DESIGN AND ANALYSIS OFCONNECTING ROD

By

Ummidi Nandababu[1], Chevala Durga Ganesh[2], Pelluri Naresh Raj[3], Polam Sai Krishna[4], K.Siva Satya Mohan[5]

[1,2,3,4]B.Tech, [5]Associate Professor, Aditya College of Engineering & Technology, Surampalem, A.P, India, 533437

ABSTRACT

The main function of a connecting rod is to convert linear motion of piston to rotary motion of crank. It is the main component of an internal combustion (IC) engine and is the most heavily stressed part in the engine. During its operation various stresses are acting on connecting rod. The influence of compressive stress is more in connecting rod due to gas pressure and whipping stress. The objective of this study is to carry out a FEA analysis of a connecting rod and obtain its stress distribution on application of the force. Geometry of connecting rod used for FEA, its generation, simplifications and accuracy is done by using Ansys. Mesh generation, the load application, particularly the distribution at the contact area, factors that decide application of the restraints and validation of the FEA model are also discussed. FEM was used to determine structural behavior under static load conditions.

1. INTRODUCTION

1.1 INTRODUCTION ABOUT CONNECTING ROD

Automobile components are in great demand these days because of increased use ofautomobiles.Theincreased demand is due to improved performance and reduced cost of thesecomponents. R&D and testing engineers should develop critical components in the shortest possibletime to minimize launch time for new products. This necessitates understanding of new technologiesand quick absorption in the development of new products.

A connecting rod is a vital component in internal combustion engines, serving as a crucial link between the piston and the crankshaft. It converts thereciprocating motion of the piston into rotary motion, which is essential for driving various mechanical systems such as vehicles, generators, and industrial machinery.

Figure1:Connecting Rod

1.2 PARTS OFA CONNECTING ROD:

Big End: The larger end of the connecting rod, which connects to the crankshaft journal. It typically has a larger diameter to accommodate the forces generated during engine operation.

Small End: The smaller end of the connecting rod, which connects to the piston pin or wrist pin. It's usually narrower in diameter compared to the big end.

Shank:The central portion of the connecting rod, between the big and small ends. It provides structural support and transfers forces between the piston and crankshaft.

Cap: In some designs, the connecting rod is composed of two separate pieces - the rod and the cap. The cap is fastened to the rod by bolts or nuts and helps secure the big end to the crankshaft.

Bolts or Fasteners: Connecting rods are secured to the crankshaft and piston using bolts or fasteners. These bolts are crucial for maintaining the integrity of the assembly under high loads and vibrations.

Bearings: Bearings are often used at both ends of the connecting rod to reduce friction and wear between the rod and the crankshaft or piston pin.

Oil Holes: Connecting rods may have oil passages or holes to facilitate lubrication and cooling of the moving parts, especially the bearings.

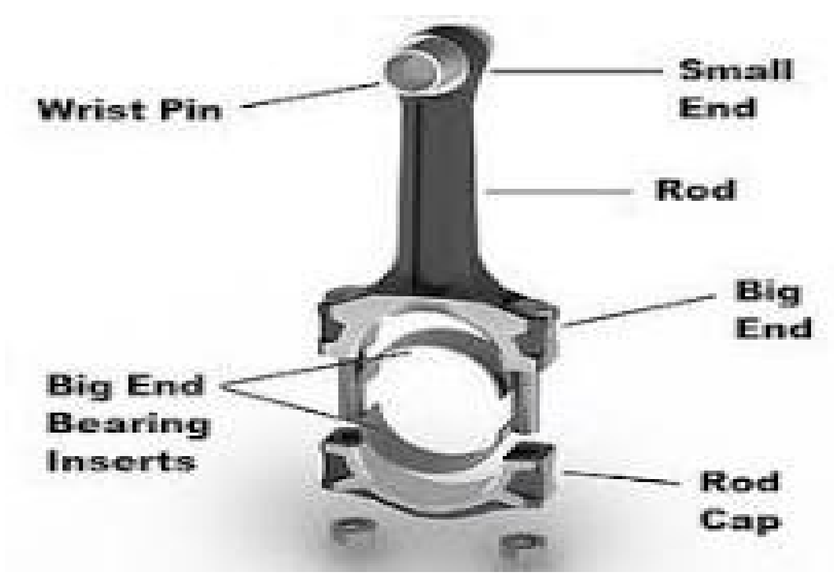


Figure 2: Schematic Parts of Connecting Rod

.

1.3 NOMENCLATUREOF CONNECTING ROD:

Connecting rods are often identified by their dimensions and specifications, which include:

Center-to-Center Length: The distance between the centerlines of the big and small ends, typically measured in millimeters or inches.

Big End Diameter: The diameter of the opening at the big end of the connecting rod, where it connects to the crankshaft journal.

Small End Diameter: The diameter of the opening at the small end of the connecting rod, where it connects to the piston pin.

Material: Connecting rods can be made from various materials such as steel, aluminum, or titanium, depending on the application and performance requirements.

Designation: Some connecting rods may have specific designations or part numbers assigned by manufacturers for identification and ordering purposes.

1.4 WORKINGOF A CONNECTING ROD :

During the operation of an internal combustion engine, the connecting rod undergoes a series of complex motions to transmit power from the piston to the crankshaft. The process can be summarized as follows:

Intake Stroke: As the piston moves downward during the intake stroke, the connecting rod follows suit, transmitting the motion to the crankshaft through the big end.

Compression Stroke: During the compression stroke, the piston moves upward, and the connecting rod reverses its motion, pushing against the crankshaft.

Power Stroke: When the air-fuel mixture ignites, the resulting expansion forces the piston downward with considerable force. The connecting rod again transmits this motion to the crankshaft, converting it into rotary motion.

Exhaust Stroke: Finally, during the exhaust stroke, the spent exhaust gases are expelled from the combustion chamber as the piston moves upward once more, and the connecting rod completes another cycle of motion

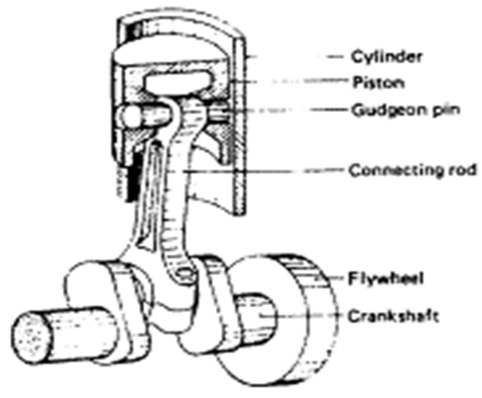


Figure 3: Internal view of Connecting Rod

1.5 LOADS&MATERIALPROPERTIES:

1.5.1 LOADS:

Heavy mechanical and thermal loads due to operating conditions.

1.5.2 MATERIALPROPERTIES:

- ➢ High strength,
- ➢ High resistance to corrosion and wear

1.6 Types of connecting rods:

Connecting rods can vary in type based on several factors such as their design, construction method, material, and application. Here are the main types of connecting rods:

1.6.1 I-Beam Connecting Rods:

Figure 4.a: I-Beam

These rods have an **I-shaped cross-section**, offering a good balance between strength and weight. They are commonly used in a wide range of engines due to their versatility and cost-effectiveness

1.6.2 H-Beam Connecting Rods:

Figure 4.b: H-Beam

H-**beam** rods have an H-shaped cross-section, providing increased strength and rigidity compared to I-beam rods. They are often used in high-performance and racing engines where additional strength is required.

1.6.3 X-Beam Connecting Rods:

Figure 4.c: X-Beam

X-beam rods feature an X-shaped cross-section, offering excellent strength and stiffness. They are designed to handle high-stress applications and are commonly used in high-performance engines.

Based on Material:

1.6.4 Steel Connecting Rods:

Made from **steel alloys**, these rods offer good strength and durability. They are commonly used in a wide range of engines, from everyday vehicles to high-performance applications.

Aluminum Connecting Rods: Aluminum rods are lightweight, which helps reduce reciprocating mass and improve engine response. They are often used in high-performance and racing engines where weight reduction is critical.

1.6.5 Titanium Connecting Rods:

Titanium rods combine strength with low weight, making them ideal for high-performance applications where weight reduction is crucial. They offer a good balance between strength and weight but can be more expensive than steel or aluminum rods.

2. LITERATURESURVEY:

A.Design and Analysis of a Connecting Rod by Anoop P from International Journal of Engineering Research & Technology (IJERT) SSN: 2278-0181 IJERTV5IS100142 Vol. 5 Issue 10, October-2016.

In this paper, we observed The main function of a connecting rod is to convert linear motion of piston to rotary motion of crank. It is the main component of an internal combustion (IC) engine and is the most heavily stressed part in the engine. During its operation various stresses are acting on connecting rod. The influence of compressive stress is more in connecting rod due to gas pressure and whipping stress.

In this paper, we observed that analysis is performed that connecting Rod made up of Aluminium has higher intensity of stress induced as compared to connecting Rod made up of Steel. Also there is a great opportunity to improve the design. Hence steel is a better choice for connecting rods.

B.Design and Analysis of a Connecting Rod by Magesh Kumar, PG Scholar International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169 Volume: 5 Issue: 6 479 – 481

In this paper, we observed that This topic deals with the past literature survey which shows that, In an internal combustion engines and compressors, connecting rod is a high volume production, critical component which is periodically subjected to high tensile, compressive and bending loads caused by the thrust and pull on the piston and by the centrifugal force of the rotating crankshaft.

Connecting rods are being manufactured by conventional method of forging. Steel can be replaced by aluminium and titanium alloys on a cost of affordability. Weight optimization is possible using composite materials without varying the allowable stresses and boundary conditions.

C.Design and Analysis of connecting rod using foriged steelby Leela Krishna Vegi, VenuGopal Veg from International Journal of Scientific & Engineering Research, Volume 4, Issue 6, June 2013 1 ISSN 2229-5518 |PAPER ID: I021991|. In this paper, we observed that The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This thesis describes designing and Analysis of connecting rod. Currently existing connecting rod is manufactured by using Carbon steel. In this drawing is drafted from the calculations.In this paper, we observed The connecting rod is the intermediate member between the piston and the Crankshaft. By checking and comparing the results of materials in inalizing the results are shown in below.

D.Design And Analysis of Connecting ROD USING ALUMINIUM ALLOY by Mohamed Abdusalam Hussin, Dr. Arvind Saran Darbari from International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169 Volume: 5 Issue: 6 479 – 481

In this paper, it is observed that Currently existing connecting rod is manufactured by using Forged steel. In this drawing is drafted from the calculations. A parametric model of Connecting rod is modeled using SOLID WORK software and to that model, analysis is carried out by using ANSYS 15.0 Software.In this paper, we observed that future scope Solid modeling of connecting rod were made according to production drawing specification and analysis under the effect of tensile and compressive loads in terms of pressure is done in ANSYS Workbench. In the present design and analysis of connecting rod using aluminum alloy 7068 T6,T6511 have been done with the help of SOLID WORK and ANSYS 15.0.

E.Design & analysis of connecting rod using ANSYS software by Vinayak Brenia from Science Direct Volume 5 6 , Part 4 , 2 0 2 2 , Pages 1 8 9 6 -1 9 0 3

In this paper, it is observed Connecting rod is the important part of I. C. Engine to work. It is the “backbone” of an engine. Analysis of a connecting rod have been carried out and presented by different researchers. Analysis shows Maximum Equivalent (Von-Mises) Stress and Maximum Normal Stress are coming less than yield Strength of connecting Rod material with FOS 1.26 and 3.406, respectively. Maximum total deformation is 0.06885mm, which is very less.

