VIBRATION AND STATIC ANALYSIS OF COMPOSITE 2-WHEELER CONNECTING ROD

Mr. S. S. Said

ME Student Siddhant College of Engineering, Pune.

Dr. L. V. Kamble

Principal Siddhant College of Engineering, Pune.

Dr. P. A. Makasare

HOD Mechanical Siddhant College of Engineering, Pune.

Prof. R. R. Kulkarni

Asst professor *Siddhant College of Engineering, Pune*.

Abstract

Composite paraphernalia are now a day vastly used in the engineering field. The general characteristics held by the conflation paraphernalia are set up to be the reason for using it in the automotive operations. The connecting rod is a major link inside of combustion machine. It connects the piston) to the crankshaft and is responsible for transferring power from the piston to the crankshaft. It has to workonhighr.p.m. because of which it has to bear severe stresses which make its design vital for internal combustion machine. In this design, design, analysis of the 2-wheeler connecting rod will be perform. The CATIA V5 R20 software has been used for designing the connecting rod 3D model and also the designed connecting rod model is imported into the ANSYS software in which the design is meshed and anatomized by using the Finite Element Method (FEM) and the result is manipulated. Modal analysis of being and conflation connecting rod will be performed. Results of the analysis are also validated by using UTM.

1. Introduction

The connecting rod is a major link inside a combustion engine. It connects the piston to the crankshaft and is responsible for transferring power from the piston to the crankshaft and sending it to the transmission. The objective of C.R. 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 shank, a small end and a big end. The cross section of shank may be rectangular, circular, tubular, I- Section, + -section or ellipsoidal-Section. It sustains force generated by mass & fuel combustion. The resulting bending stresses appear due to eccentricities, crank shaft, case wall deformation & rotational mass.

Fig. 1.1Connecting rod

There are different types of accoutrements and product styles used in the creation of connecting rods. The most common types of Connecting rods are sword and aluminum. The most common types of manufacturing processes are casting, forging and powdered metallurgy. Connecting rods are extensively used in variety of machines similar as, in- line machines, V- machine, opposed cylinder machines, radial machines and opposed- piston machines. A connecting rod consists of a leg- end, a cutter section, and a coil- end. Leg- end and coil- end perforations at the upper and lower ends are machined to permit accurate fitting of comportments. The function of connecting rod is to transmit the thrust of the piston to the crankshaft. Figure shows the part of connecting rod in the conversion of repaying stir into rotary stir. A four- stroke machine is the most common type. The four strokes are input, contraction, power, and exhaust. Each stroke requires roughly 180 degrees of crankshaft gyration, so the complete cycle would take 720 degrees. Each stroke plays a veritably important part in the combustion process. In the input cycle, while the piston moves over, one of the faucets open. This creates a vacuum, and an air- energy admixture is smelled into the chamber (Figure 1). During the alternate stroke contraction occurs. In contraction both faucets are closed, and the piston moves overhead and therefore creates a pressure on the piston, see Figure 2. The coming stroke is power. During this process the compressed air- energy admixture is burned with a spark, causing a tremendous pressure as the energy becks. The forces wielded by piston transmitted through the connecting rod moves the crankshaft, see Figure 3. Eventually, the exhaust stroke occurs. In this stroke, the exhaust stopcock opens, as the piston moves back overhead, it forces all the air out of the chamber and therefore which completes the cycle of crankshaft gyration Figure 4.

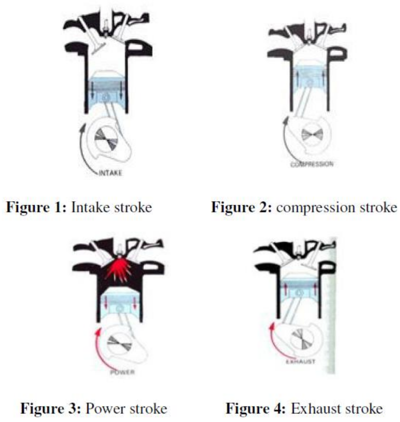


Fig. 1.2. The cycle of crankshaft rotation

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. The optimization of connecting rod had already started as early year 1983 by Webster and his team. However, each day consumers are looking for the best from the best. That's why the optimization is really important especially in automotive industry. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on car performance. Hence, it is effect on the car manufacture credibility. Change in the structural design and also material will be significant increments in weight and performance of the engine.

