method.

ANSYS finite element analysis software enables engineers to perform the following tasks:

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions
- Study physical responses, such as stress levels, temperature



distributions, or electromagnetic fields

- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible

The ANSYS program has a compressive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS Program. Users can input data using a mouse, a keyboard, or a combination of both. A graphical user interface is available throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialog boxes, tool bar and online documentation.

ORGANIZATION OF THE ANSYS PROGRAM

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The begin level acts as a gateway in to and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When we first enter the program, we are at the begin level.

At the processor level, several processors are available; each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor (PREP7) is where we build the model, the solution processor (SOLUTION) is where we apply loads and obtain the solution, and the general postprocessor (POST1) is where we evaluate the results and obtain the solution. An additional postprocessor (POST26), enables we to evaluate solution results at specific points in the model as a function of time.

PERFORMING A TYPICAL ANSYS ANALYSIS

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

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

The following table shows the brief description of steps followed in each phase.

Pre-Processor	Solution processor	Post-processor
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	Load definition	View animated results



Material definition	Solve	
Mesh generation		
Model display		

PRE-PROCESSOR:

The input data for an ANSYS analysis are prepared using a preprocessor. The general preprocessor (PREP 7) contains powerful solid modeling and mesh generation capabilities, and is also used to define all other analysis data with the benefit of database definition and manipulation of analysis data. Parametric input, user files, macros and extensive online documentation are also available, providing more tools and flexibility.

For the analyst to define the problem. Extensive graphics capability is available throughout the ANSYS program, including isometric, perspective, section, edge, and hidden-line displays of three-dimensional structures—graphs of input quantities and results, and contour displays of solution results.

The pre-processor stage involves the following:

- Specify the title, which is the name of the problem. This is optional but very useful, especially if a number of design iterations are to be completed on the same base model.
- Setting the type of analysis to be used, e.g., Structural, Thermal, Fluid, or electromagnetic, etc
- Creating the model. The model may be created in pre-processor, or it can be imported from another CAD drafting package via a neutral file format.

- Defining element type, these are chosen from element library.
- Assigning real constants and material properties like Young's modulus, Poisson's ratio, density, thermal conductivity, damping effect, specific heat, etc
- Apply mesh. Mesh generation is the process of dividing the analysis continuum into a number of discrete parts of finite elements.

MESHING:

Before meshing the model and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it.

Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. If we want this type of mesh, we must build the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh.

FREE MESHING:

In free meshing operation, no special requirements restrict the solid model. Any model geometry even if it is regular, can be meshed. The element shapes used will depend on whether we are meshing areas or



volumes. For area meshing, a free mesh can consist of only quadrilateral elements, only triangular elements, or a mixture of the two. For volume meshing, a free mesh is usually restricted to tetrahedral elements. Pyramid shaped elements may also be introduced in to the tetrahedral mesh for transitioning purposes.

MAPPED MESHING: We can specify the program use all quadrilateral area elements, all triangular area elements or all hexahedra brick volume elements to generate a mapped mesh. Mapped meshing requires that an area or volume be “regular”, i.e., it must meet certain criteria. Mapped meshing is not supported when hard points are used. An area mapped mesh consists of either all quadrilateral elements or all triangular elements

STRUCTURAL STATIC ANALYSIS:

A static analysis calculates the effects of study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed, i.e. the loads and the structure's responses are assumed to vary slowly with respect to time. The kinds of loading that can be applied in static analysis include:

- Externally applied forces and pressures.

- Steady state inertial forces
- Imposed displacement
- Temperatures
- Fluences (for nuclear swelling)

ANALYSIS OF CRANK SHAFT

Before analyzing the crankshaft by using the software the stresses developed in the crankshaft are:

The bearing pressures are important in the design of the crankshaft. The maximum permissible bearing pressure depends upon the max. Gas pressure, journal velocity, amount and method of lubrication & change of bearing pressure.