F.Design and Analysis of Connecting Rod using Different Materials by Mr. H D. Nitturkar from International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 07 Issue: 03 | Mar 2020

In this paper, it is observed that - The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This thesis describes designing and Analysis of connecting rod. Currently, existing connecting rod is manufactured by using Forged steel.In this paper, we observed that Solid modeling of connecting rod was made in NX 10 according to design procedure used and analysis under the effect of tensile and compressive loads in terms of pressure is done in ANSYS Workbench.

2. From analysis it is observed that the minimum stresses among all loading conditions, were found at crank end cap as well as at piston end. So the material can be reduced from those portions, thereby reducing material cost.

G.Design & analysis of connecting rod using by B.krishna kanth from International Journal & Magzine of engineering, technology, management and Research

In this paper, it is observed The intermediate component between crank and piston is known as connecting rod. The objective of connecting rod is to transmit push & pull from the piston pin to the crank pin and then converts reciprocating motion of the piston into the rotary motion of crank. The components are big m shank, a small end and a big end. The cross section of shank may be rectangular, circular, tubular, I-Section, + -section or ellipsoidal-Section. In this paper, we obeserved that thesis describes designing and Analysis of connecting rod. Currently existing connecting rod is manufactured by using Carbon steel. In this drawing is drafted from the calculations. A parametric model of Connecting rod is modelled using CATIA V5 R20 software and to that model, analysis is carried out by using ANSYS 13.0 Software. Finite element analysis of connecting rod is done by considering the materials, viz...

H.**Design Optimization and Analysis of a Connecting Rod using ANSYS** by G. Naga Malleshwara Rao from International Journal of Science and Research (IJSR), India Online ISSN: 2319-7064

In this paper, it is observed The main Objective of this work is to explore weight reduction opportunities in the connecting rod of an I.C. engine by examining various materials such as Genetic Steel, Aluminum, Titanium and Cast Iron. This was entailed by performing a detailed load analysis. Therefore, this study has dealt with two subjects, first, static load and stress analysis of the connecting rod and second, Design Optimization for suitable material to minimize the deflection. In the first of the study the loads acting on the connecting rod as a function of time are obtained.

In this paper, it is observed that This work investigated weight reduction and the suitable bettermaterial for minimizing deflections in connecting rod. Load analysis was performed which comprised of the connecting rod, small and big ends of connecting rod using analytical techniques and computer based mechanism simulation tools. FEA was then performed using the results from load analysis to gain insight on the structural behavior of the connecting rod and to determine the design loads for optimization. The following conclusions can be drawn from this study.

I.Design & analysis of connecting rod using Under Different Material Condition by Sasank Shekhar Panda from International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181,Vol. 10 Issue 04, April-2021,

In this paper, it is observed that Piston cylinder, crank, connecting rod, crank shaft and so on are the additives of IC engine in which connecting rod is the primary component. In IC engine maximum stressed component is connecting rod. A connecting rod act as a lever armby transmitting motion from piston to crankshaft. In this paper it is obeserved that This is a general study on the connecting rod made up of different materials along with the design of connecting rod. The main objective of this paper is to optimize weight and get material for making the connecting rod that has a better lifespan and withstands extreme loading conditions

J.Finite Element Analysis and Design Optimization of Connecting Rod by Ashwini Mane from Int. Journal of Engineering Research and Application ISSN : 2248-9622, Vol. 6, Issue 7, (Part -1) July 2016, pp.64-68

In this paper, it is observed that The objective of this study is to improve the design of connecting rod of single cylinder four stroke Otto cycle engine by shape optimization. The main objective of this study is weight reduction of connecting rod and improving its performance without affecting its functionality. Finite element analysis is one of the most important tools of CAD/CAM CAE. For this study ANSYS analysis software is used for modeling, analysis and shape design optimization

In this paper, it is observed that Existing design and optimized (modified) design is analyzed on similar platform. Similar platform means the loading conditions are exact same in both of the design. The additional parameters such as meshing methodology and mesh size is tried to keep with minimum deviation. Designs are meshed with all tetrahedral higher order element with same body size of 1 mm. Special care needs to take at filets, because these are critical stress location areas as per mentioned in [6] Mesh size across all the fillet is considered a 0.5 mm to get more accurate results

3.GEOMETRICMODELING:

3.1 INTRODUCTIONTO ANSYS DESIGN MODELER:

To study Ansys Design Modeler Geometry We will focus on following fundamental areas like Geometry import, Sketch Mode and 3D Geometry

Geometry import is nothing but importing geometry from other programs like Solidworks, Catia etc in STEP or IGS format. Sketch Mode is drawing sketch in 2D as per specifications and Convert them into 3D Models. 3D Geometry derived from Sketch entities such as extrusions, revolves and surface models etc

Length Units: When a new DM session is started a dialog box allows selection of the

Desired length unit can be set as default and units cannot be changed in mid-session.

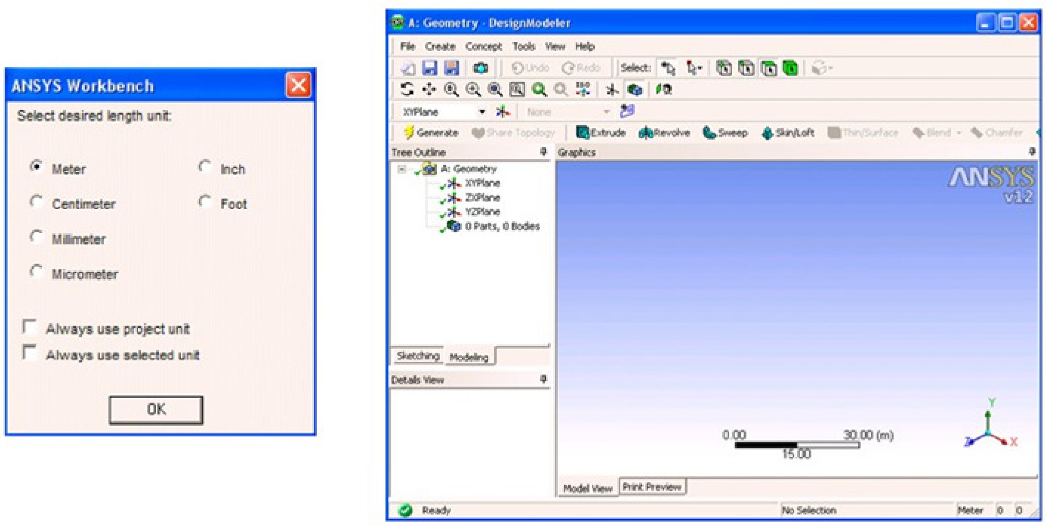
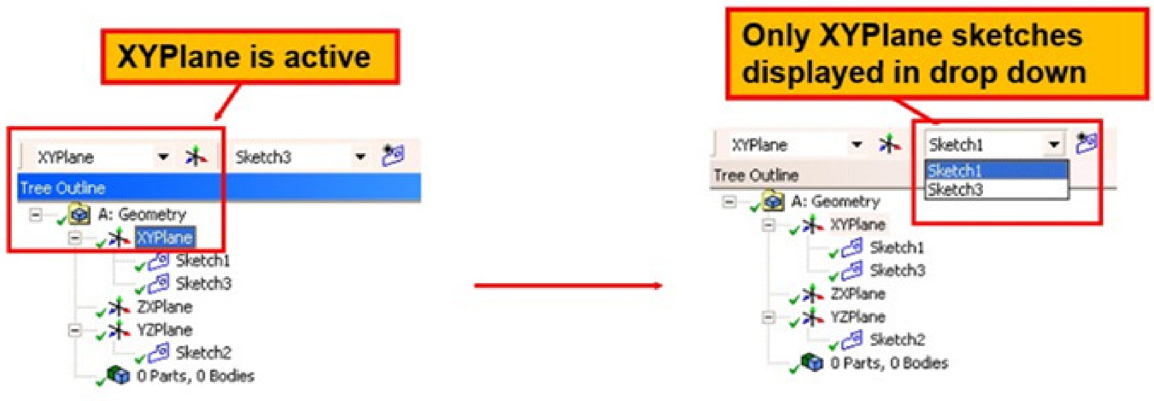


Figure 5: Ansys Design Modeler

DM sketches are created on planes. In a new DM session there are three default orthogonal planes in place at the global origin xy, zx and yz. Users may create and position new planes as needed by defining origin. Users can create as many planes as needed.

New Sketch creates new one on the active plane. There are placed in Tree beneath their associated plane. Drop down list references sketches on currently active plane.



Shortcut to create a new from face plane and sketch using existing geometry:

Highlight desired surface to place new plane

Switch to the sketch tab and begin sketching

New Plane and sketch are automatically created

Figure 6: Sketching Modeling

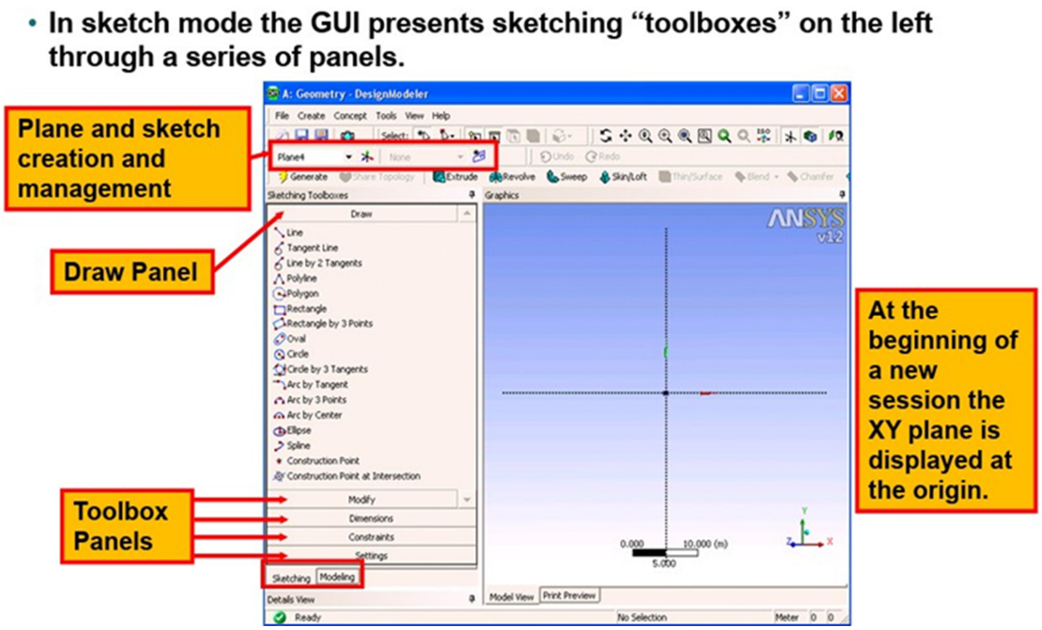


Figure 7: Sketching ToolBox

By default DM is in auto-constraint mode. Auto constraints allow new sketch entitites to automatically snap to a location or orientation. The cursor indicates the kind of constraint that will be applied.

