International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 14, Issue 1 [January. 2025] PP: 58-80

Rigid Body Dynamics And Shape Optimization Analysis Of Connecting Rod

Dr. C. Dhayananth Jegan[1], S. Ajith kumar[2], Amuthan T R[3], Kannan R[4], Abish R[5]

[1]Associate Professor, [2]Assistant Professor, [3,4,5]UG Student
[1,2,3,4,5]Department of Mechanical Engineering
Stella Mary's College of Engineering
Aruthenganvilai, Azhikal Post, Kanya Kumari District, 629202,
Tamil Nadu
Corresponding Author S. Ajith Kumar

Abstract*-The main function of the Connecting Rod in IC engine is to receive the force from piston and to transmit the energy to the crankshaft. The main aim of this Project is to Model & Assemble the Piston, Connecting Rod & Crankshaft for a 4-stoke air-cooled 200cc for Higher Efficiency Engine. Using Rigid Body dynamics, we can simulate the entire assembly and find out the axial load acting on connecting rod during simulation. Modeling, Assembly of Piston, Connecting rod and Crankshaft is done in PTC CREO software & Rigid body Analysis and stress analysis of connecting rod is done by ANSYS Finally we would optimize the shape of a connecting rod using shape optimization techniques in ANSYS. The aim of the optimization has been to minimize the respective Von Mises stresses which occur at connected rod in both cases, i. e. compressive loads coming from the gas pressure at maximum engine output and the bending loads resulting from the inertia force at the maximum engine power. The weight of the connecting rod should be maintained to prevent increasing of the inertia force. As a result we are going to compare the stress value for existing and proposed model of Connecting Rod*

Keywords: *Connecting Rod, Rigid Body Dynamics, Stress Analysis, Shape Optimization*

Date of Submission: 05-01-2025 Date of acceptance: 16-01-2025

I. INTRODUCTION

An internal combustion engine (ICE) is a heat engine where the combustion of a fuel occurs with an oxidizer (usually air) in a combustion chamber that is an integral part of the working fluid flow circuit. In an internal combustion engine the expansion of the high-temperature and high-pressure gases produced by combustion apply direct force to some component of the engine. The force is applied typically to pistons, turbine blades, rotor or a nozzle. This force moves the component over a distance, transforming chemical energy into useful mechanical energy. The first commercially successful internal combustion engine was created by Étienne Lenoir around 1859 and the first modern internal combustion engine was created in 1864 by Siegfried Marcus.

The term internal combustion engine usually refers to an engine in which combustion is intermittent, such as the more familiar four-stroke and two-stroke piston engines, along with variants, such as the six-stroke piston engine and the Wankel rotary engine. A second class of internal combustion engines use continuous combustion: gas turbines, jet engines and most rocket engines, each of which are internal combustion engines on the same principle as previously described. Firearms are also a form of internal combustion engine.

Internal combustion engines are quite different from external combustion engines, such as steam or Stirling engines, in which the energy is delivered to a working fluid not consisting of, mixed with, or contaminated by combustion products. Working fluids can be air, hot water, pressurized water or even liquid sodium, heated in a boiler. ICEs are usually powered by energy-dense fuels such as gasoline or diesel, liquids derived from fossil fuels. While there are many stationary applications, most ICEs are used in mobile applications and are the dominant power supply for vehicles such as cars, aircraft, and boats.

Fig1. 1 IC Engine

Typically an ICE is fed with fossil fuels like natural gas or petroleum products such as gasoline, diesel fuel or fuel oil. There's a growing usage of renewable fuels like biodiesel for compression ignition engines and bioethanol or methanol for spark ignition engines. Hydrogen is sometimes used, and can be made from either fossil fuels or renewable energy.

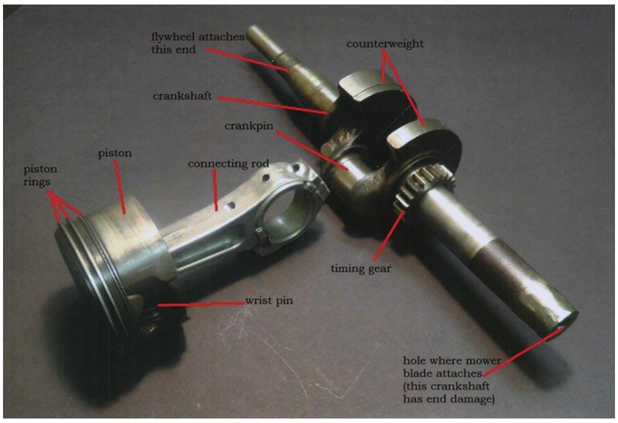


Fig1. 2 Assembly connecting rod with piston and crank shaft

1.1 TYPES OF ENGINE

1.1.1 TWO STROKE

The defining characteristic of this kind of engine is that each piston completes a cycle every crankshaft revolution. The 4 process of intake, compression, power and exhaust take place in only 2 strokes so that it is not possible to dedicate a stroke exclusively for each of them. Starting at TDC the cycles consist of:

1. **Power:** While the piston is descending the combustion gases perform work on it as in a 4-stroke engine. The same thermo dynamical considerations about the expansion apply.
2. **Scavenging:** Around 75° of crankshaft rotation before BDC the exhaust valve or port opens, and blowdown occurs. Shortly thereafter the intake valve or transfer port opens. The incoming charge displaces the remaining combustion gases to the exhaust system and a part of the charge may enter the exhaust system as well. The piston reaches BDC and reverses direction. After the piston has travelled a short distance upwards into the cylinder the exhaust valve or port closes; shortly the intake valve or transfer port closes as well.
3. **Compression:** With both intake and exhaust closed the piston continues moving upwards compressing the charge and performing a work on it. As in the case of a 4-stroke engine, ignition starts just before the piston reaches TDC and the same consideration on the thermodynamics of the compression on the charge.

1.1.2 FOUR STROKE

The top dead center (TDC) of a piston is the position where it is nearest to the valves; bottom dead center (BDC) is the opposite position where it is furthest from them. Astroke is the movement of a piston from TDC to BDC or vice versa together with the associated process. While an engine is in operation the crankshaft rotates continuously at a nearly constant speed. In a 4-stroke ICE each piston experiences 2 strokes per crankshaft revolution in the following order. Starting the description at TDC, these are

1. **Intake, induction or suction:** The intake valves are open as a result of the cam lobe pressing down on the valve stem. The piston moves downward increasing the volume of the combustion chamber and allowing air to enter in the case of a CI engine or an air fuel mix in the case of SI engines that do not use direct injection. The air or air-fuel mixture is called the charge in any case.
2. **Compression:** In this stroke, both valves are closed and the piston moves upward reducing the combustion chamber volume which reaches its minimum temperature and density increase; an approximation to this behaviour is provided by the ideal gas law. Just before the piston reaches TDC, ignition begins. In the case of a SI engine, the spark plug receives a high voltage pulse that generates the spark which gives it its name and ignites the charge. In the case of a CI engine the fuel injector quickly injects fuel into the combustion chamber as a spray; the fuel ignites due to the high temperature.Whenthe piston is at TDC, The piston performs work on the charge as it is being compressed; as a result its pressure,
3. **Power or working stroke:** The pressure of the combustion gases pushes the piston downward, generating more work than it required to compress the charge. Complementary to the compression stroke, the combustion gases expand and as a result their temperature, pressureand density decreases. When the piston is near to BDC the exhaust valve opens. The combustion gases expand irreversibly due to the leftover pressure—in excess of back pressure, the gauge pressure on the exhaust portis called the blow down.
4. **Exhaust:** The exhaust valve remains open while the piston moves upward expelling the combustion gases. For naturally aspirated engines a small part of the combustion gases may remain in the cylinder during normal operation because the piston does not close the combustion chamber completely; these gases dissolve in the next charge. At the end of this stroke, the exhaust valve closes, the intake valve opens, and the sequence repeats in the next cycle. The intake valve may open before the exhaust valve closes to allow better scavenging.

While a 4-stroke engine uses the piston as a positive displacement pump to accomplish scavenging taking 2 of the 4 strokes, a 2-stroke engine uses the last part of the power stroke and the first part of the compression stroke for combined intake and exhaust. The workrequired to displace the charge and exhaust gases comes from either the crankcase or a separate blower. For scavenging, expulsion of burned gas and entry of fresh mix, two main approaches are described: 'Loop scavenging', and 'Uniflow scavenging', SAE news published in the 2010s that 'Loop Scavenging' is better under any circumstance than 'Uniflow Scavenging.

1.2 PARTS IN CONNECTING ROD ASSEMBLY

1.2.1 PISTION

A **piston** is a component of reciprocating engines, reciprocatingpumps,gascompressorsandpneumaticcylinders,amongothersimilarmechanisms. It is the moving component that is contained by a cylinder and is made gas-tight by piston rings. In an engine, its purpose is totransfer force from expanding gas in the cylinderto thecrankshaft via a piston rod and/or connecting rod. In a pump, the function is reversed and force is transferred from the crankshaft to the piston for the purpose of compressing or ejecting the fluid in the cylinder. In some engines, the piston also acts as a valve by covering and uncovering ports in the cylinder wall.

It has five types of piston:

Trunk pistons are long relative to their diameter. They act both as a piston and cylindrical crosshead. As the connecting rod is angled for much of its rotation, there is also a sideforce that reacts along the side of the piston against the cylinder wall. A longer piston helps to support this. Trunk pistons have been a common design of piston since the early days of thereciprocating internal combustion engine. They were used for both petrol and diesel engines, although high speed engines have now adopted the lighter weight slipper piston.