The following types of stresses induced in the crankshaft:

1. Bending stress; & 2. Shear stress due to torsional moment on the shaft.

Most of the crankshaft failures are caused by a progressive fracture due to repeated bending or reversed torsional stress.

Here the max. Stress acting on the body is calculated and a theory of failure of Von-mises stresses is taken into consideration.

At first the crankshaft is designed according to the specifications. The specifications are taken from the drafting figure. After that the model is imported to the analyzing software and then the stresses are calculated the following steps are followed:

SPECIFYING GEOMETRY

First the geometry of the structure to be analyzed is defined. This can be done either by entering the geometric information in the finite element package through the keyboard or mouse, or by importing the model from a solid modeler like CATIA.



SPECIFY ELEMENT TYPE AND MATERIAL PROPERTIES

Next, the material properties are defined. In an elastic analysis of an isotropic solid these consist of the Young's Modulus and the Poisson's Ratio of the material.

MESHING THE OBJECT

Then, the structure is broken into small elements. This involves defining the types of

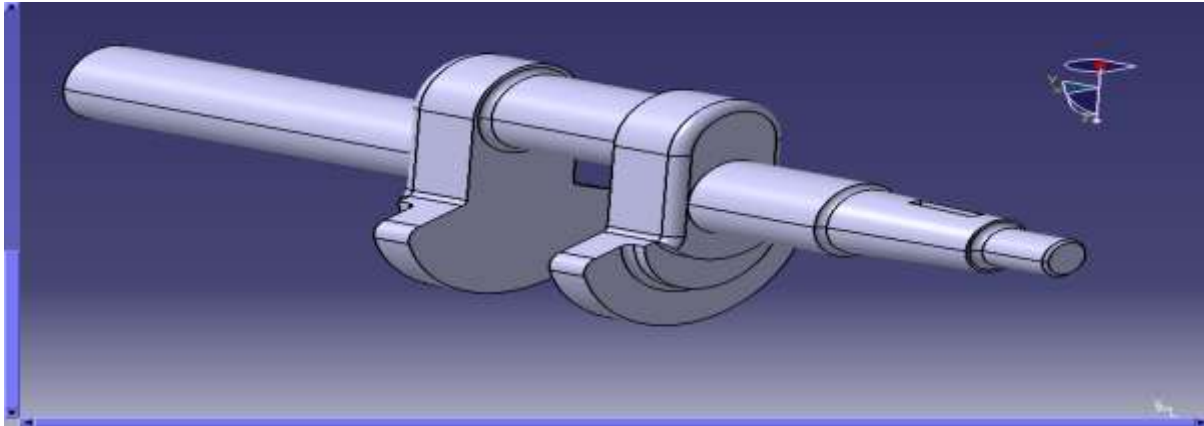
elements into which the structure will be broken, as well as specifying how the structure will be subdivided into elements (how it will be meshed).

This subdivision into elements can either be input by the user or, with some finite element programs can be chosen automatically by the computer based on the geometry of the structure (this is called auto meshing)