FEA approach deals with structural analysis along with various parameters which affects its working & define best solution to overcome the barriers associated with it. The structural analysis allows stresses & strains to be calculated in FEA, by using the structural model. The structural analysis performed to create high & low stresses region from the input of the material, loads, boundary condition. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry & boundary condition. It is used to improve optimize design.

1.1. PROBLEM IDENTIFICATION:

Each day consumers are looking for the swish from the swish. That's why the material optimization is really important in sedulity. Material Optimization of the element is to make the lower time to produce the product that is stronger, lighter and lower cost. The design and weight of the connecting rod influence on performance. Hence, it's effect on the manufacture credibility. The tensile and compressive stresses are produced due to pressure, and bending stresses are produced due to centrifugal effect & curiosity. So the connecting rods are designed generally of I- section to give maximum inflexibility with minimum weight. Change in the structural design and also material will be significant supplements in weight and performance.

OBJECTIVES:

- The main aim of the project is to determine the natural frequency of existing and composite connecting rod
- In this Project, the static and modal FEA of the connecting rod has been performed by the use of the ANSYS software.
- Natural frequency of optimize connecting rod are validated by using FFT analyzer and impact hammer test.
- Comparative analysis between FEA & Experimental results.

1.2. Scope

- To study the existing connecting rod for possible designing.
- Learning and use of CATIA and ANSYS software.
- Loads and boundary conditions shall be applied to the model in the pre-processor. The input deck for the designated solver shall be prepared.
- Suitable solver for structural analysis (like ANSYS) would be deployed for finding the solution.
- Recommendation to be made upon evaluating the results.
- Physical experimentation towards validation hypothesis proposed to be carried out.
- Conclusion to be inferred over the work done.

1.3. Expected Outcomes:

- Better structural and vibrational stiffness of connecting rod
- Increased mechanical properties throughout the structure.
- Nearly 10-15% of weight reduction.

Tools and Software Used: (As per requirement)

CAD:
Catia v5

CAE:
Ansys workbench

Manufacturing and Testing:

- Turning , milling, SPM, hand lay – up method etc.
- FFT Analyzer with DAS (Data Acquisition System)

2. Literature Review

2.1. Literature Survey:

D.Gopinatha,Ch.V.Sushmab[1] "Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis", The main objective of research was to explore weight reduction opportunities for the production of forged steel,aluminium and titanium connecting rods. This has entailed performing a detailed load analysis. Therefore, this study has dealtwith two subjects, first, static load stress analysis of the connecting rod for three materials, and second, optimization for weightof forged steel connecting rod.In this research, firstly a proper geometrical model was developed using CATIA. Then the model is imported to theHYPERMESHwhich is a finite element pre-processor that provides a highly interactive and visual environment to analyzeproduct design performance and the Finite Element model was developed. The stresses were found in the existing connecting rodfor the given loading conditions using Finite Element Analysis software ANSYS 11.0.The topology optimization technique isused to achieve the objectives of optimization which is to reduce the weight of the connecting rod.

Mohammed Mohsin Ali Ha , Mohamed Haneef b [2] “Analysis of Fatigue Stresses on Connecting Rod Subjected toConcentrated Loads At The Big EndConnecting rod is modeled using CATIA software and FE analysis is carried out using ANSYS Software .Load distributionplays important role in fatigue life of the structure. Bush failure changes the loading direction and distribution. Present study isconcentrated around the fatigue life due to concentrated load and cosine type load distribution on the bigger end.The connecting rod analysis is carried out to check the fatigue life and alternating stress development dueto service and assembly loads with variation in load distribution.