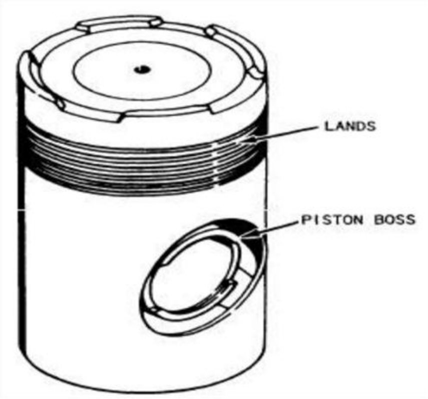


Fig1. 3 Trunk head piston

A characteristic of most trunk pistons, particularly for diesel engines, is that they have a groove for an oil ring below the gudgeon pin, in addition to the rings between the gudgeon pin and crown. The name 'trunk piston' derives from the 'trunk engine', an early design of marine steam engine. To make these more compact, they avoided the steam engine's usual piston rod with separate crosshead and were instead the first engine design to place the gudgeonpin directly within the piston. Otherwise these trunk engine pistons bore little resemblance to the trunk piston; they were extremely large diameter and double-acting. Their 'trunk' was a narrow cylinder mounted in the centre of the piston.

Crosshead pistons: Large slow-speed Diesel engines may require additional support for the side forces on the piston. These engines typically use crosshead pistons. The main piston has a large piston rod extending downwards from the piston to what is effectively a second smaller-diameter piston.The main piston is responsible for gas sealing and carries the piston rings. The smaller piston is purely a mechanical guide. It runs within a small cylinder as a trunk guide and also carries the gudgeon pin. Because of the additional weight of these pistons, they are not used for high-speed engines.

Fig1. 4 Cross head piston

A **slipper piston** is a piston for a petrol engine that has been reduced in size and weight as much as possible. In the extreme case, they are reduced to the piston crown, support for the piston rings, and just enough of the piston skirt remaining to leave two lands so as to stop the piston rocking in the bore. The sides of the piston skirt around the gudgeon pin are reduced away from the cylinder wall. The purpose is mostly to reduce the reciprocating mass, thus making it easier to balance the engine and so permit high speeds.

Fig1. 5 Slipper head piston

A secondary benefit may be some reduction in friction with the cylinder wall, since the area of the skirt, which slides up and down in the cylinder is reduced by half. However most friction is due to the piston rings, which are the parts which actually fit the tightest in the bore and the bearing surfaces of the wrist pin, the benefit is reduced.

Deflector pistons are used in two-stroke engines with crankcase compression, where the gas flow within the cylinder must be carefully directed in order to provide efficientscavenging. With cross scavenging,the transfer (inlet to the cylinder) and exhaust ports are on directly facing sides of the cylinder wall. To prevent the incoming mixture passing straight across from one port to the other, the piston has a raised rib on its crown. This is intended to deflect the incoming mixture upwards, around the combustion chamber.[1] Much effort, and many different designs of piston crown, went into developing improved scavenging. The crowns developed from a simple rib to a large asymmetric bulge, usually with a steep face on the inlet side and a gentle curve on the exhaust. Despite this, cross scavenging was never as effective as hoped. Most engines today use Schnuerle porting instead. This places a pair of transfer ports in the sides of the cylinder and encourages gas flow to rotate around a vertical axis, rather than a horizontal axis.

Fig1. 6 Deflector head

Steam engine piston: Steam engines are usually double-acting (i.e. steam pressure acts alternately on each side of the piston) and the admission and release of steam is controlled by slide valves,piston valves or poppet valves. Consequently, steam engine pistons are nearly always comparatively thin discs: their diameter is several times their thickness. (One exception is the trunk engine piston, shaped more like those in a modern internal-combustion engine.) Another factor is that since almost all steam engines use crossheads to translate the force to the drive rod, there are few lateral forces acting to try and "rock" the piston, so a cylinder-shaped piston skirt isn't necessary.

Fig1. 7 Steam engine piston

1.2.2 CONNECTING ROD

In modern automotive internal combustion engines, the connecting rods are most usually made of steel for production engines, but can be made of and aluminium alloys (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for a combination of lightness with strength, at higher cost) for high-performance engines, or of cast iron for applications such as motor scooters. They are not rigidly fixed at either end, so that the angle between the connecting rod and the piston can change as the rod moves up and down and rotates around the crankshaft. Connecting rods, especially in racing engines, may be called "billet" rods, if they are machined out of a solid billet of metal, rather than being cast or forged.

Fig1. 8 Connecting rod

The small end attaches to the piston pin, gudgeon pin or wrist pin, which is currently most often press fit into the connecting rod but can swivel in the piston, a "floating wrist pin" design. The big end connects to the bearing journal on the crank throw, in most engines running on replaceable bearing shells accessible via the connecting rod boltswhich hold the bearing "cap" onto the big end. Typically there is a pinhole bored through the bearing and the big end of the connecting rod so that pressurized lubricatingmotor oil squirts out onto the thrust side of the cylinder wall to lubricate the travel of the pistons and piston rings. Most small two-stroke engines and some single cylinder four-stroke engines avoid the need for a pumped lubrication system by using a rolling-element bearing instead, however this requires the crankshaft to be pressed apart and then back together in order to replace a connecting rod.

1.2.3 CRANK SHAFT

A crankshaft related to crank is a mechanical part able to perform a conversion between reciprocating motion and rotational motion. In a reciprocating engine, ittranslates reciprocating motion of the piston into rotational motion; whereas in a reciprocating compressor, it converts the rotational motion into reciprocating motion. In order to do the conversion between two motions, the crankshaft has "crank throws" or "crankpins", additional bearing surfaces whose axis is offset from that of the crank, to which the "big ends" of the connecting rods from each cylinder attach.

Fig1. 9 Crank shaft

It is typically connected 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 torsional 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.

II. METHODOLOGY

In this report, the Rigid Body and Fatigue analysis are carried out, Hear we are design engine connecting rod assemble to stress concentration by changing material properties from the forged steel into aluminium alloy. There are some strategy to fail the connecting rod by fatigue occur in cyclic engine unbalanced reciprocating masses. Usually, failure occurs while stress value is excised the ultimate stress value and fatigue life of material.

2. 1 OBJECTIVE

In this work, the Finite Element model and solution techniques required for the precise computation of rigid body analysis for finding forces existed in joints, it is related to the rigid body dynamics. Afterwards, the stress value range is finding method of FEA we evaluated to overcome acquired from existing calculated engine expansion pressure is consider force. The objective of this study focuses on the reduction of over design from engine connecting rod, which will be calculated by means of FEA method. Further it will be changing by material properties from the engine specification, after the stress analysis was followed to propose the model.

2.2 BASIC PROCEDURE

The flow chart will give a methodology is followed in the present work. This flow chart is show in a step by step procedure in below:

Step 1: This input parameter for the engine connecting rod assembly as per the dimensions in present model.

Step 2: The Design model is done by the Creo Modelling software respect to input parameter mentioned in dimension.

Step 3: To check the quality of the Model dimension to proceed the analysis.

Step 4: The model is get in to the ansys Geometry modular, by importing external geometry command.

Step 5: In ansys Workbench solver perform the Rigid Body Analysis, its mean RBD for finding the force existed between joints only.

Step6: Joints forces are calculated has per the 2cycles consider 4 strokes.

Step 7: Then perform Static Structural analysis is finding the stress for engine connecting rod assembly to find factor of safety.

Step 8: Stress value compared to ultimate stress value in static structural result, which give a factor of safety range.

Step 9: As per the resultsperform Static Structural analysis is finding the stress for engine connecting rod assembly for a new optimized model.

Step 9: Same can repeat , Stress value compared to ultimate stress value in static structural result, which give a factor of safety range.

Step 10: The comparison of all the results it’s give an idea aboutoptimized model is proposed.