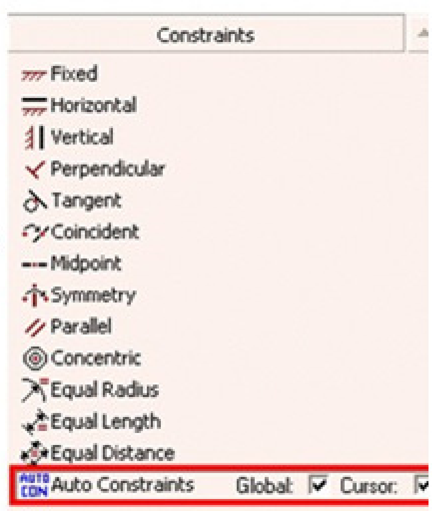
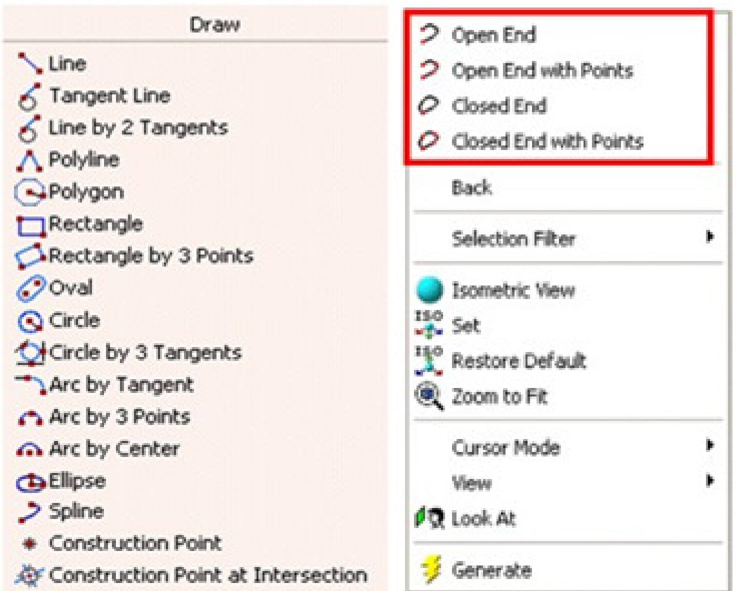


Figure 8: Constraints

Once a plane and sketch have been specified you can begin creating new geometry from the draw tool box. Remember some operations will require a right click to complete



Move function allows placement of dimension to be modified. Animate to view dynamic changes applied to the selected dimension. Dimensions can be displayed as the dimension can be displayed as the dimension value and or name.

3.2 3D MODELLING:

Bodies and parts

3D Features

Boolean Operations

Feature Direction

Feature Type

Primitives

Design Modeler is primarily intended to provide geometry to an analysis environment. For this reason we need to see how DM treats various geometries. Design Modeler contains three different body types like solid body – body has surface area and volume. Surface body-body has surface area but no volume. Line body-body consists entirely of edges, no area, and no volume.

By default, DM places each body into one part by itself. Individual parts will always be meshed separately. If bodies in separate bodies share faces, the meshes on those shared faces will not be matched. Multiple bodies in a single part will have matched meshes on shared faces when meshed.

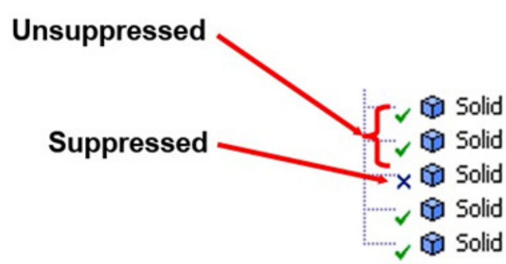
By default, DM will merge new geometry with existing geometry to maintain a single body.

This can be controlled by working with either frozen or active bodies

You can toggle between frozen and active states for using the Freeze and Unfreeze tools.

- There are <u>two body states</u> in DM:
  - *Active*:
    - Body can be modified by normal modeling operations (cannot be sliced)
    - Active bodies are displayed in blue in the Feature Tree View
    - The body's icon in the Feature Tree View is dependent on its type - solid, surface, or line
  - *Frozen*: (>Tools>Freeze)
    - Main Purpose:
      - Provides alternate method for Assembly Modeling.
    - A Frozen body is immune to all modeling operations except slice, blend, chamfer, face delete and split edges.
    - To move all active bodies to the Frozen state, use the Freeze feature (freeze applies to all)
    - To move *individual* bodies from the frozen to active, select the body and use the Unfreeze feature.
  - Frozen bodies are displayed as transparent in the Tree View.

- Body Suppression:
  - Suppressed bodies are not plotted.
  - Suppressed bodies are not sent to other Workbench modules for meshing or analysis, nor are they included in the model when exporting to a Parasolid file (.x_t).
  - In the tree outline an "X" is shown for each suppressed body

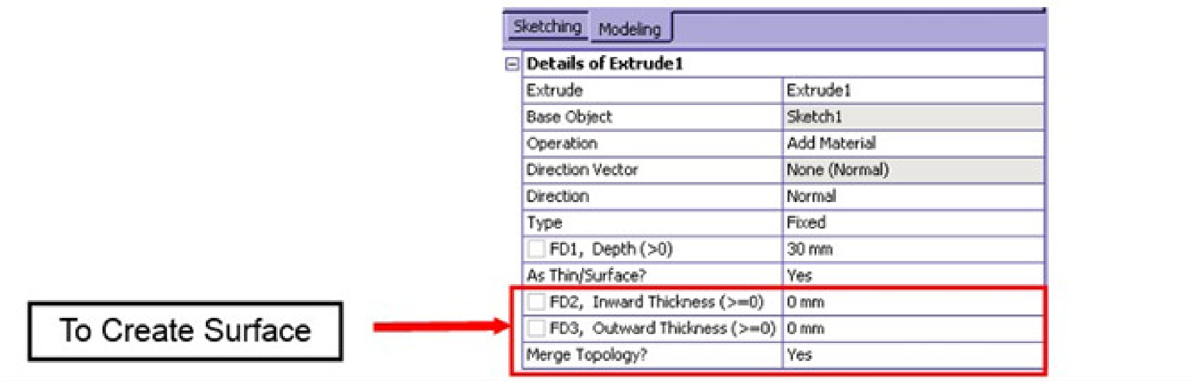


3.2.1 3D FEATURE:

3D features are Extrude, Sweep, Revolve, Skin/Loft, Thin/Surface.

The effect of the feature creation is determined by the type of the feature, the Boolean operations performed as it is created, and the extent of the feature – fixed, to next, through all etc.

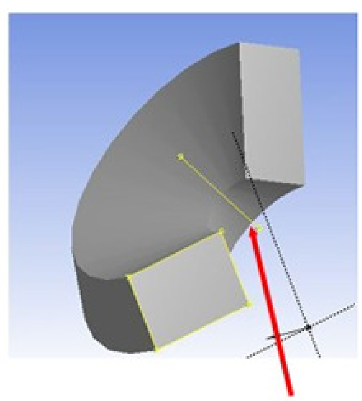
- **Extrusions:**
  - **Extrusions include solids, surfaces, and thin-walled features**
    - **To create surfaces, select "as thin/surface" and set the inner and outer thickness to zero**
  - **The active sketch is the default input but can be changed by selecting the desired sketch in the Tree View**
  - **The Detail View is used to set the Extrude depth, direction, and Boolean operation (Add, Cut, Slice, Imprint, or Add Frozen)**
  - **The Generate button completes the feature creation**
  - **Note: the section on Feature Type shows various extrusion examples**



Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Type	Fixed
FD1, Depth (>0)	30 mm
As Thin/Surface?	Yes
FD2, Inward Thickness (>=0)	0 mm
FD3, Outward Thickness (>=0)	0 mm
Merge Topology?	Yes

3.2.2 REVOLVE:

- **Revolve:**
  - **Active sketch is rotated to create 3D geometry**
  - **Select axis of rotation from details**
    - **If there is a disjoint (free) line in the sketch, it is chosen as the default axis of revolution**
  - **Direction Property for Revolve:**
    - **Normal: Revolves in positive Z direction of base object**
    - **Reversed: Revolves in negative Z direction of base object**
    - **Both - Symmetric: Applies feature in both directions. One set of angles will apply to both directions**
    - **Both - Asymmetric: Applies feature in both directions. Each direction has its own angle property**
  - **The Generate button completes the feature creation**

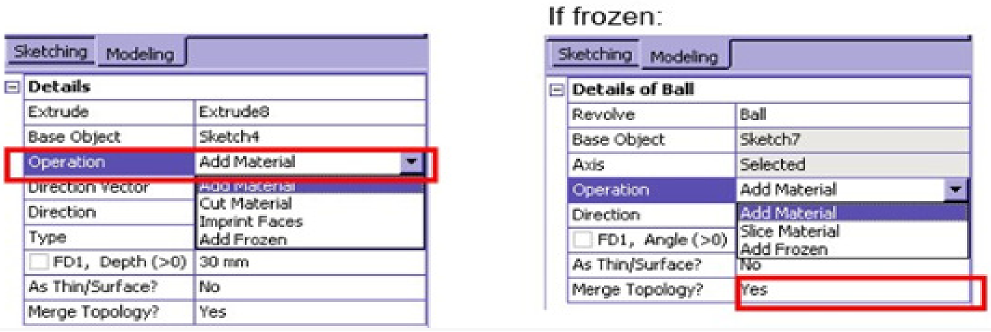


Sketch with Disjoint Line

3.2.3 BOOLEAN OPERATIONS:

You can apply five different Boolean operations to 3D features:

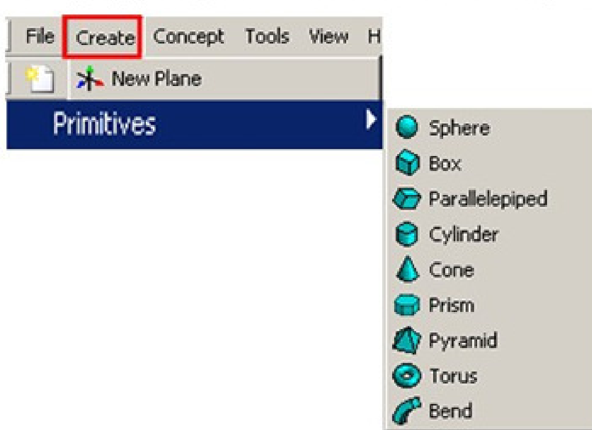
- **Add Material : creates material and merges it with the *active* bodies.**
  - **It is always available**
- **Cut Material: removes material from *active* bodies**
- **Slice Material: slices *frozen* bodies into pieces.**
  - **Available only when ALL bodies in the model are frozen**
- **Imprint Faces: Similar to Slice, except that only the faces of the bodies are split, and edges are imprinted if necessary (no new bodies created)**
- **Add Frozen: Similar to Add Material, except that the feature bodies are not merged with the existing model but rather added as *frozen* bodies**
  - **Line bodies are immune to Cut, Imprint, and Slice operations**