Amit Kumar, Bhingole P.P. Dinesh Kumar [3]Dynamic Analysis of Bajaj Pulsar 150cc Connecting RodUsing ANSYS 14.0. This paper deals with analysis of Bajaj Pulsar 150cc connecting rod in dynamic loading conditions. In this study theconnecting rods modulate and simulated for the dynamicanalysis by using catia software for modelling-design ofconnecting rod and ansys 14.0 for dynamic analysis. Usingavailable high strengthen alloy is used for the connecting rodof Bajaj pulsar 150cc for the weight reduction to reducemoment of inertia. Dynamic analysis is carried out fordetermine the von mises stress, strain, and total deformation iscalculated under loading conditions of compression andtension at crank end and pin end of connecting rod.

Pravardhan S. Shenoy, Ali Fatemi,[4] “Connecting Rod Optimization for Weight and Cost Reduction”The objective of this study was to optimize a forged steelconnecting rod for its weight and manufacturing cost,taking into account recent developments.An optimization study was performed on a steel forgedconnecting rod with a consideration for improvement inweight and production cost. Since the weight of theconnecting rod has little influence on its total productioncost, the cost and the weight were dealt with separately.Reduction in machining operations, achieved by changein material, was a significant factor in manufacturing costreduction. Weight reduction was achieved by using aniterative procedure. Literature survey suggests cyclicloads comprised of static tensile and compressive loadsare often used for design and optimization of connectingrods.

Anil Kumar, Kama deep Grover, BalvinderBudania [5] “Optimization of Connecting Rod Parameters usingCAE Tools”, Aim of this work is to optimize weight and reduce inertia forces on the existing connecting rod, which is obtained by changing such design variables in the existing connecting rod design. The model was developed in Pro/E wildfire 5.0 and then imported as paranoid (xt) form in ANSYS workbench. In this work finite element analysis of the single cylinder four stroke petrol engine connecting rod is considered as case study. The Von Mises stress, strain and total deformation determined for the same loading conditions and compared with the existing results. Based on the observation of static FEA and the load analysis result, the load for the optimization study was selected same as on existing connecting rod. The current work consists of static structural analysis.

SatishWable, DattatrayS.Galhe, RajkumarL.Mankar [6] The main objective of this study is to review the weight optimization of a connecting rod in an automobile engine. To get the idea about designing the connecting rod, various stresses to be considered while designing the connecting rod and different materials used and comparing the result of all materials. To know the different software and Finite Element Method (FEM) packages useful for the modeling and analysis of connecting rod.All researches mentioned in this study give the idea about designing of the connecting rod. It explains about the various stresses to be considered while designing the connecting rod and different materials used and comparing the result of all material.

Mr. Sahel, Mr. JitenSaini [7] In this study static and modal analysis is performed .The S-N approach by modified Goodmancriterion to the fatigue life prediction of the connecting rods is also presented. The model is developed using SolidModeling software-Solidworks2013 .Further finite element analysis is done using Ansys14 Workbench to determine the von-misses stresses and strains, fatigue life and modal frequencies under different loading conditions. This study is based on the static structural module. Further analysis of connecting rod can be done under dynamic environment.

Samper NasirMomin, R.J. Gowanda [8]This study incorporates FEA modal analysis and experimental modal analysis of connecting rod. A parametric model of Connecting rod is modeled using CATIA V5 R19 software and finite element analysis is carried out by using ANSYS Software. Finite element method is used to determine natural frequencies of a connecting rod and compare results with FFT analyzer. FFT analysis is done by hanging the connecting rod at small end and experimental results were compared with FEM.

Hitesh Kumar, Vijay Kumar Sharma [9]The main idea of this study is to do analysis of connecting rod and get idea of stress producing compressive loading. And then give idea about weight reduction opportunities in connecting rod of an I.C. engine by examining two materials, AISI 1040 carbon steel and AISI 4340 alloy steel. This has entailed performing a detailed load analysis. Therefore, this study has deal with two subjects, first, static load and stress analysis of the connecting rod and second optimization for weight reduction and shape.

Gaba. Peeyush, Sethi APS [10] from the viewpoint of functionality, connecting rods must have the highest possible rigidity and fatigue strength at the lowest weight. Due to its large volume production, it is quite logical that optimization of the connecting rod for its weight or volume will result in large-scale savings. It will also achieve the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy. The major stress induced in the connecting rod is a combination of axial and bending stresses in operation. This paper deals with the stress analysis of connecting rod and guidelines for its finite element simulation. The definitions of critical load cases and the High Cycle Fatigue (HCF) have been explained.