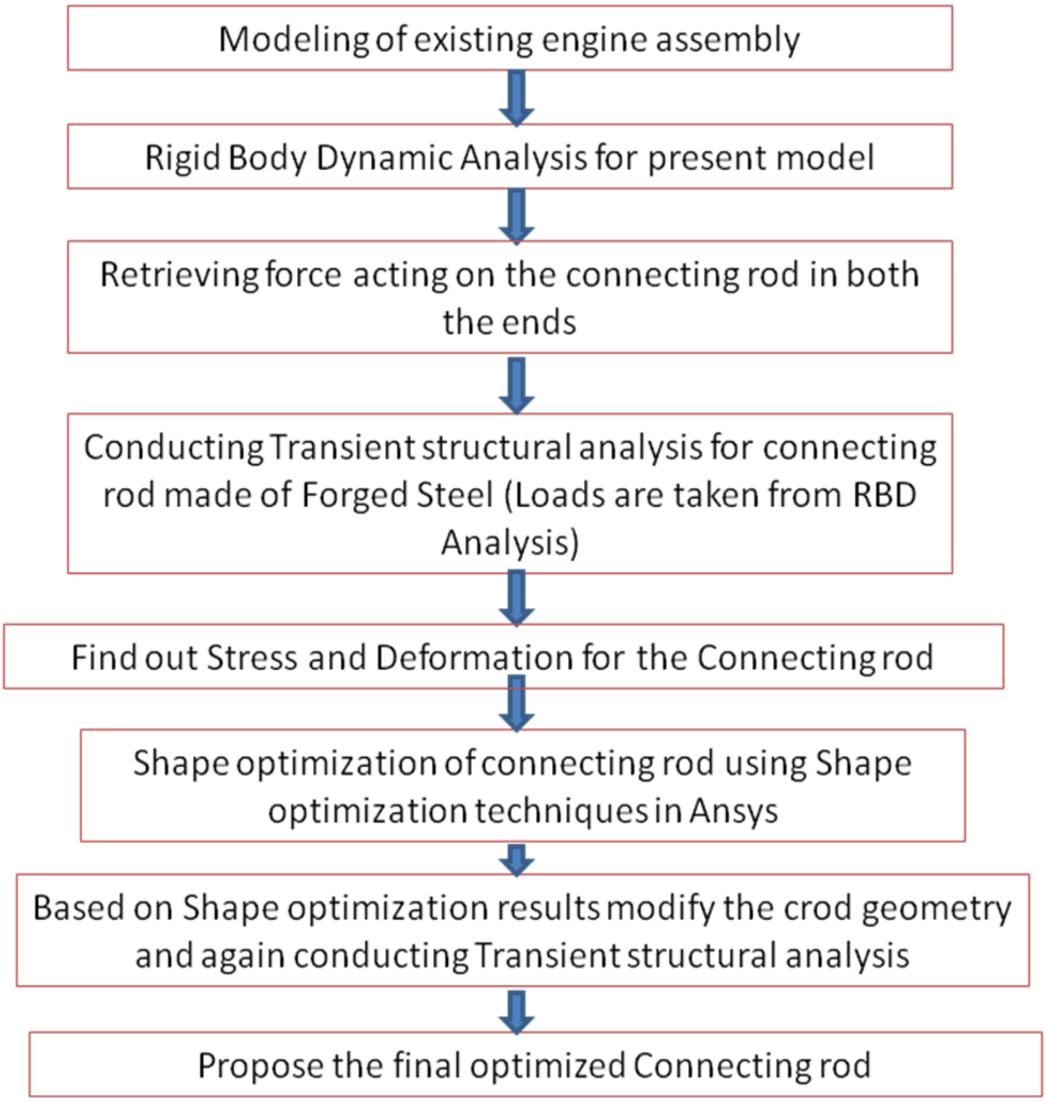
2.3 ASSUMPTION

The following assumption are made to develop the engine connecting rod

- Engine expansion stroke, it created a pressure force on engine piston that will convert into force.
- Force is applied on the piston head, in RBD, bodies are consider as rigid body to find the joints exiting force.
- Center of gravity is act at all body in center position.
- In results from RBD joints forces are applied in static structural analysis constrain are given to results
- The boundary conditions are made to procedure

2.4 LOAD CASES

- Expansion stoke pressure is consider has force
- RBD results are applied.
- Center of gravity is coincided



3.1 INTRODUCTION RIGID BODY ANALYSIS

Multibody simulation consists of analyzing the dynamic behavior of a system of interconnected bodies comprised of flexible and/or rigid components. The bodies may be constrained with respect to each other via a kinematically admissible set of constraints modeled as joints. These systems can represent an automobile, a space structure with antenna deployment capabilities, an aircraft as an assemblage of rigid and flexible parts, a robot with manipulator arms, and so on. In all such cases, the components may undergo large rotation, large displacement, and finite strain effects.

The following additional topics offer more information to help you understand multibody simulation and how the ANSYS program supports it:

- Benefits of the Finite Element Method for Modeling Multibody Systems
- Overview of the ANSYS Multibody Analysis Process
- The ANSYS-ADAMS Interface
- Learning More About Multibody Dynamics

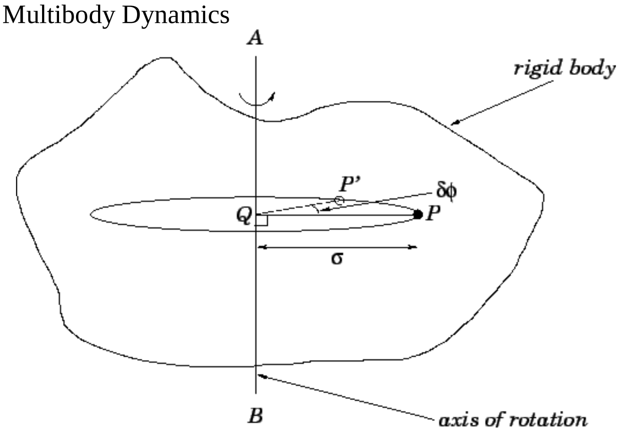


FIg3. 1 Rigid Body Dynamics

3.2 BENEFITS OF MULTI BODY ANALYSIS

Multibody systems have conventionally been modeled as rigid body systems with superimposed elastic effects of one or more components. These methods have been well documented in multibody dynamics literature. A major limitation of these methods is that nonlinear large-deformation, finite strain effects, or nonlinear material cannot be incorporated completely into model.

The finite element (FE) method used in ANSYS offers an attractive approach to modeling a multibody system. While the ANSYS multibody analysis method may require more computational resources and modeling time compared to standard analyses, it has the following advantages:

- The finite element mesh automatically represents the geometry while the large deformation/rotation effects are built into the finite element formulation.
- Inertial effects are greatly simplified by the consistent mass formulation or even point mass representations.
- Interconnection of parts via joints is greatly simplified by considering the finite motions at the two nodes forming the joint element.
- The parameterization of the finite rotation has been well documented in the literature and can be easily incorporated into the joint element formulations thereby enabling complete simulation of a Multibody system.

ANSYS has an extensive library of elements available for modeling the flexible, rigid, and joint components. You can model the material behavior of the flexible components using one of several material models. ANSYS also provides modal and transient dynamics capabilities to analyze the spatial and temporal effects in a multibody simulation. Extensive postprocessing capabilities are also available to interpret the analysis results. You can perform multibody simulation on a wide variety of mechanical systems. Typical applications include automobiles and automobile components, aircraft assemblages, spacecraft applications, and robotics.

3.3 MODELING IN A MULTI BODY SIMULATION

. The finite element modeling of a Multibody mechanism depends on the degree of complexity that you require. For example, it is often possible to create a quick, initial approximation of the flexible and rigid parts of a mechanism using standard beam elements and rigid beam/link elements. Alternatively, you can perform detailed modeling of the flexible part using 3-D solid elements (or shell or solid-shell elements), and the rigid part using the ANSYS program's extensive contact capabilities.

The following topics related to multibody analysis modeling are available:

- Modeling Flexible Bodies in a Multibody Analysis
- Modeling Rigid Bodies in a Multibody Analysis
- Connecting Multibody Components with Joint Elements

3.3.1 MODELING FLEXIBLE BODIES IN A MULTIBODY ANALYSIS

Consider a slider-crank mechanism as shown in the following figure. The crank is considered to be rigid and the connecting link is assumed to be flexible. The link connects the crank to the sliding block (or piston). The simplified finite element model of the slider-crank mechanism is also shown.

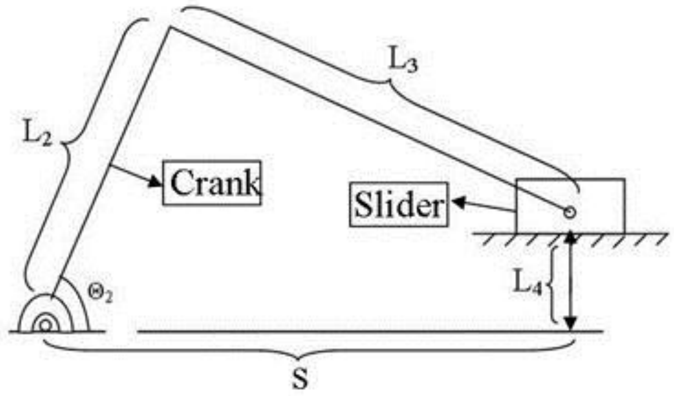


FIg3. 2 Crank Slider mechanism

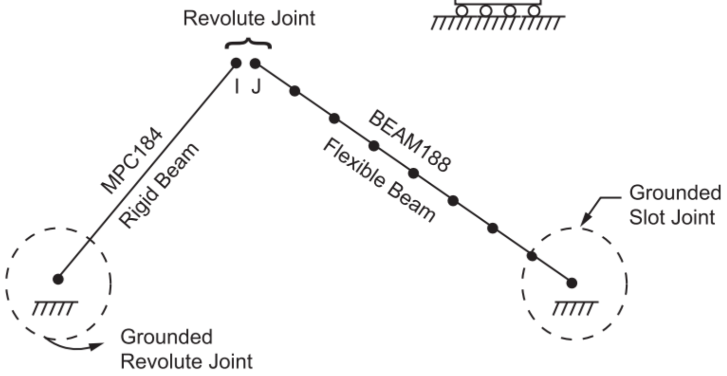


FIg3. 3 Joints of the process

The slider-crank mechanism has these characteristics:

- The rigid crank is modeled with an MPC184 Rigid Beam element.
- The rigid crank is connected to ground with a “grounded” MPC184 Revolute Joint element.
- The connecting link is flexible and modeled with BEAM188 elements.
- The rigid crank and the connecting link are connected to each other by a MPC184 Revolute Joint element.
- The connecting link moves within a “grounded” MPC184 Slot Joint that approximates a slider block.

As a quick first attempt, you can model the flexible mechanism with some simple approximations to the flexible and rigid parts. You can also model the connecting link in detail to study the deformation, stresses, etc.

3.4 MODELING RIGID BODIES IN A MULTIBODY ANALYSIS

Rigid bodies are widely used for numerical simulation of Multibody dynamic applications. A rigid body can be connected to other rigid bodies via joint elements. It can also be connected to flexible bodies to model mixed rigid-flexible body dynamics.

In a finite-element model, certain relatively stiff parts can be represented by rigid bodies when stress distributions and wave propagation in such parts are not critical. An advantage of using rigid bodies rather than deformable finite elements is computational efficiency. Elements that belong to the rigid bodies have no associated internal forces or stiffness. The motion of the rigid body is determined by a maximum of six degrees of freedom (DOFs) at the pilot node.

For transient dynamic analyses, stiff bodies can excite high-frequency modes, resulting in a small time increment in order to obtain a stable solution. Rigid bodies do not, however, excite any frequency modes; therefore, using rigid bodies to represent stiff regions may allow a relatively large time increment.