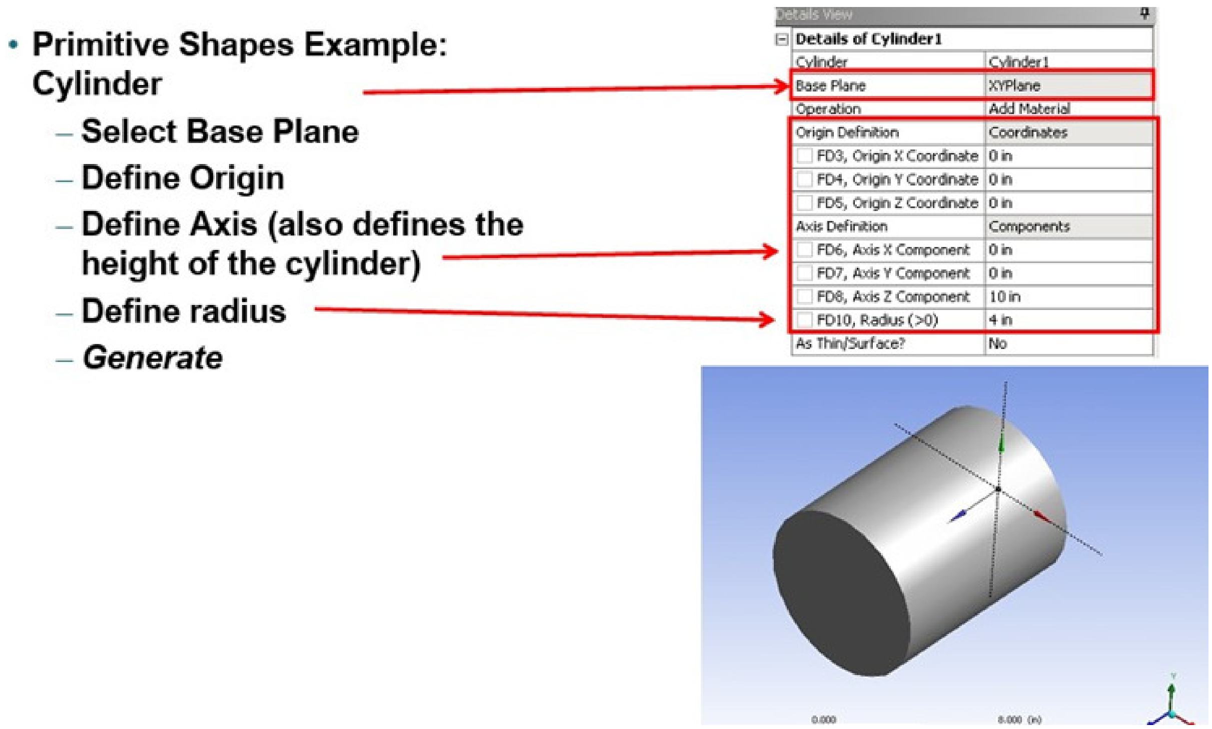


3.3.4 PRIMITIVES:

Primitive Shapes: *Create>Primitives*

- **Quickly create models by defining primitive shapes like spheres, cylinders etc..**
- **Does not require sketches**
- **Requires a *Base Plane* and several point and / or direction inputs**
- **Inputs can be defined by typing in coordinates or by selecting existing geometry.**





3.3 GEOMETRIC MODELLINGOF THE CONNECTING RODINANSYS DESIGN MODELER:

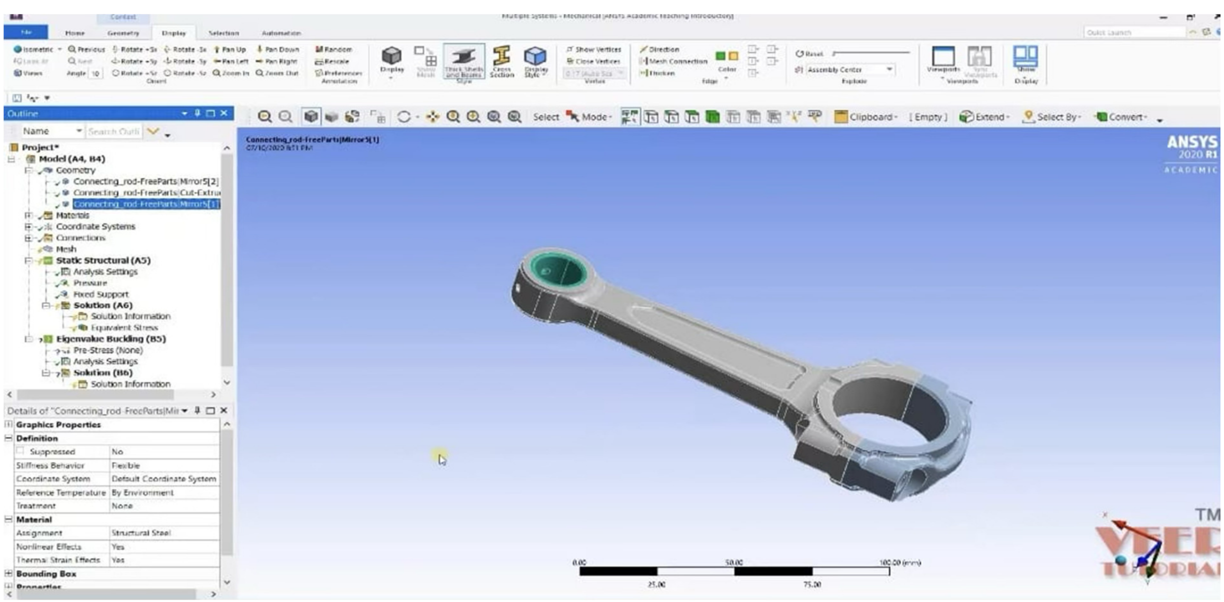


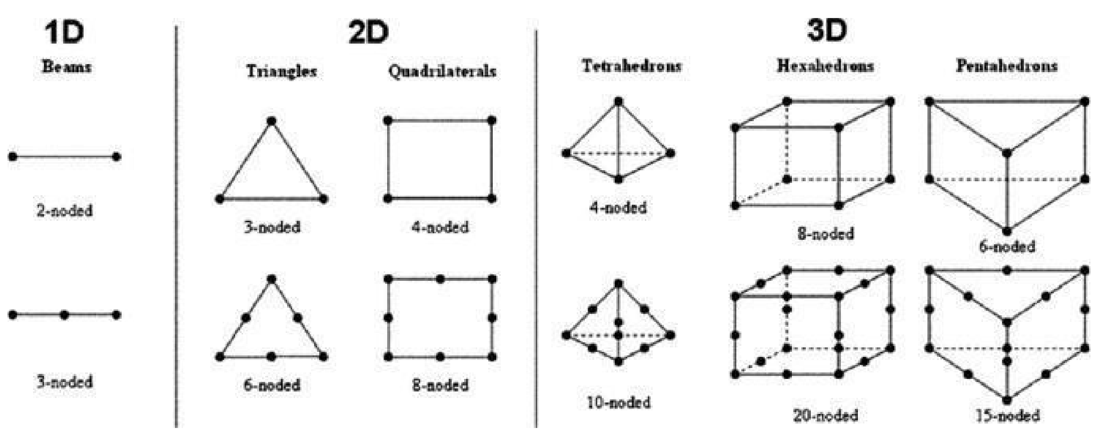
Figure 9: Geometeric Modelling of Connecting Rod

4. INTRODUCTION TO FINITE ELEMENT ANALYSIS

Introduction to Finite Element Analysis:

Finite Element Methods instils the need for comprehensive evaluation and checking when interpreting results. Engineering is at the heart of modern life. Today, engineers use computers and software in the design and manufacture of most products, processes and systems.

Finite element analysis (FEA) is an indispensable software tool in engineering design, and indeed in many other fields of science and technology. The essence of Finite Element Analysis; what is it and why do we carry out FEA?As an example of its use, we will look briefly at the case of finite element analysis



Many FEA programs also are equipped with the capability to use multiple materials within the structure such as:

- Isotropic, identical throughout

- Orthotropic, identical at 90 degrees
- General anisotropic, different throughout

4.2 Types of Engineering Analysis:

Structural analysis: It consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation as in**.**

Vibrational analysis: It is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the material which, in turn, may cause resonance and subsequent failure. Fatigue analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material.

Heat Transfer analysis: The conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermo properties in the material that yield linear heat diffusion.

Linear and nonlinear structural analysis: To determine the behavior of the structure under specific conditions.

Fatigue analysis: To determine the lifespan of the design.

Results of Finite Element Analysis:

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

In practice, a finite element analysis usually consists of three principal steps:

1. Preprocessing: The user constructs a model of the part to be analysed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical “preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

2. Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

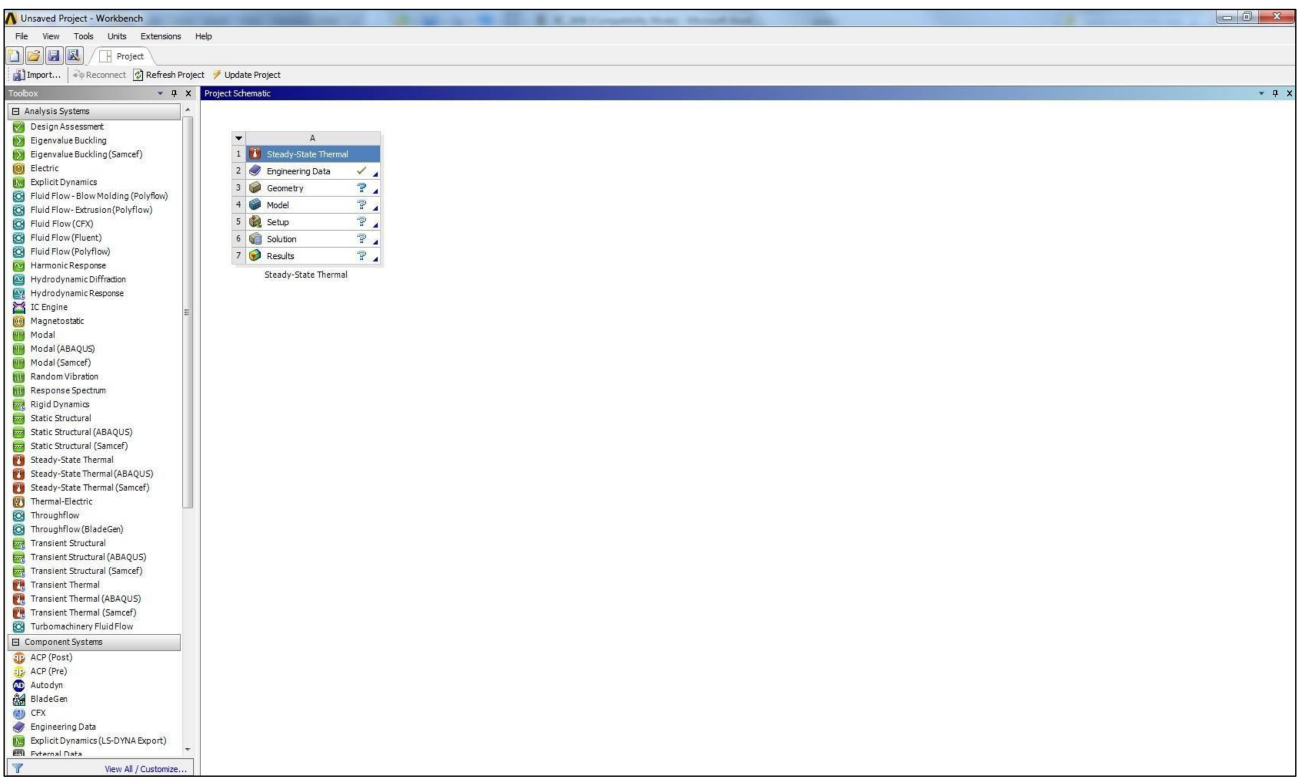
$$Kijuj = fi$$

Where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

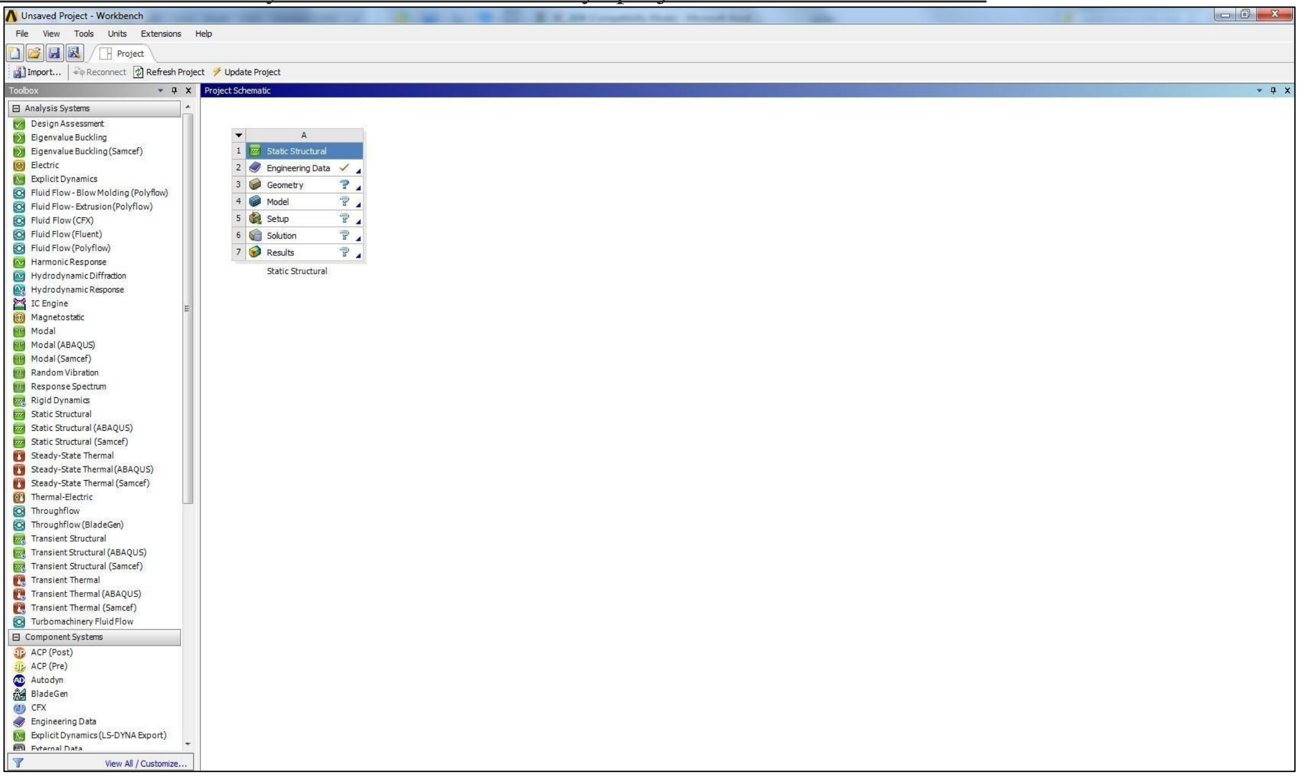
3. Post processing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

<u>Introduction to the Ansys Workbench Project Schematic:</u>

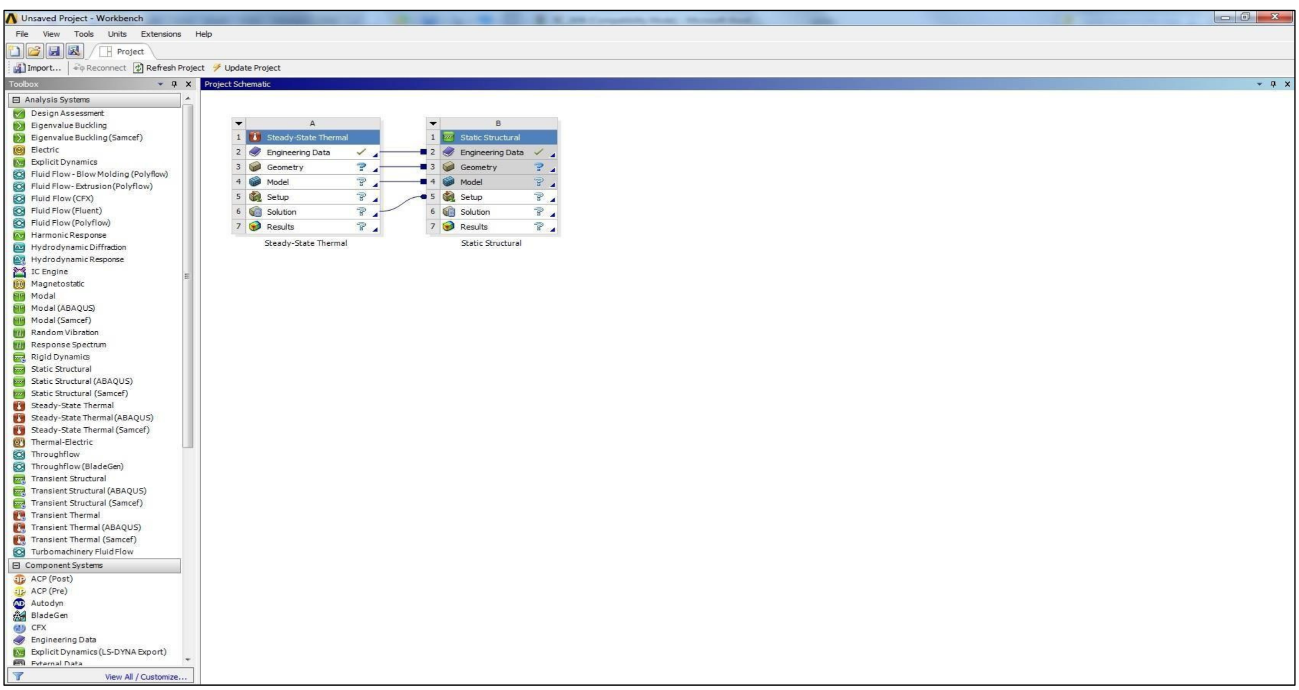
State-state thermal analysis is carried out in Ansys project schematic as follows:



Static Structural analysis is carried out in Ansys project schematic as follows:



Coupled field analysis – Combination of the Steady State thermal analysis and Static Structural Analysis is as follows:



Organization of the ANSYS program:

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or routine) level

Begin level acts as a gateway into 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 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 solutions results at specific points in the model as a function of time.

Material Models:

ANSYS permits a few diverse material models like:

Linear flexible material models (isotropic, orthotropic, and anisotropic).

- Non-direct material models (hyper flexible, multi straight versatile, inelastic and Viscos flexible)
- Heat moves material models (isotropic and orthotropic)
- Temperature subordinate material properties and creep material models.

Loads:

The word stacks in ANSYS phrasing incorporates limit conditions and remotely or inside applied compelling capacities, as represented in Loads. Instances of burdens in various orders are:

Structural: Removals, powers, pressures, temperatures (for warm strain), Gravity.

Thermal: temperatures, heat stream rates, convections. Inner warmth age, limitless surface.

ANALYSIS TYPES:

The accompanying kinds of investigation are conceivable utilizing ANSYS

1.Structural Analysis: Static Analysis

2. Thermal Analysis: Steady state thermal analysis

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.
- Fluencies (for nuclear swelling).

System Configurations:

In the current work, the computational mathematical examination is finished utilizing ANSYS adaptation 16.0 run as standalone or workbench-host application, having 4GB RAM and 1TB hard disk with Windows 11 working framework.

5. RESULTS & DISCUSSIONS

Analysis on connecting rod by using ansys 2024work bench software.The analysis of connecting rod models are carried out using ANSYS software using Finite Element Method. Firstly the model files prepare inthe SOLIDWORKS SOFTWARE

Ⅰ.Geometric Model afterbeing imported intoAnsysWorkbench :

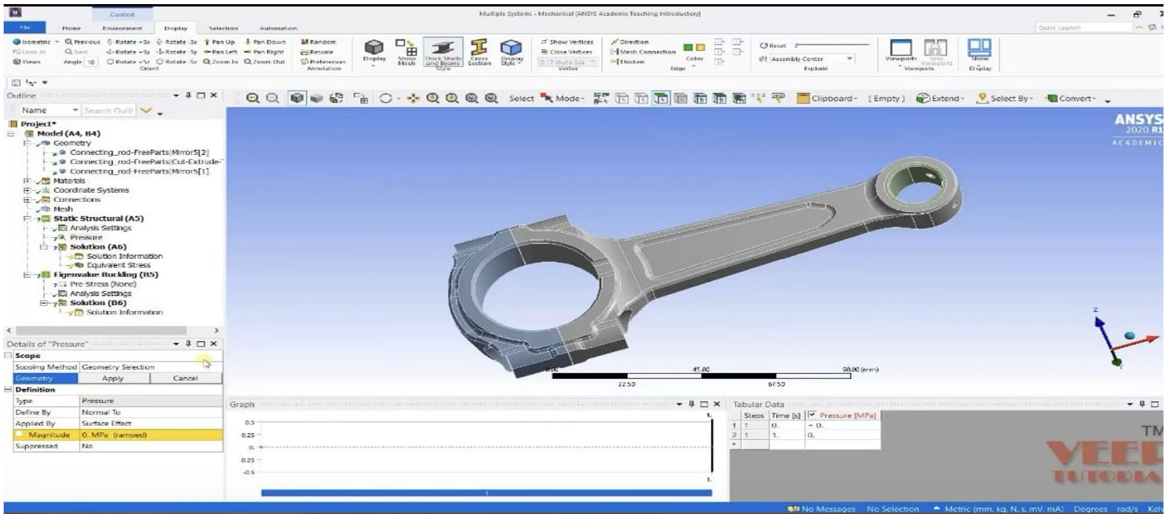


Figure15 : GeometricModelafterbeingimportedintoAnsys

Property			
Material Data	**Young'sModulus (Pa)**	**Poisson Ratio**	**Density Kg/m3**
AluminumAlloy	0.7E+11	0.33	2770
Titanimum Alloy	9.6E+10	0.36	4620
42CrMo4	21E+11	0.30	7830
Aluminum metal matrix(kS1275)	9.8E+10	0.33	2800

Table1:Material Properties

Load & fixed support:

Fixed support

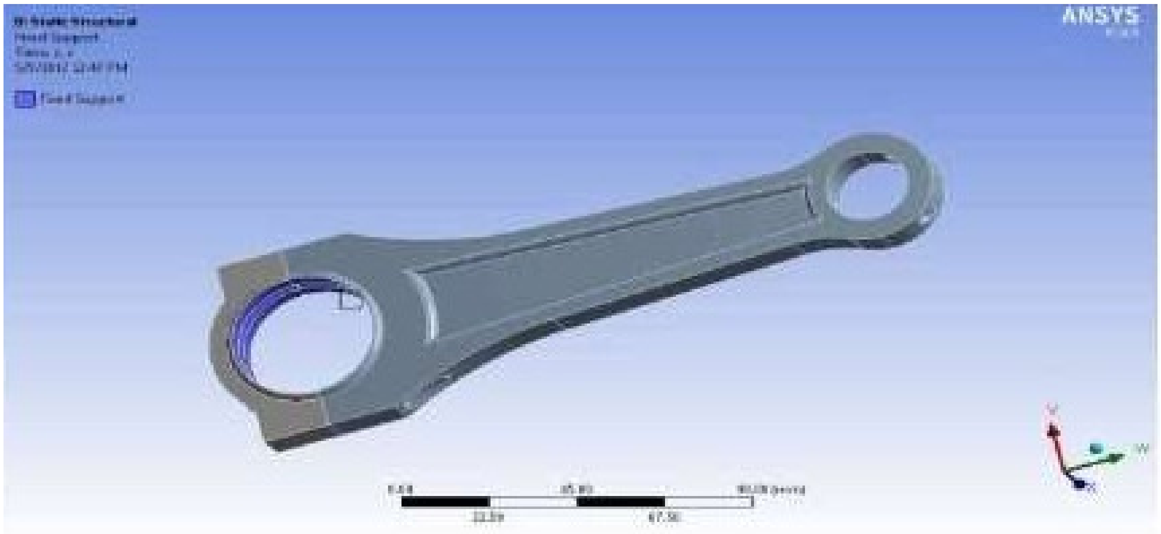


Figure 16.a: Load Support

Load

Load at 16 MPA

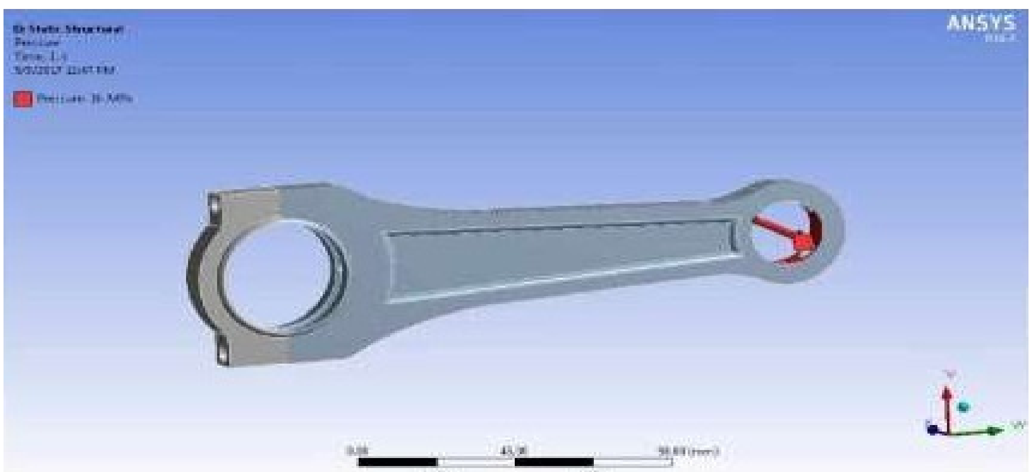


Figure 16.b: Fixed Support

MESHING:

Meshing is probably the most important part in any of the computer simulations, because it can show drastic changes in results you get. Meshing meansyou create a mesh of some grid-points called 'nodes'.It's done with a variety of tools & options available in the software. The results are calculated bysolving therelevant governing equations numerically at each of the nodes of the mesh. The governing equations arealmost always partial differential equations, and Finite element method is used to find solutions tosuch equations. The pattern and relative positioning of the nodes also affect the solution, the computational efficiency & time.