B. Kuldeep[11] From the Modal analysis and the mechanical performance tests of the developed con rods, the following conclusions are drawn in terms of their mechanical and dynamic behaviour, Component testing indicates that the composite con rod can with1stand a 13.5 % higher compressive load and a 16.5 % higher tensile load than that of the Al7075 con rod.In failure analysis, deformation under tensile loading is higher in an Al7075 con rod then that with a composite con rod. In the area between the I-beam and piston pin end, the con rod's predominant fracture location was found. In contrast to composite con rods, which have cracks that begin maybe from the surface or the sub-surface, the Al7075 con rod's crack began at the surface. The damping behaviour of a component is considerably affected by the material's inherent damping property. Modal analysis shows that the composite con rod has a higher natural frequency because it is stiffer. This means that it takes a bigger shock to damage the structure.

S.Kaliappan[12] The new mechanism reduces the oscillation angle drastically compared with conventional mechanism. The side thrust of piston is reduced due to the reduction in oscillation angle. The magnitude of side thrust is high in conventional engine. Separation force on innovative connecting rod gives rise to an additional side thrust where there is no such additional side thrust in conventional engine. The net side thrust of engine with new connecting rod is still lower than the net side thrust of conventional engine. The power is transmitted from connecting rod through gear. So creation of noise will be a major problem. Best transmission angle occurs for maximum part of each stroke. It enhances the transmission of force from connecting rod to drive shaft.

Aisha Muhammad[13] In any manufacturing process, product quality and cost are the two critical, essential factors that need to be carefully selected by the company to maximize profit without compromising the product quality. This paper demonstrates the structural and optimization analysis of a connecting rod suitable for diesel engine applications. The structural analysis assesses the endurance of the connecting rod when subjected to static load along with the deformation characteristics. Weight and structural optimization of the connecting rod together through Finite Element Method using ANSYS is carried out and presented. The processes were performed under a loading of 30KN static force on a connecting rod of structural steel material. Further comparison of the analysis result before and after the optimization is done to ascertain the available optimal design.

Dilip Verma[14] An optimized solution is the minimum or the maximum value that an objective function can take under a given set of constraints. The load cycle consists of compressive gas load corresponding to maximum torque and dynamic tensile load corresponding to maximum inertia load. A finite element routine is first used to calculate the displacements and the stresses in the connecting rod, which is further used in another routine to calculate the total life. For this optimization problem, high priority is given to the weight of the connecting rod. Change in the material, there by resulting in significant reduction in the machining cost is the key factor in the optimization process. During optimization, weight and cost are dealt separately.

Anurag Vijayvergiya[15] In this paper the structural and buckling analysis of different cross-section type of connecting rod based on finite element method is done using ANSYS WORKBENCH. Three different cross-section I section, H-section, rectangular-section are select. On comparing the result for all type of connecting rods, it is observed that the value of total deformation, equivalent elastic strain and von misses stress are less for I-section, moderate for H-section and high for rectangular section and also the safety factor is high for I-section. From buckling analysis it is observed that first buckling load is high for H section and very less for rectangular section.

2.2. Summary of review

• Review of literature shows that many authors have reported the design to find bestconfiguration of the Connecting rod in terms of the geometry and they also study on modification of geometry and strength to withstand sudden change in stress while in operating.

• Analysis is done to check the effect of variable design parameter with some boundary condition.

• Also, finding the effective design which will reduce the weight of structure without compromising on strength.

• Therefore it is important to determine the factors like stresses, deformation, force etc. which influences the failure, finding the alternate design.

• New design must be able to withstand all of the loads it is going to undertaken

3. Methodology

Literature Survey:

Using the knowledge from literature review, we can know how the CAD model is to be prepared. The conditions required for applying various constraints and how the loads are applied is briefed about in the technical papers referred.

CAD Model Generation

- Getting input data on dimensions of connecting rod.
- Creating 3D model in CATIA.

Determination of loads:

Determination of different loads and boundary condition acting on the component by studying various ref papers, and different resources available.

Testing and Analysis

- Meshing the CAD model and applying the boundary conditions.
- Solve for the solution of meshed model using ANSYS.