The following topics about rigid body modeling are available:

- Defining a Rigid Body
- Rigid Body Degrees of Freedom
- Rigid Body Boundary Conditions
- Representing Parts of a Complex Model with Rigid Bodies
- Connecting Joint Elements to Rigid Bodies
- Modeling Contact with Rigid Bodies

3.4.1 DEFINING A RIGID BODY

A rigid body in ANSYS consists of a set of target nodes called rigid body nodes and a single pilot node. The associated target elements use the same real constant ID. The motion of the rigid body is governed by the degrees of freedom (DOFs) at the pilot node, allowing accurate representation of the geometry, mass, and rotary inertia of the rigid body.

3.4.2 RIGID BODY DEGREES OF FREEDOM

The pilot node has both translational and rotational degrees of freedom (DOFs). The active DOFs at the pilot node depend on the defined type of target elements. Use TARGE169 for a 2-D rigid body which contains UX, UY and ROTZ DOFs. Use TARGE170 for 3-D rigid body which contains UX, UY, UZ and ROTX, ROTY, ROTZ DOFs.

The DOFs of rigid body nodes are based on the DOFs of the connected elements and applied boundary conditions (BCs). Rigid body nodes that connect to solid elements involve only the translational degrees of freedom. Rigid body nodes that connect to shell, beam, follower, and joint elements also involve the rotational DOFs.

For standalone rigid body nodes not connected to any other elements, the associated DOFs are subject to applied boundary conditions, as shown:

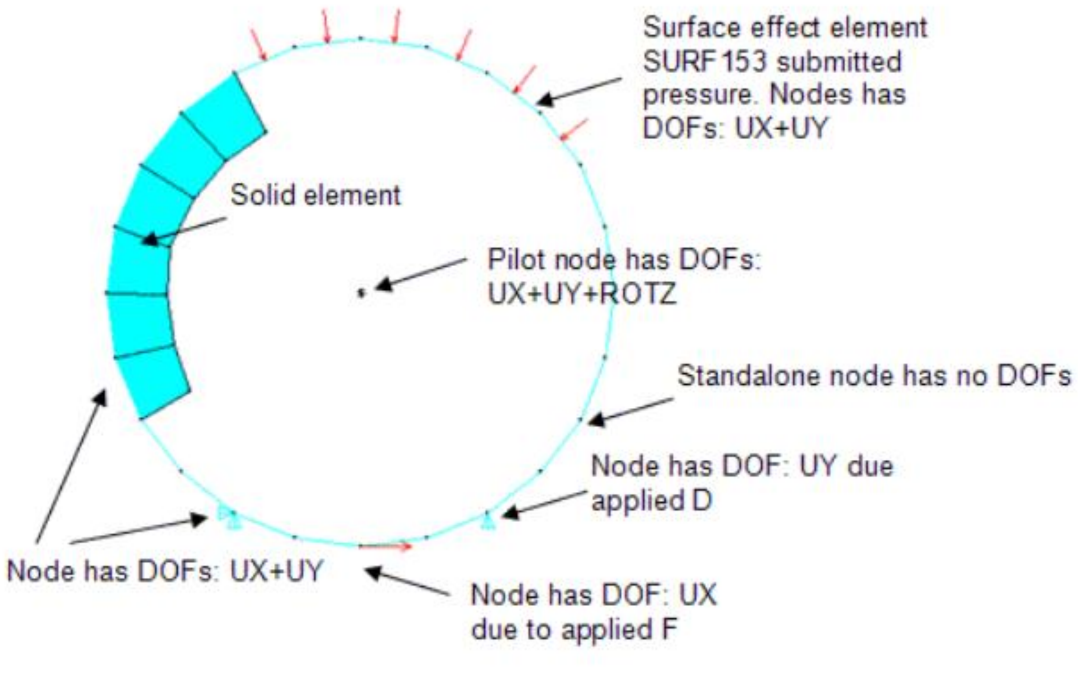


FIg3. 4 DOF of 2DTable3.1

3.4.3 RIGID BODY BOUNDARY CONDITIONS

Constrained boundary conditions (BCs) for the rigid body are usually applied on the rigid body pilot node. Reaction forces can be obtained for DOFs at the constrained nodes. A combination of rigid body constraints and constrained boundary conditions applied to several rigid body nodes other than the pilot node can lead to overconstrained models. In such cases, ANSYS issues overconstraints warnings and attempts to remove the redundant constraints if possible. If the specified BCs are not consistent with the rigid body constraint, the model becomes inconsistently overconstrained. You must verify the overconstrained model and prevent conflicting overconstraints.

3.4.4 REPRESENTING PARTS OF A COMPLEX MODEL WITH RIGID BODIES

Using rigid bodies to represent certain portions of a complex model is more efficient than using flexible finite elements. In the early stage of finite element model development, you can treat certain stiff parts or discretized elements that are far away from the region of interest as the rigid bodies. In a later stage, you can remove the rigid body definition and add the flexible discretized elements back for a detailed and accurate finite element analysis.

By selecting or deselecting target elements or the flexible finite elements, you can easily switch back and forth between rigid body and flexible body definition.

The following table shows the general steps involved when defining a rigid body as compared to defining a flexible body:

Rigid Body Definition Process	Flexible Body Definition Process
1. Select the associated finite elements with defined mass density 2. Perform a partial element solution to obtain mass properties 3. Add a point mass element to the center of body. rigid body 4. Add a target element whose node (pilot node) shares the point mass node. 5. Generate target elements on the exterior surface of the pre-mesh body. 6. Unselect the associated finite elements. 7. Connect joint elements to target nodes.	1. Unselect the relevant point mass and target elements. 2. Reselect the associated finite elements. 3. Define the material properties for the flexible body. 4. Select the nodes on the exterior surface of the body that you want to connect to this pilot node. 5. Create target elements on this surface.

Table3. 1 Definition Process

3.4.5 CONNECTING JOINT ELEMENTS TO RIGID BODIES

Joint elements can be connected to any rigid body nodes and the pilot node. You can define connections between rigid bodies, or between a rigid body and a flexible body.

3.4.6 MODELING CONTACT WITH RIGID BODIES

Contact between two rigid bodies is modeled by specifying a contact surface on one rigid body and a target surface on another rigid body. Use either the augmented Lagrange algorithm or penalty algorithm (KEYOPT(2) on the contact element) for modeling contact between rigid bodies to avoid redundant overconstraint between rigid body constraints and contact constraints.

You cannot use the multipoint constraint (MPC) algorithm (KEYOPT(2)) and bonded or no-separation contact behavior (KEYOPT(12)) to connect two rigid surfaces; doing so would cause the model to be overconstrained, resulting in an abnormal termination of the analysis. You can simply replace the bonded contact pair by adding an additional rigid body which connects two pilot nodes.

ANSYS allows two rigid bodies that are connected or overlap each other through rigid body nodes or the pilot node. To prevent overconstraints, ANSYS merges two rigid bodies into one rigid body internally and treats the second pilot node as a regular rigid body node.

MPC bonded contact between a flexible body and a rigid body is possible. The contact surface in an MPC bonded contact pair, however, should always belong to the flexible body; otherwise, the MPC bonded constraints and rigid body constraints are redundant.

3.5 CONNECTING MULTIBODY COMPONENTS WITH JOINT ELEMENTS

The relative motion between the two nodes is characterized by six relative degrees of freedom.

Revolute Joint Constrained degrees of freedom: UX, UY, UZ, ROTX, ROTY.

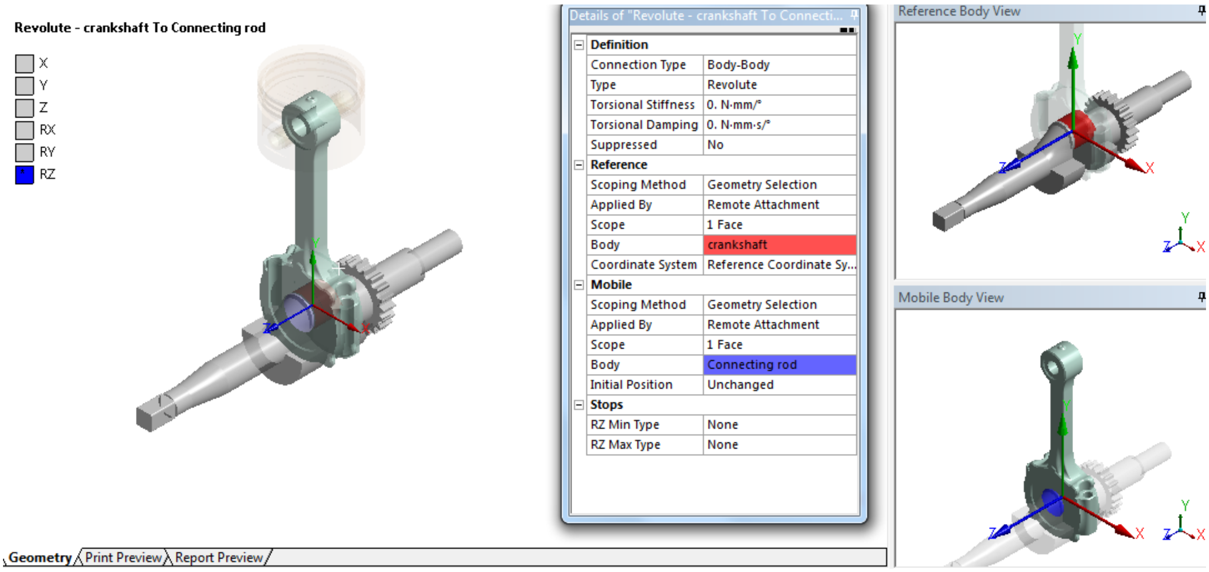


FIg3. 5 Revolute Joints (connecting rod to Crank shaft)