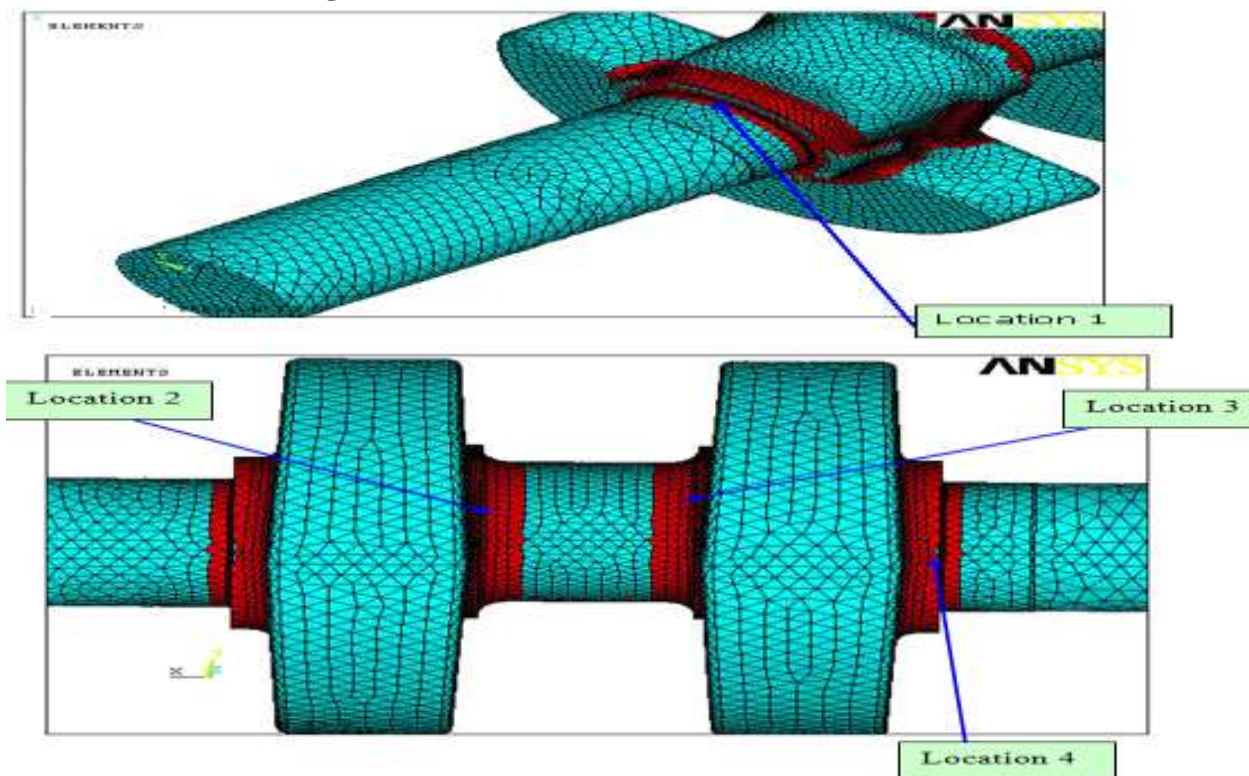
Material Properties of Crankshaft

Model	Material	H	Grade	ASTMA	Tensile Strength (Mpa)	Young's Modulus (Gpa)	Poisson's Ratio
MLA (TDC)	Ductile Iron	209	65-45-12	536-84(1993)	448	172	0.28
	Austempered Ductile Iron		80-55-06	536-84(1993)	552	165.47	0.28
AW	Steel (Forged) Low carbon Steel	145	C1118 C1524 C1117	Modern steels and their properties “ Bethlehem Steel Corporation Handbook” (Vol1)	-	207	0.29
MLA (Elliot)	Austempered Ductile Iron	219					
	Austempered Ductile Iron	209	80-55-06	536-84 (1993)	552	165.47	0.28

MODEL OF CRANKSHAFT



Mesh in the Critical Regions:



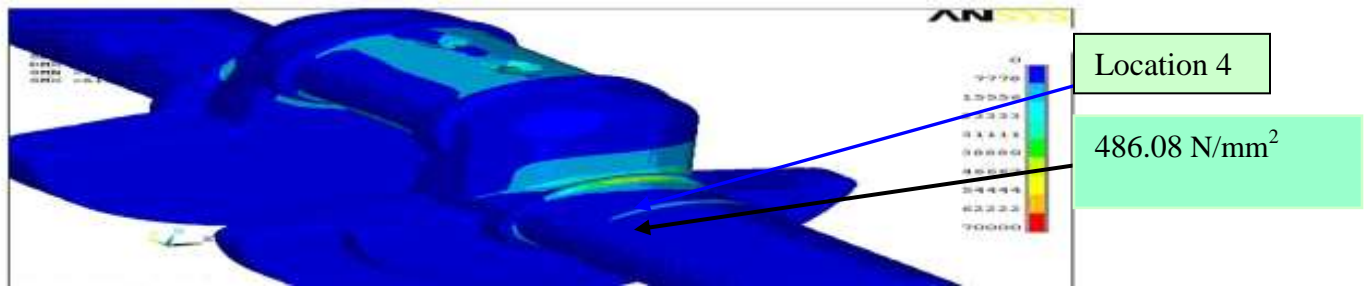
Boundary conditions:

LOADING CONDITIONS

A distributed load was applied from connecting rod to crank shaft over a swept 45-degree area on the crankpin by the width of the connecting rod bearing as shown below

Load applied from cylinder 1 = 2293.76kgs (13.5 deg after TDC)

MODIFIED SHAFT: Von misses stress:



ORIGINAL SHAFT BENDING:

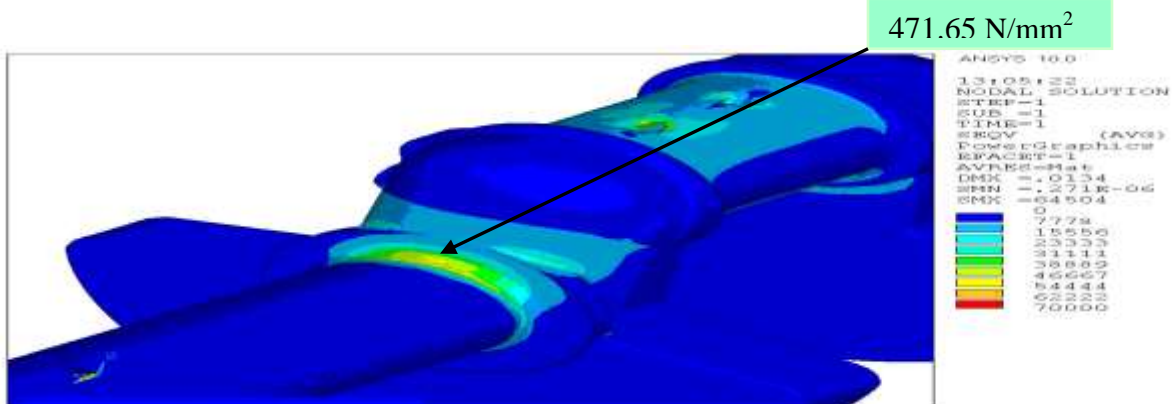


MODIFIED SHAFT BENDING:

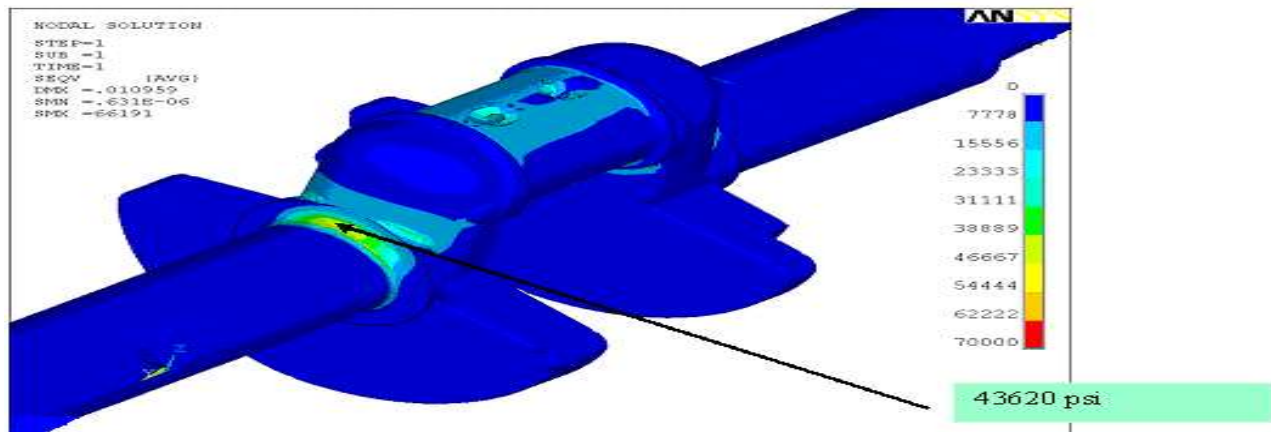


CASE-2 (LOAD APPLIED FROM CYLINDER2)