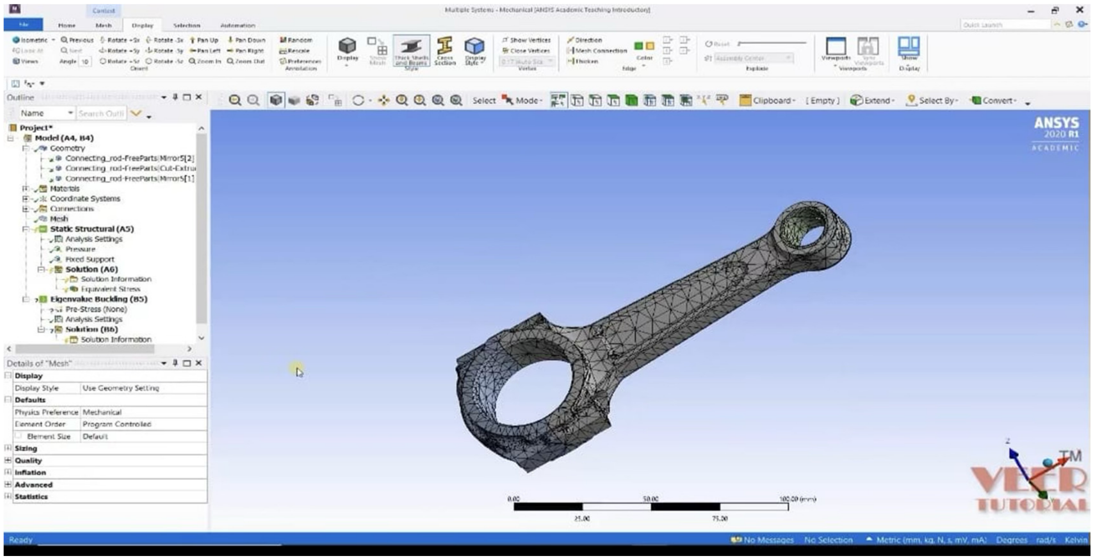


Figure 17 : Meshing Of a Connecting Rod in ANSYS

Mesh Type: Tetrahedral
No. of nodes: 16190
No. of elements: 8821

STRUCTRUAL ANALYSIS RESULTS

Material: Aluminium Alloy

Maximum Stress

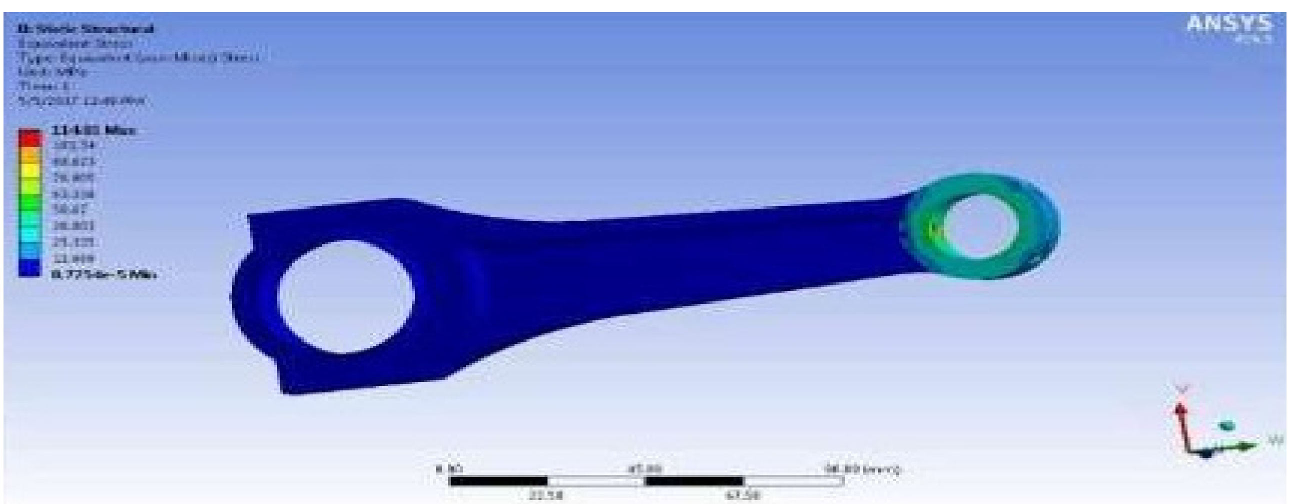


Figure 18.a: Maximum Stress of Aluminium Alloy

Total Deformation

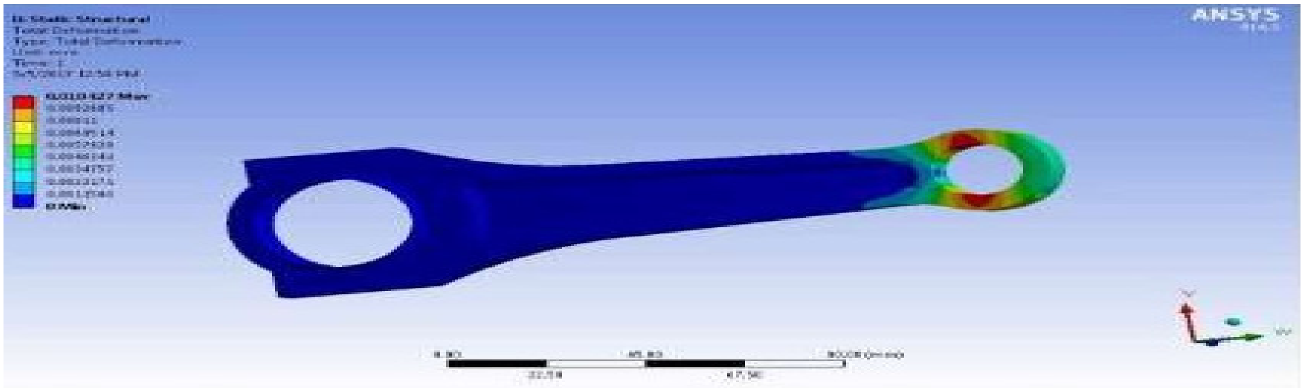


Figure18.b: Total Deformation of Aluminium Alloy

Maximum Strain

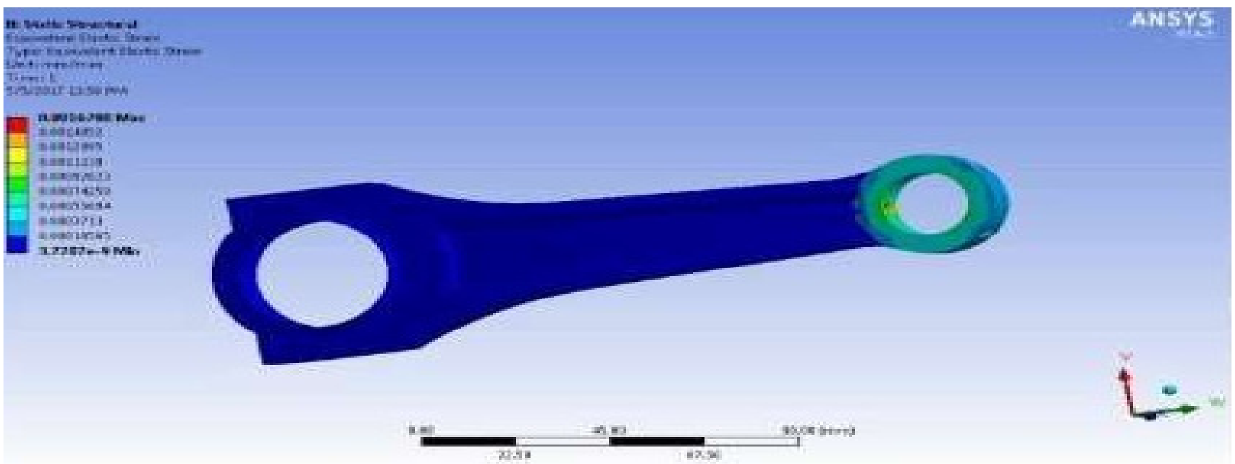


Figure 18.c: Maximum Strain ofAluminium Alloy

Material: Titanium Alloy

Maximum Stress

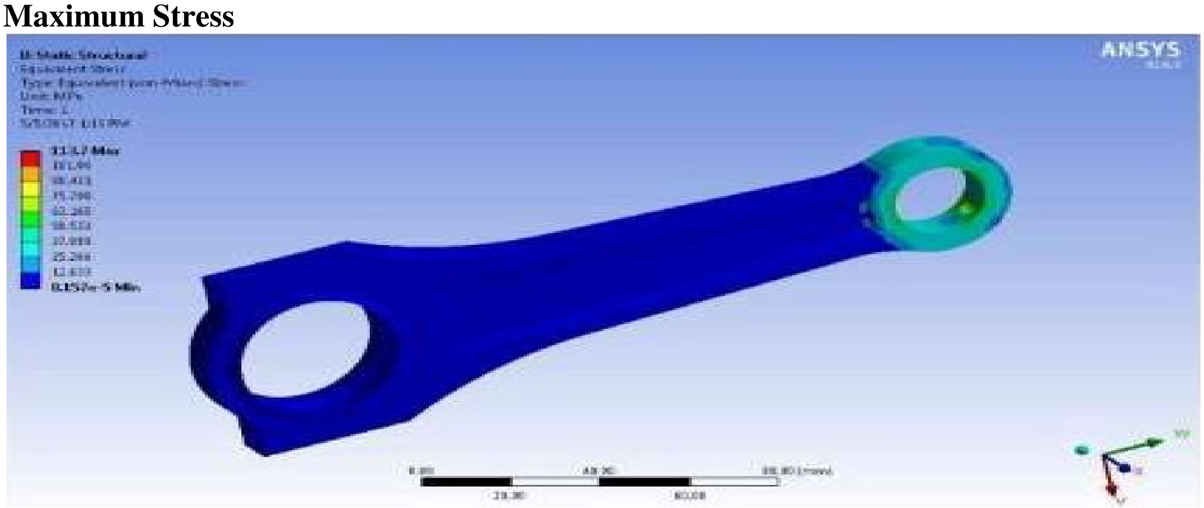


Figure 19.a: Maximum Stress of Titanium Alloy

Total Deformation

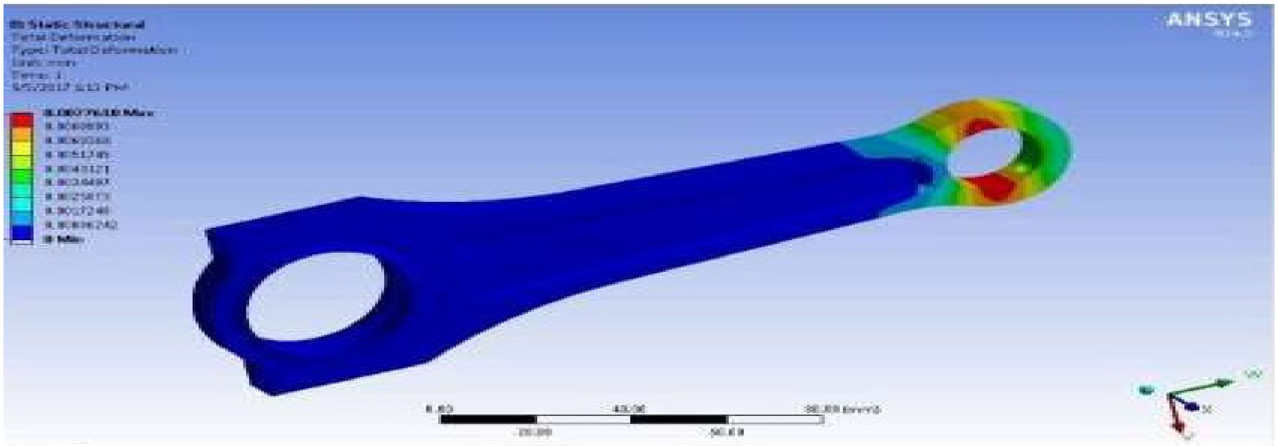


Figure 19.b: Total Deformation of Titanium Alloy

Maximum Strain

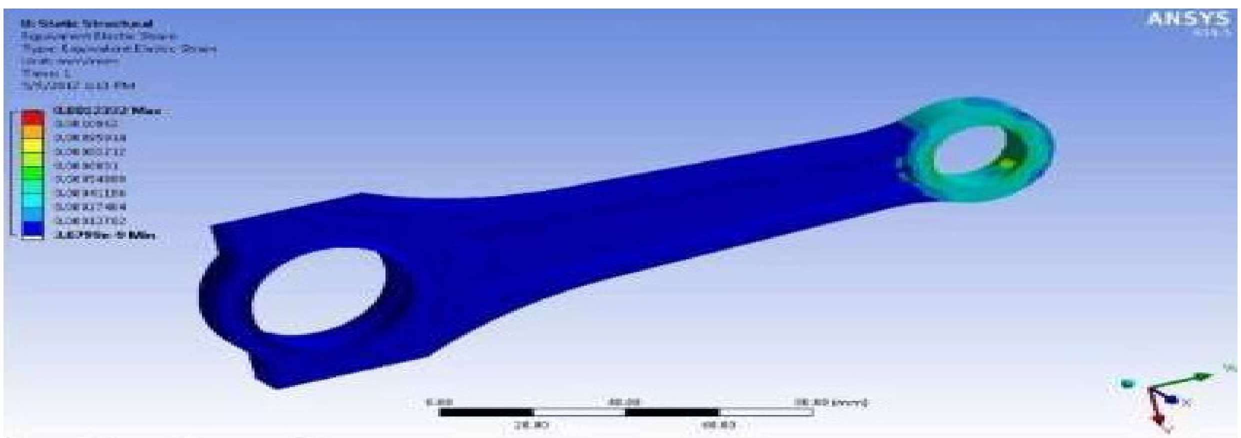


Figure 19.c: Maximum Strain of Titanium Alloy

Material: 42CrMo4

Maximum Stress

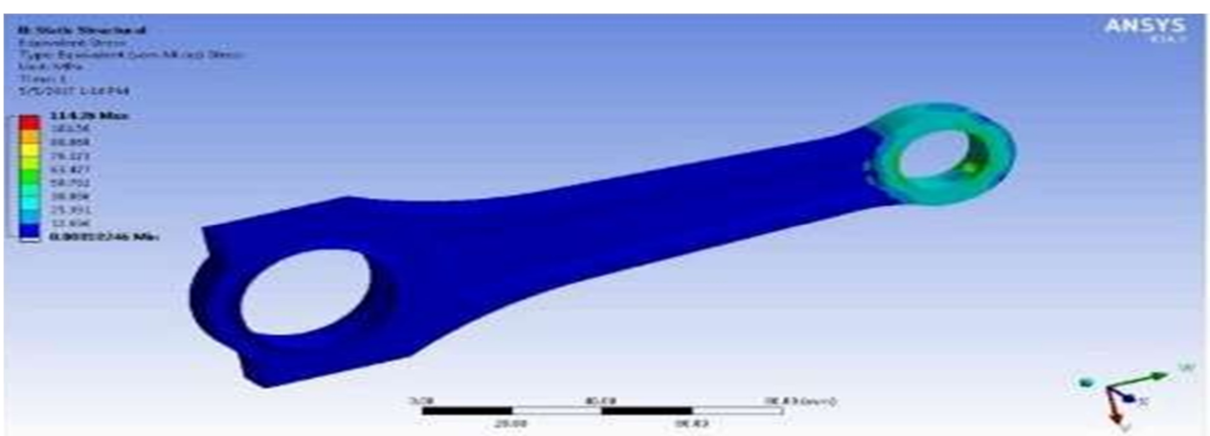


Figure 20.a: Maximun Stress of 42CrMo4

Total Deformation

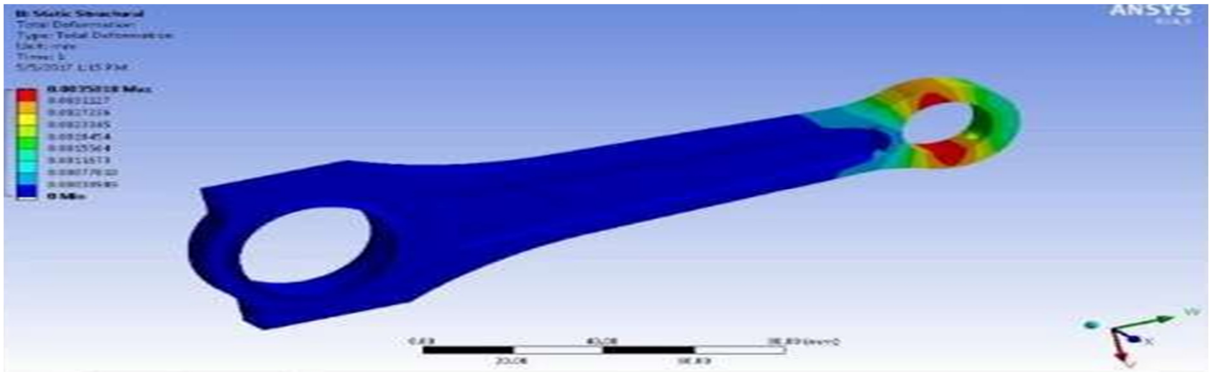


Figure20.b: Total Deformation of 42CrMo4

Maximum Strain

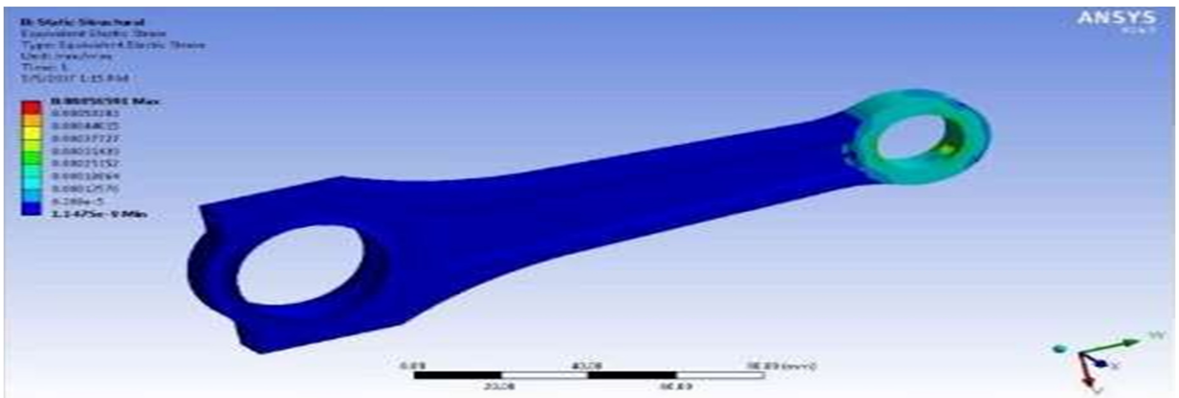


Figure 20.c: Maximum Strain of 42CrMo4

Material: Al Metal Matrix

Maximum Stress

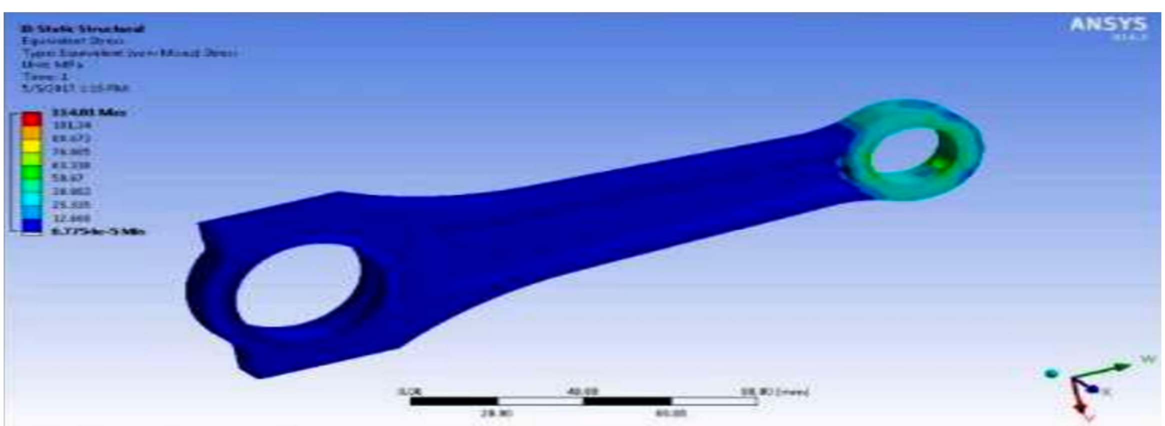


Figure 21.a:Maximum Stress of Al Metal Matrix

Maximum Strain

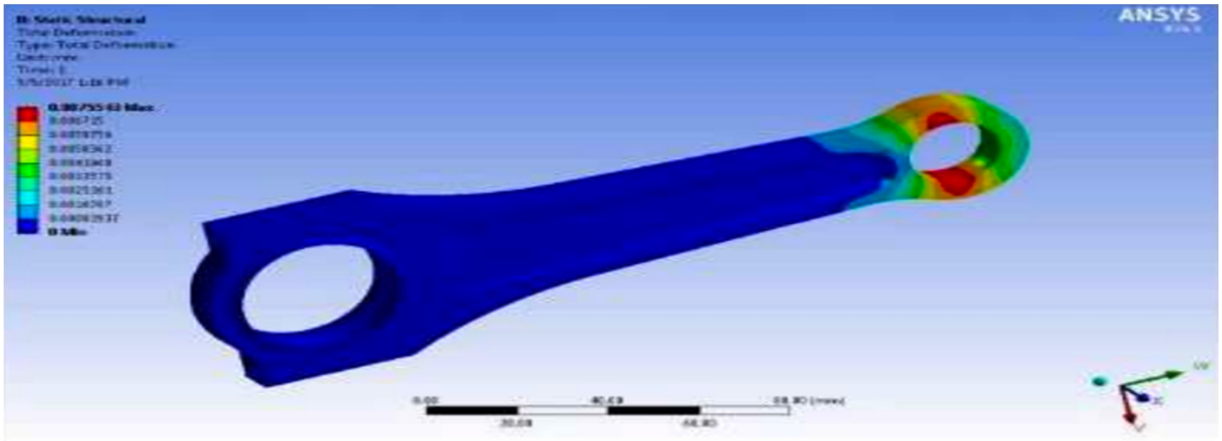
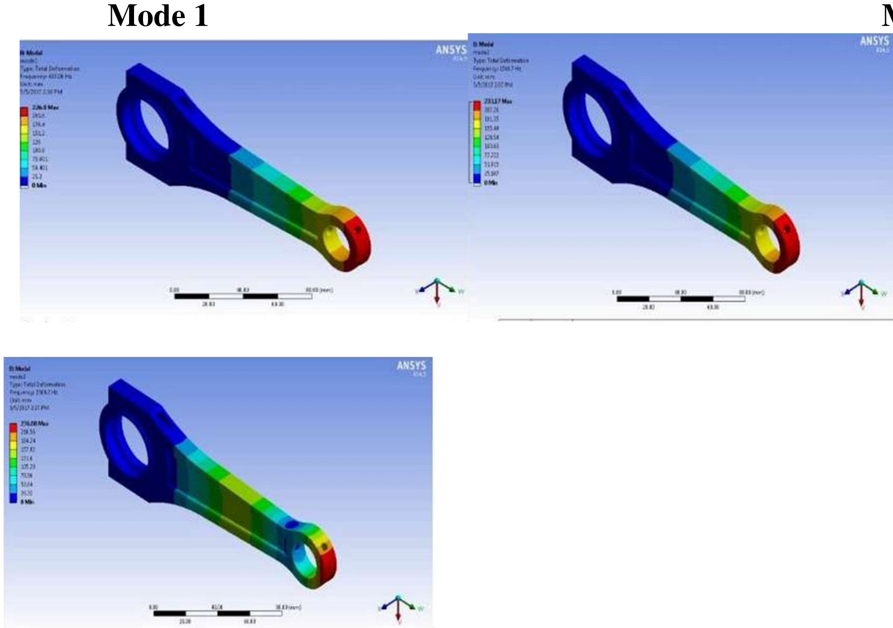


Figure 21.b:Maximum Stress of Al Metal Matrix

MODAL ANALYSIS RESULTS

Material: Aluminium Alloy

Mode 1 **Mode 2**



Mode 3

Figure 22: Different Modes of Aluminium Alloy

RESULT TABLE

Stuctural Analysis:

Materials	Max Stress (N/mm2)	Total Deformation (mm)	Max strain
Aluminum Alloy	114.01	0.010427	0.0016708
Titaniumn Alloy	113.70	0.0077618	0.0012332
42CrMo4	114.26	0.0035018	0.00056591
Aluminum metal matrix (KS1275)	114.01	0.0075018	0.0012105

Table2:MaterialData

Model Analysis:

Materials	Model 1		Mode 2	
	Deformation	**Frequency**	**Deformation**	**Frequency**
Aluminum Alloy	226.8	437.06	233.17	1560.7
Titaniumn Alloy	175.71	394.4	180.44	1404.1
42CrMo4	134.83	446.17	138.77	15897.
Aluminum metal matrix (KS1275)	225.58	510.72	231.91	236.88

Table 3: Model Analysis Data

- For comparisons of the results obtained From the static analysis result tables it is concluded that 42CMo4 show least stress and least deformation & strain value on same static load condition.
- From the Modal analysis result tables it is concluded that 42CMo4 shows Less deformation results for given frequency. Hence for both Structural and Modal Analysis 42CrMo4 (Special Alloy Steel) it is best suitable material for connecting rod.

6.1 CONCLUSION:

Brief study about connecting rod & it's working is done in this project

- Modeling and analysis of connecting rod is done.Modeling of connecting rod is done in solid works 2016 design software.
- The file is saved as igs to import in ansysworkbench .The analysis in ansys is extremely important prior to the fabrication of connecting rod.
- The static structural, & Dynamic analysis (modal analysis) has carried out in the ansys 14.5 software package for connecting rod by different materials like alaninum alloy.
- Titanium alloy 42crmo4 (special steel alloy) and Aluminum metal matris (KS1275) .The material properties and brief explanation about composites has given.
- The utmost stress, strain and defirmation values of static analysis are tabulated on load condition of 16 MPA.
- From result we conclude that on given load condition of 16MPA Titanium alloy showing less deformation, while special alloy steel 42CrMo4 showing less deformation vaine.
- Hence the materials with low stress values are also preferable for the fabrication of connecting rod.As titanium is very costly material its application is limited can only use for aero space industries.

- Aluminum metal matrix which is less costly and less weight ratio showing nearly same stress value of titanium and less deformation. value after 42CrMo4 can consider best material for automobile other than general material.