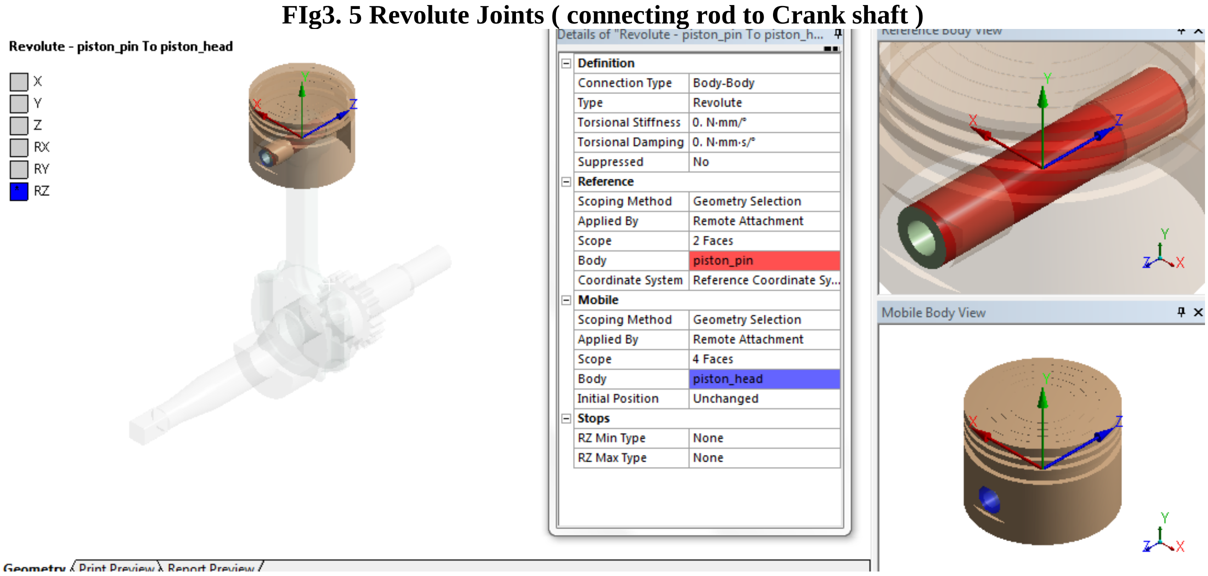


FIg 3. 6a Revolute Joints (Piston pin to Piston head)

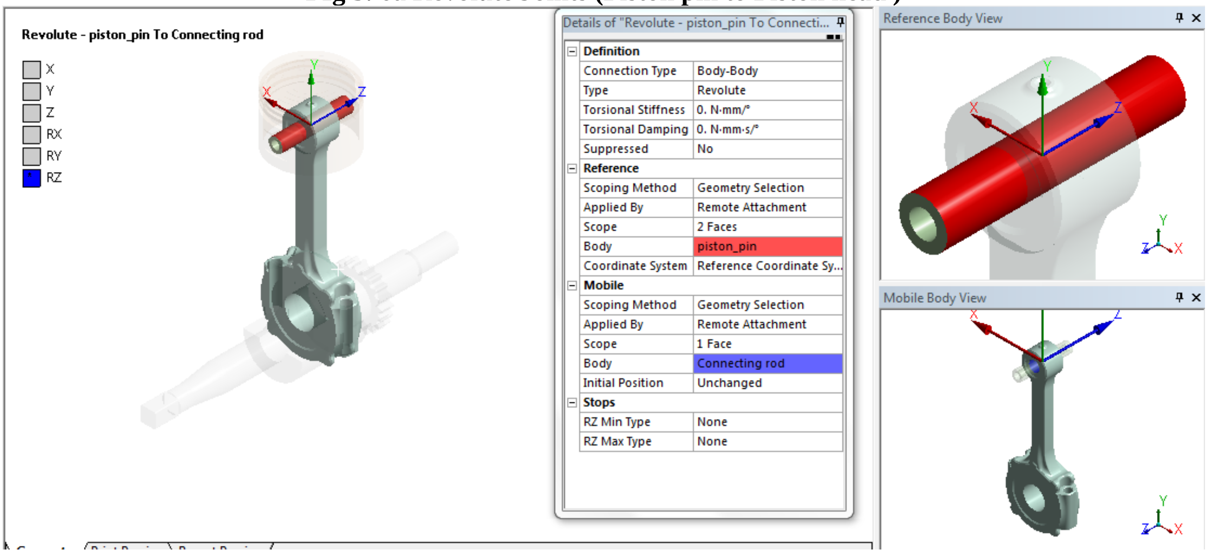


FIg 3. 7b Revolute Joints (Piston pin to Connecting Rod)

Universal Joint Constrained degrees of freedom: UX, UY, UZ, ROTY

FIg3. 8 Universal Joint

Slot Joint Constrained degrees of freedom: UY, UZ

FIg3. 9 Slot Joint

Translational Joint Constrained degrees of freedom: UY, UZ, ROTX, ROTY, ROTZ

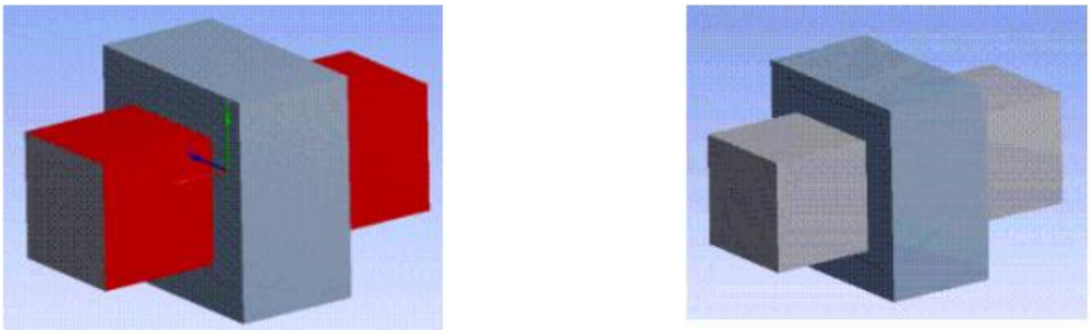


FIg3. 10 Translational Joint

Cylindrical Joint Constrained degrees of freedom: UX, UY, ROTX, ROTY

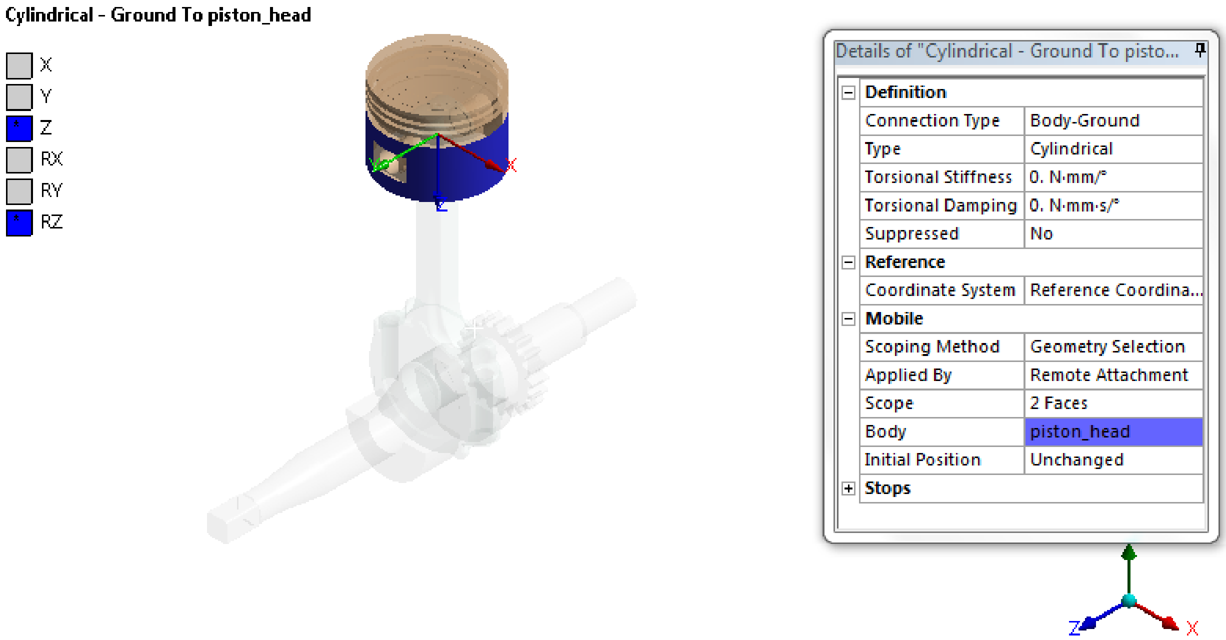


FIg3. 11 Cylindrical Joint

Spherical Joint Constrained degrees of freedom: UX, UY, UZ

FIg3. 12 Spherical Joint

Planar Joint Constrained degrees of freedom: UZ, ROTX, ROTY

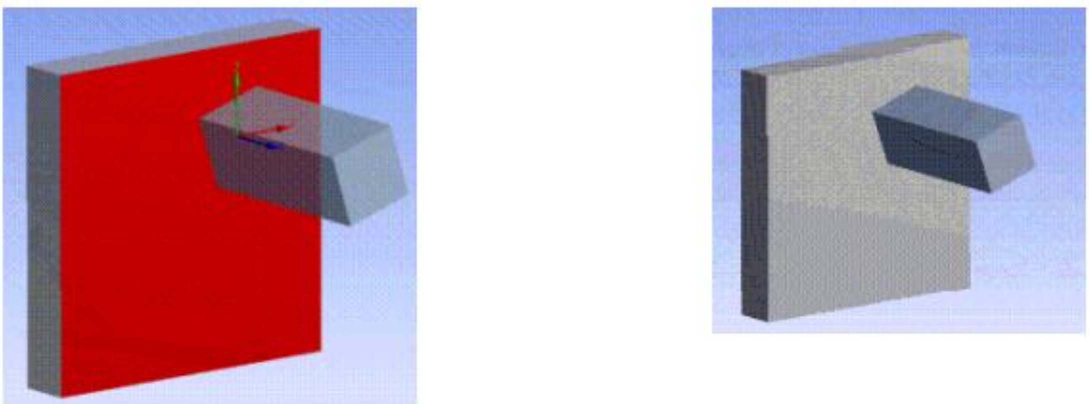


FIg3. 13 Planar Joint

3.3 OVERVIEW OF THE ANSYS MULTIBODY ANALYSIS PROCESS

A multibody simulation involves the same general steps necessary for any ANSYS nonlinear analysis. The following table describes the multibody analysis process:

3.3.1 BUILD THE MODEL

A flexible mechanism usually consists of flexible and/or rigid parts connected together with joint elements. You can model the flexible parts with any of the 3-D solid, shell, or beam elements available in the ANSYS element library. (For more information, see Building the Model in the Basic Analysis Guide,

Rigid bodies are modeled using MPC184 Rigid Link or Rigid Beam elements, or by using the extensive contact capabilities available in ANSYS.