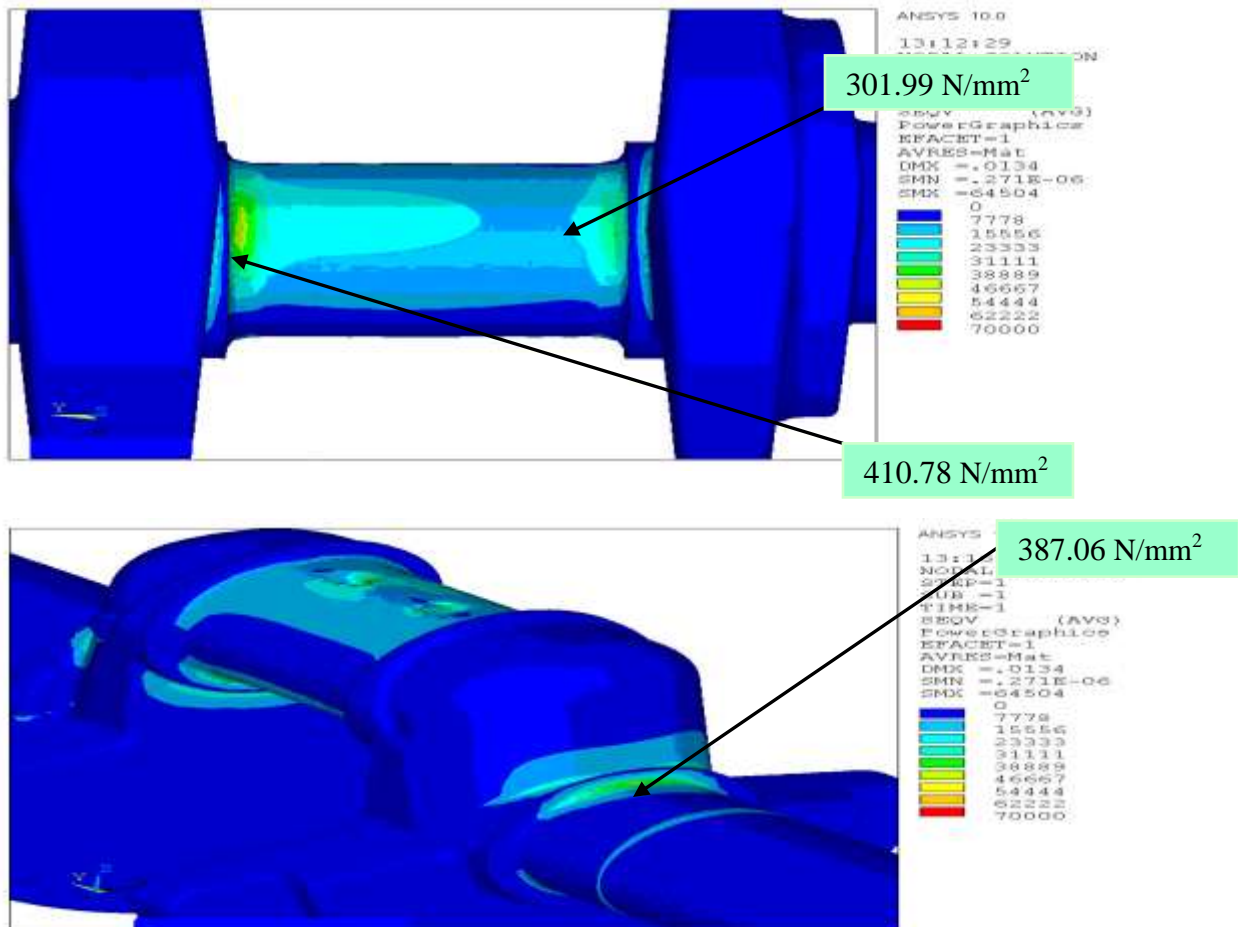
ORIGINAL SHAFT: Von misses stress :Location1