- Model analysis (Dynamic) is performed on three different modes; deformation values with respect to different freqencies are noted and tabulated for each material.

7. REFERENCES

[1] Kuldeep B, Arun L.R, Mohammed Faheem, "Analysis and optimization of connecting rod using ALFASiC composites", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, pp 2480-2487, June 2013.

[ii] AmbrishTiwari, Jeetendra Kumar Tiwari, Sharad Kumar Chandrakar, "Fatigue Analysis of Connecting Rod Using FEA to Explore Weight and Cost Reduction Opportunities or a Production of Forged Steel Connecting Rod", International Journal of Advanced Mechanical Engineering, Volume 4, Issue 7, pp. 783-802, 2014.

[ii] A.K. Bagha et al. Finite element analysis of VGCF/pp reinforced square representative volume element to predict its mechanical properties for different Mater. Today Proc. (2 0 2 1)

[iv] S.S. Godara et al. Analysis of frontal bumper beam of automobile vehicle by using carbon Mater. Today Proc. (2 0 2 0)

[v] Leela Krishna Vegi, Venu Gopal Vegi, "Design And Analysis of Connecting Rod Using Forged steel", International Journal of Scientific & Engineering Research, Volume 4, Issue 6, June -2013 ISSN 2229-5518.

[vi] Afzal, A. and A. Fatemi, 2004. "A comparative study of fatigue behavior and life predictions of Aluminum steel and PM connecting rods". SAE Technical Paper.

[vii] Chen, N., L. Han, W. Zhang and X. Hao, 2006. "Enhancing Mechanical Properties and Avoiding Cracks by Simulation of Quenching Connecting Rod". Material Letters, 61: 3021-3024.

[viii] El – S ed, M.E.M. d E.H. Lu d, ŗşşŖ. âS uc u l optimiz io i h f igue life co s i s,È Engineering Fracture Mechanics, 37(6): 1149-1156.

[xxx] Jahed Motlagh, H.M. Nouban and M.H. Ashraghi, 2003. "Finite Element ANSYS". University of Tehran Publication, PP: 990.

[x] RS KLhurmi, JK Gupta, “machine Design”, S.Chand Publications, New Delhi, 4th edition, 2005.

[xi]S.Rammamrutham, “Strength of Materials”, Dhanpatrai & Sons, New Delhi, 14th edition, 2004

[xii] Mukesh Kumar, Veerendra Kumar” Finite Element Analysis of I.C Engine Connecting Rod: A Review”,International Journal Of Engineering Sciences & Research Technology July, 2014

[xii] Abhinavgautam, K Priya Ajit “Static Stress Analysis of Connecting Rod Using FEA Approach”, Journal of Mechanical and Civil Engineering (IOSR-JMCE) Volume 10, Issue 1 (Nov. - Dec. 2013)

[xiv] F. Desai, K. Jagtap and A Deshpande, Numerical and Experimental Analysis of Connecting Rod, International Journal of Emerging Engineering Research and Technology, 4(2), 2014, PP 242-249

[xv] R. Yang, D. Deversit and J. Alliset, Shape optimization of connecting rod pin end using genetic model. Ford Motor Company, 23400 Michigon Avenue, vp1100,dearbon,mi48124, USA.1992

[xvi] MarthanapalliHariPriya and K. Manohar Reddy “Materialized Optimization of Connecting Rod for Four Stroke Single Cylinder Engine”, International Journal of Computational Engineering Research, Volume, 03, Issue 10, October 2013

[xvii] Vikas Singh, Sumit Kr. Verma, Harish Chandra Ray, Vishal Kr. Bharti, and AbhineshBhaskar “Design and Analysis of Connecting Rod for Different Material Using Ansys Workbench 16.2”, International Journal for Research in Applied Science &EngineeringTechnology (IJRASET), Volume 5, Issue V, May 2017

[xviii] Prof. N.P.Doshi, 1 Prof .N.K.Ingole "ANALYSIS OF CONNECTING ROD USING ANALYTICAL AND FINITE ELEMENT METHOD"INTERNATIONAL JOURNAL OF MODERN ENGINEERING RESEARCH" (IJMER) www.ijmer.com Vol.3, Issue.1, Jan Feb. 2013, ISSN: 2249-6645.

[xi]FOLGAR Analysis of Connecting Rod Using Analytical and Finite Element" International Journal of Modern Engineering Research (IJMER), JanFeb. 2013, Vol.3, Issue.1, pp-65-68

[xx] Shenoy, P. S. and Fatemi, A. Connecting rod optimiz- ation for weight and cost reduction. SAE Technical Paper 2005-01-0987, 2005.