The flexible and/or rigid parts are connected using MPC184 joint elements. For example, two parts may be simply connected such that the displacements at the joining position are identical. In other cases, the connection between two parts may involve a more sophisticated joint such as the planar joint or universal joint. In modeling these joints, suitable kinematic constraints are imposed on the relative motion (displacement and rotation) between the two nodes forming the joint. An overview of the types of joint elements used in a Multibody analysis is available in Connecting Multibody Components with Joint Elements.

3.3.2 DEFINE ELEMENT TYPES

To properly perform a flexible Multibody simulation, which involves flexible and rigid components joined together with some form of kinematic constraints, use appropriate structural, joint, and contact element types.

3.3.3 DEFINE MATERIALS

Defining the material properties for multibody components is no different than defining them in any other ANSYS analysis. Define linear and nonlinear material properties via the MP or the TB command. For more information, see Defining Material Properties in the Basic Analysis Guide.

The MPC184 joint elements also allow you to define material properties so that you can control their behavior along the "free" or "uncon-strained" DOFs. For example, you can issue a TB,JOIN command to introduce a torsional spring behavior for a revolute joint to model the resistance to the rotation along the revolute axis.

3.3.4 MESH THE MODEL

Use the ANSYS meshing commands to mesh multibody components. For more information, see the Modeling and Meshing Guide.

Issue the LMESH command to mesh rigid bodies defined by MPC184 Rigid Beam or Rigid Link elements. Use the Contact Wizard to mesh rigid bodies defined via the contact capabilities in ANSYS.

Two nodes define joint elements and no special meshing commands are necessary to define them.

3.3.5 SOLVE THE MODEL

The solution phase of a Multibody analysis adheres to standard ANSYS conventions.

3.3.6 REVIEW THE RESULTS

You can use POST1 (the general postprocessor) and POST26 (the time history postprocessor) to review results.

4.1 INTRODUCTION FOR FATIGUE

Sress is the weakening of a material caused by repeatedly loads. It is the progressive and localised structural damage that occurs when a material is subjected to cyclic loading. The nominal maximum stress values that cause such damage may be much less than the strength of the material typically quoted as the ultimate tensile stress limit, or the yield stress limit.

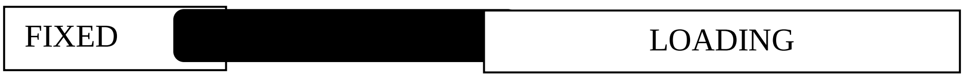


Fig4. 1 Fatigue

Fatigue occurs when a material is subjected to repeated applying loading and unloading. If the loads are above a certain threshold, microscopic cracks will begin to form at the stress concentrators such as the surface, persistent slip bands (PSBs), and grain interfaces.[1]Eventually a crack will reach a critical size, the crack will propagate suddenly, and the structure will fracture. The shape of the structure will significantly affect the fatigue life; square holes or sharp corners will lead to elevated local stresses where fatigue cracks can initiate. Round holes and smooth transitions or fillets will therefore increase the fatigue strength of the structure.

4.2 FATIGUE LIFE

ASTM defines fatigue life N_f, as the number of stress cycles of a specified character that a specimen sustains before failure of a specified nature occurs.For some materials, notably steel and titanium, there is a theoretical value for stress amplitude below which the material will not fail for any number of cycles, called a fatigue limit, endurance limit, or fatigue strength.

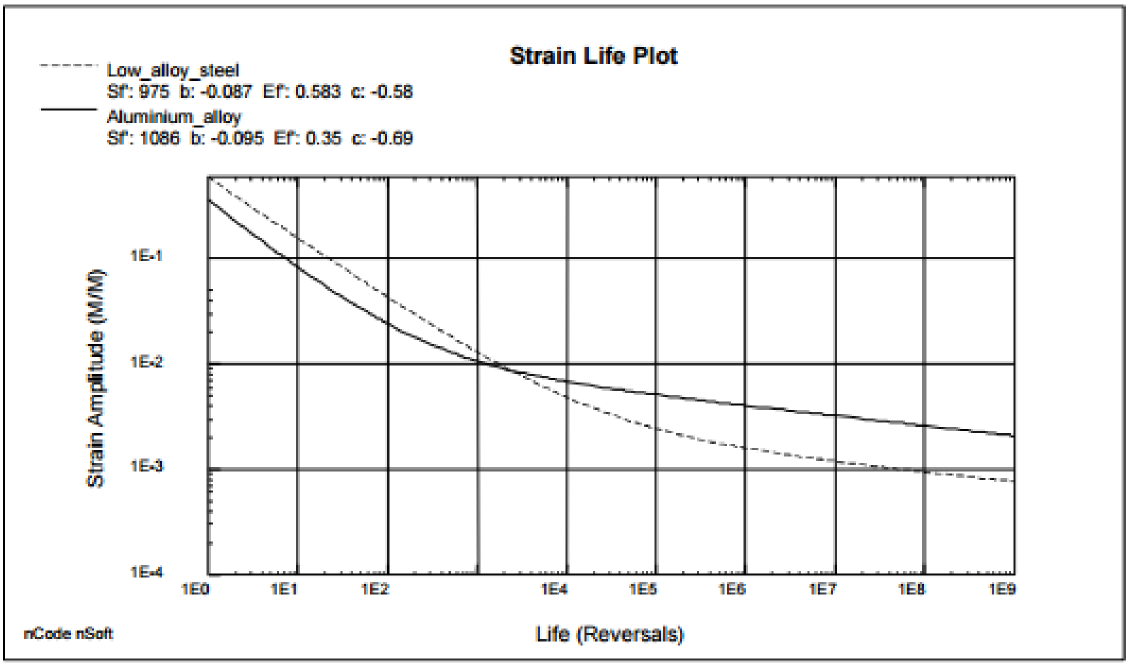


Fig4. 2 Strain life

Engineers have used any of three methods to determine the fatigue life of a material: the stress-life method, the strain-life method, and the linear-elastic fracture mechanics method. One method to predict fatigue life of materials is the Uniform Material Law (UML). UML was developed for fatigue life prediction of aluminium and titanium alloys by the end of 20th century and extended to high-strength steelsand cast iron.

4.3 CHARACTERISTICS OF STESS

- In metal alloys, when there are no macroscopic or microscopic discontinuities, the process starts with dislocation movements, which eventually form persistent slip bands that become the nucleus of short cracks.
- Macroscopic and microscopic discontinuities as well as component design features which cause stress concentrations (holes, keyways, sharp changes of direction etc.) are common locations at which the fatigue process begins.
- Fatigue is a process that has a degree of randomness (stochastic), often showing considerable scatter even in well controlled environments.
- Fatigue is usually associated with tensile stresses but fatigue cracks have been reported due to compressive loads.
- The greater the applied stress range, the shorter the life.
- Fatigue life scatter tends to increase for longer fatigue lives.
- Damage is cumulative. Materials do not recover when rested.
- Fatigue life is influenced by a variety of factors, such as temperature, surface finish, metallurgical microstructure, presence of oxidizing or inert chemicals, residual stresses,scuffing contact (fretting), etc.
- Some materials (e.g., some steel and titanium alloys) exhibit a theoretical fatigue limitbelow which continued loading does not lead to fatigue failure.
- High cycle fatigue strength (about 104 to 108 cycles) can be described by stress-based parameters. A load-controlled servo-hydraulic test rig is commonly used in these tests,with frequencies of around 20–50 Hz. Other sorts of machines—like resonant magnetic machines—can also be used, to achieve frequencies up to 250 Hz.
- Low cycle fatigue (loading that typically causes failure in less than 104 cycles) is associated with localized plastic behavior in metals; thus, a strain-based parameter should be used for fatigue life prediction in metals. Testing is conducted with constant strain amplitudes typically at 0.01–5 Hz.

4.4 PROCEDURES TO PERFORM FEA BY ANSYS

In ANSYS, there are five typical steps for performing a finite element analysis.
The first step is together the data of the problem. The second step is to build a finite element modelfortheapplicationproblem.
This step consists of activities such as defining units, selecting types of elements, defining material properties, and creating the finite element model. As forde fining a system of units, it should be note that the ANSYS program does not assume a system of units. Thus, the users are responsible to maintain the consistency of system of units for all the input data in the ANSYS program As for selecting element types the decision is based on the characteristics ofelementtypetobestmodel/fitthatapplicationproblemgeometricallyandphysically. Depending on the element types material properties may be line aro non-linear isotropic ,orthotropic, oran isotropic; and constant temperature-independent ortemperature-dependent.
There are two methods to create a finite element model in ANSYS: automatic meshing (also called the Solid Modeling in ANSYS terminology) and manual meshing (also called the direct generation in ANSYS terminology). In automatic meshing, user sare required to have a solid model available prior to the creation of a finite element model. When such a solid model become savailable, the users then can instruct ANSYS to automatically develop a finite element model (nodes and elements). The purpose fusing automatic meshing is tore lieve the user of the time- consuming task of building a complicated finite element model. However, this method requires significant amounts of CPU time and sometimes fails to maintain the connectivity of nodes and elements.