MODIFIED SHAFT: Von misses stress :Location1



ORIGINAL SHAFT: Von misses stress :Location2&3



ORIGINALSHAFT: BENDING:



RESULTS AND CONCLUSION

The stresses are calculated and the results are tabulated above. The von-misses stresses are calculated and the maximum Stresses acting on the given and the stresses acting at the various locations are represented in graph.

To perform static analysis on crankshaft to reduce the stress at concentrated areas.

The material, which we have used for analyzing the crankshaft, is mild steel.

Comparison Table:

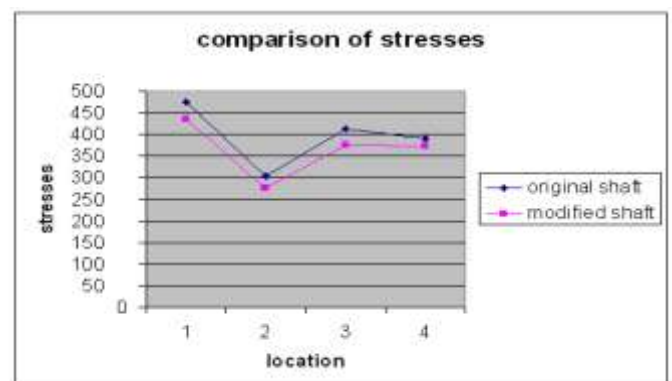
Case-1: Load Applied From Cylinder 1

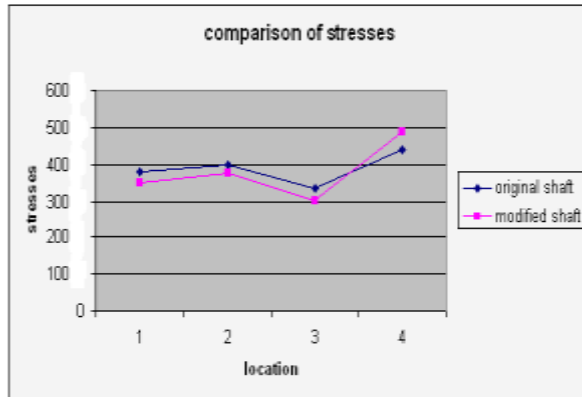
STRESS IN Psi	ORIGINAL SHAFT	MODIFIED SHAFT
Location-1	471.65	432.71
Location-2	301.99	274.73
Location-3	410.78	371.51
Location-4	387.06	370.11
MAXIMUM DIS (In)	0.34036	0.2783

Case-2: Load Applied From Cylinder 2:

STRESS IN Psi	ORIGINAL SHAFT	MODIFIED SHAFT
Location-1	375.51	347.2
Location-2	394.32	371.69
Location-3	333.31	300.9
Location-4	436.48	486.08
Maximum Dis (In)	0.3243	0.2537

GRAPHS:





CONCLUSION:

The crankshaft with modified dimensions is compared to the original dimensions and it is found that the crankshaft with modified dimensions is giving better results than the original one. The deformation of the design is lesser than the original model. We suggest the modified model for the required engine.

BIBLIOGRAPHY:

1. Internal Combustion Engines

-By

V.Ganeshan

2. Introduction To Finite Elements

-By

Thirupathi R.Chanrupatla

3. Finite Elements Methods

-By

Bathe

4. Machine Design

-By

R.S.Khurmi And J.K.Gupta

5. Internal Combustion Engines

-By

Mathur And Sharma

6. Introduction To Finite Element Method

-By

University Of Cincinnati