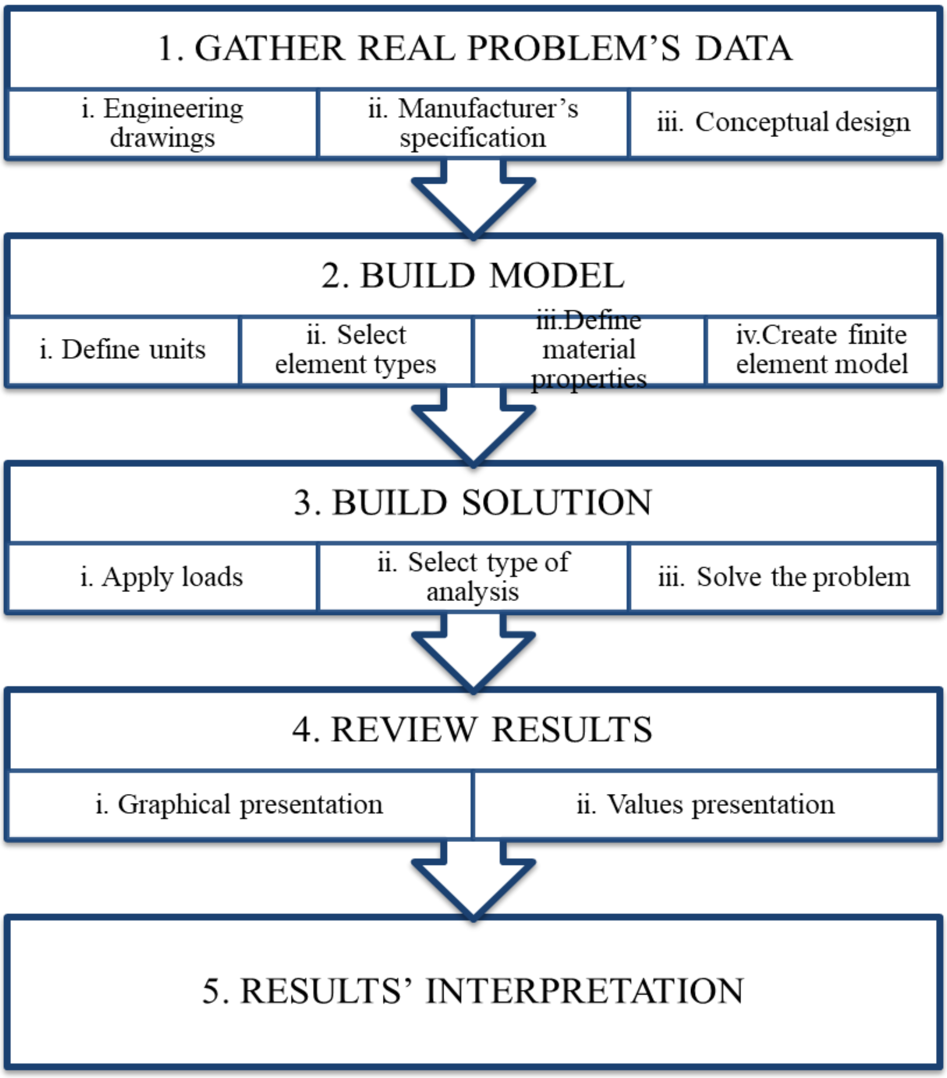


Fig 4.3 General Procedure to perform FEA by ANSYS

4.4 MESHING MODEL

MESHING SIZE

MIN SIZE 1.00mm

MAX SIZE 2.00mm

MESHING ELEMENT

TETRAHEDRONS

Statistics	
Nodes	11813
Elements	55960
Mesh Metric	None

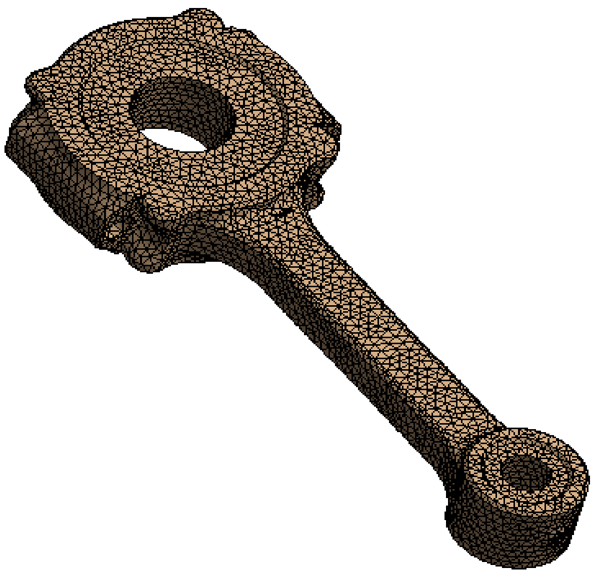


Fig 4.4 Meshing

4.5 BOUNDARY CONDITION

A Boundary condition, that is required to be satisfied at all or part of the boundary of a region in which a set of differential conditions is to be solved.

Fig4.5 Boundary conditions

4.6 MATERIAL

Properties of Outline Row 3: Froged Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			

V. RESULTS AND DISCUSSION

5.1 RIGID BODY ANALYSIS Boundary condition

Force and Angular displacement applied to the connecting rod

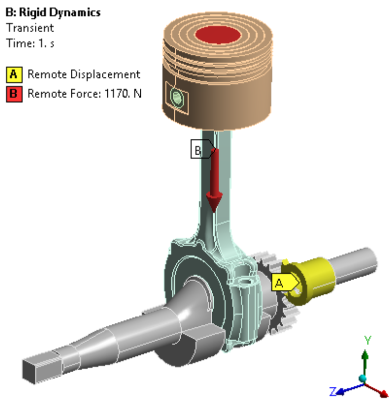
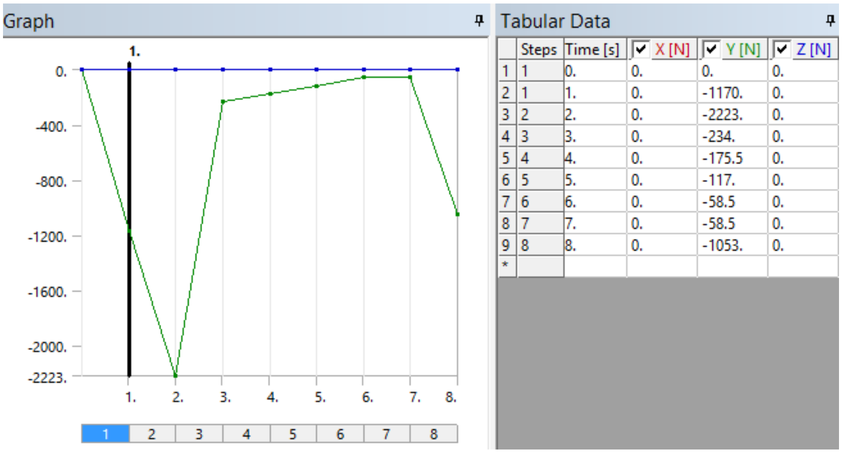


Fig 5. 1 Forces excited in Crank End



	Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0.	0.	0.	0.
2	1	1.	0.	-1170.	0.
3	2	2.	0.	-2223.	0.
4	3	3.	0.	-234.	0.
5	4	4.	0.	-175.5	0.
6	5	5.	0.	-117.	0.
7	6	6.	0.	-58.5	0.
8	7	7.	0.	-58.5	0.
9	8	8.	0.	-1053.	0.
*					

Fig 5.2 Gas force

THE FORCE EXITED ON THE CONNECTING ROD BIG END

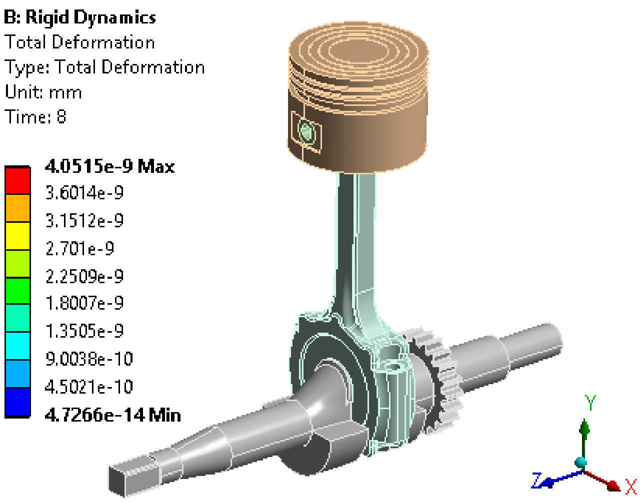


Fig 5.3 Forces excited on BIG End

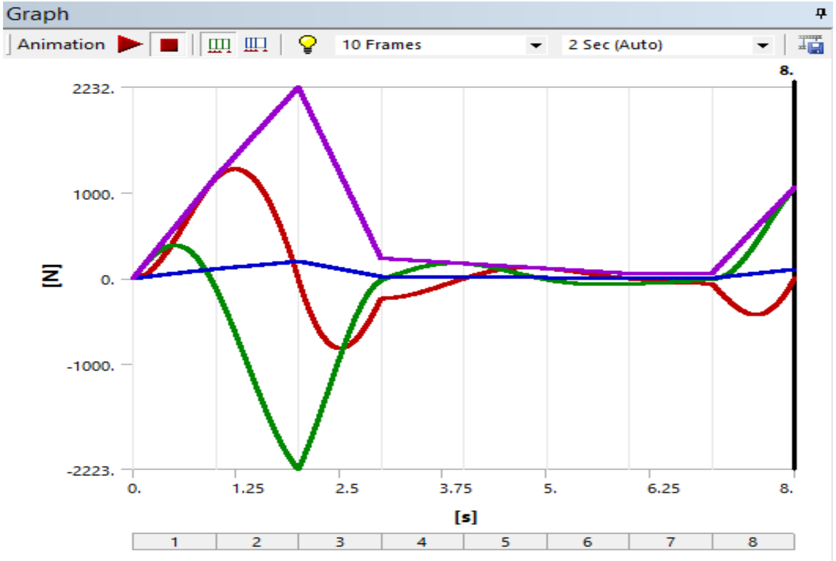


Figure 5.4 Forces on big end

TOTAL DEFORMATIOM

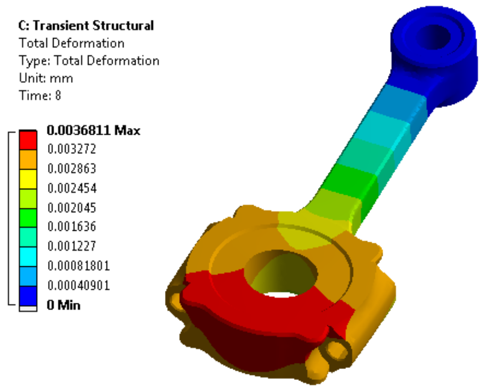


Fig 5.5 Total Deformation on existing model

EQUIVALENT STRESS

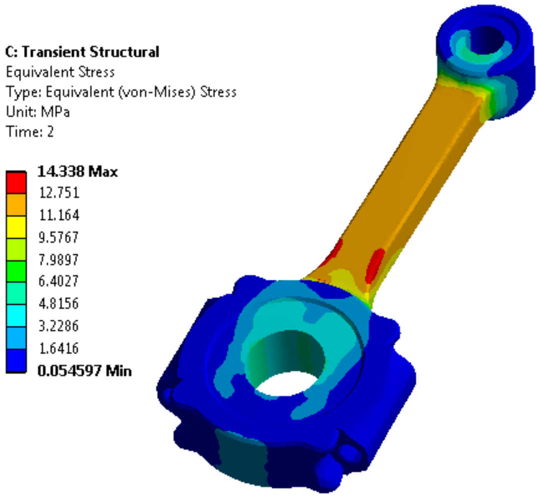


Fig 5.6 Equivalent stress on existing model

5.3 SHAPE OPTIMIZATION ANALYSIS

SHAPE OPTIMIZATION RESULTS

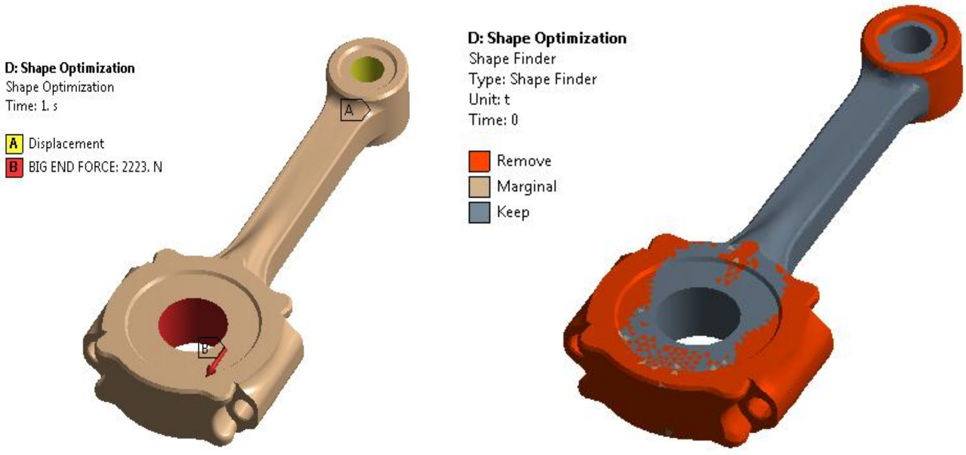
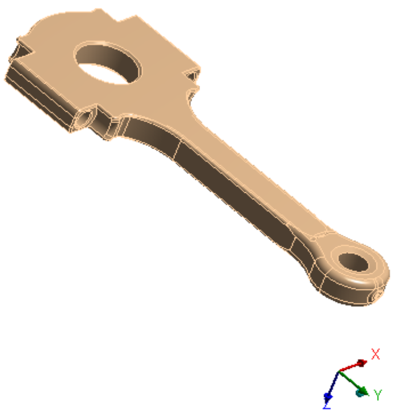


Fig 5.7 shape optimization

MODIFIED CONNECTING ROD

Properties	
Volume	59965 mm^3
Mass	0.47072 kg
Centroid X	6.5157 mm
Centroid Y	-49.77 mm
Centroid Z	4.9241 mm



Details of "Shape Finder"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Target Reduction	50. %
Suppressed	No
Results	
Original Mass	0.74131 kg
Marginal Mass	3.8086e-003 kg
Optimized Mass	0.37995 kg

TRANSIENT STUCTURAL ANALYSIS FOR MODIFIED MODEL

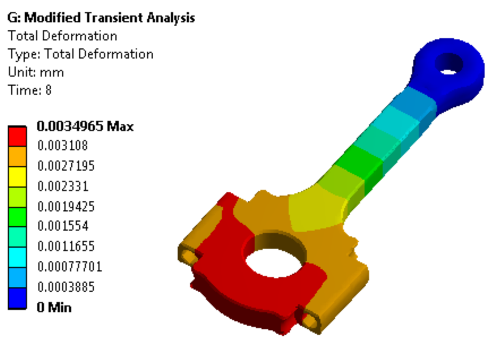


Fig 5.8 Total Deformation for modified

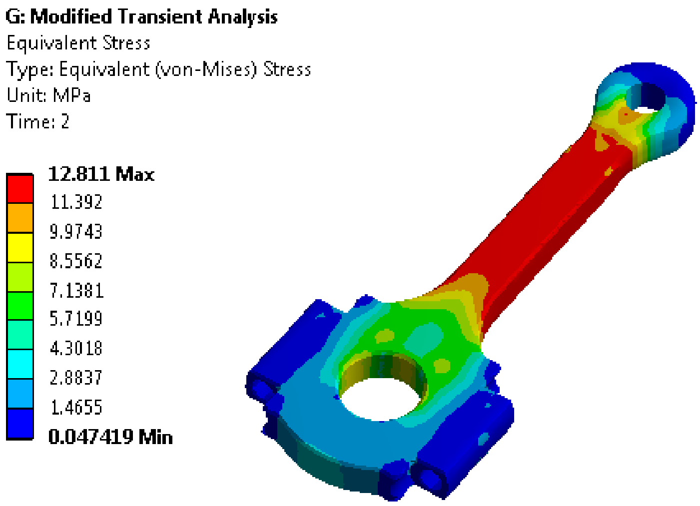


Fig 5.9 Equivalent stress for modified

VI. CONLUSION

The main aim of the project is to reduce the weight and cost ratio of the connecting rod for petrol engines by calculating the forces through Rigid Body Dynamics. The optimized results are compared in below table,

S.No	Model	Total Deformation (mm)	Equivalent Stress (MPa)	Weight (Kg)
1.	Existing	0.0036	14.33	0.741
2.	Modified	0.0034	12.81	0.470

Table 9.1 Comparing the Results

From the above discussion and points tabulated, the connecting rod was optimized by around 40% in weight using basic principles of engineering. A weight savings of around 0.27 kg material is achieved in one connecting rod. The optimized model also met the stress criteria which means the stresses observed in both the baseline and optimized model were well within the safety limit. Hence the safety of the optimized model is ensured.

References

[1]. J. Smith and P. Brown, "Design and optimization of connecting rods for high-performance engines," J. Mech. Eng., vol. 45, no. 3, pp. 121–130, Mar. 2021.
[2]. S. Kumar, R. Singh, and T. Patel, "Stress analysis of connecting rods using finite element method," in Proc. Int. Conf. Mech. Eng. Res., Aug. 2019, pp. 56–62.
[3]. A. Johnson, "Dynamic simulation of IC engine components using rigid body mechanics," J. Eng. Simul., vol. 12, no. 4, pp. 89–97, Apr. 2020.
[4]. R. Kumar, "Optimization of engine connecting rod using ANSYS," Int. J. Mech. Eng. Tech., vol. 8, no. 6, pp. 230–238, Jun. 2018.
[5]. C. Lee and M. Kim, "Finite element analysis of engine components for structural integrity assessment," Mech. Des. Lett., vol. 30, no. 2, pp. 155–162, Feb. 2017.
[6]. B. Wang and Z. Li, "Lightweight design of connecting rods using shape optimization," Proc. Inst. Mech. Eng. C, vol. 233, no. 3, pp. 512–520, Mar. 2019.
[7]. T. Nguyen and L. Zhang, "Analysis of inertia forces in high-performance engine components," Int. J. Automot. Eng., vol. 15, no. 5, pp. 43–50, May 2021.
[8]. D. Patel and A. Mehta, "Application of PTC Creo for modeling IC engine components," in Proc. Nat. Conf. Innov. Mech. Eng., Dec. 2018, pp. 102–108.
[9]. K. Singh, "Performance analysis of air-cooled engines with optimized connecting rods," J. Therm. Eng., vol. 5, no. 2, pp. 111–120, Feb. 2020.
[10]. P. Roberts, "Evaluation of compressive and bending loads in connecting rods," Eng. Struct. J., vol. 22, no. 7, pp. 97–104, Jul. 2017.
[11]. L. Chang and J. Lee, "Stress and fatigue analysis in IC engine components under dynamic loading," Int. J. Adv. Eng. Tech., vol. 9, no. 4, pp. 210–218, Apr. 2020.
[12]. M. Thomas, "Shape optimization techniques for improving stress distribution in engine components," Mater. Perform. Eng., vol. 18, no. 5, pp. 431–440, May 2019.