Re-Design, Analysis and Results

- Making changes in CAD model for optimization.
- Carrying different iteration by removing material or changing topology based on ANSYS results.

- Check the maximum stress ensuring it is well within the safe region.

Fabrication, Experimental validation and Result

- Fabrication of prototype.
- Suitable experimentation and comparison with present connecting rod.
- Validation of result by comparing with software results.

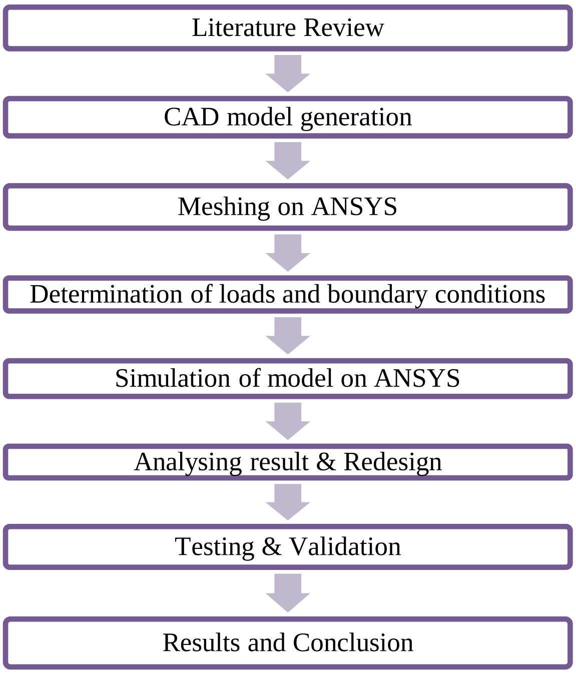


Fig.3.1. Flowchart of project methodology.

Design of Connecting Rod

The chapter Design and Analysis of connecting rodof dissertation includes making design og connecting rod using reverse engineeeirng. Dimensions of the existing connecting rod through reverse engineering have been measured and CAD model of a connecting rod have been prepared in CATIA V5. The finite element analysis is carried out by using Hypermesh and ANSYS as post-processor.

CAD (Computer-Aided Design):

Computer-aided design (CAD), also known as computer aided design and drafting (CADD), is the use of computer technology for the process of design and design documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provides the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes.

CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

CATIA

Computer **A**ided **T**hree dimensional **I**nteractive **A**pplication (**CATIA**) is a software from Dassault systems, a France based company. CATIA delivers innovative technologies for maximum productivity and creativity, from the inception concept to the final product.

CATIA provides three basic platforms-

- P1 for small and medium sized process oriented companies that wish to grow towards large scale digitized product definition.
- P2 for advanced design engineering companies that require product, process, and resource modelling.
- P3 for high-end design applications and is basically for automotive and aerospcae industry, where high quality surfacing is used.

CAD Model of Connecting Rod:

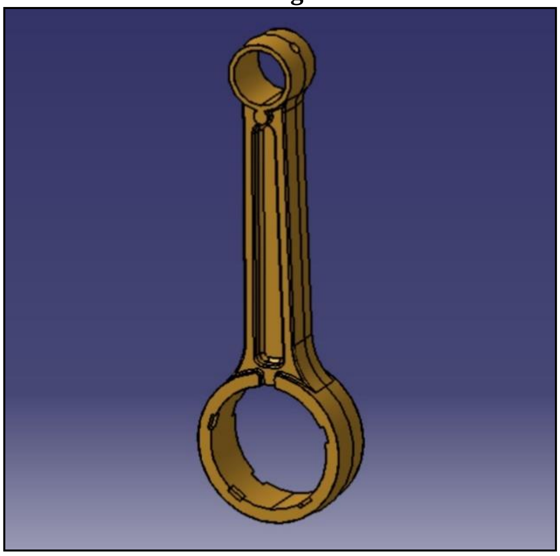


Fig.3.3. 3D model of connecting rod

ANALYSYS:

The **finite element method** (**FEM**), is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally require the solution to boundary value problems for partial differential equations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain.[1] To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

Studying or analyzing a phenomenon with FEM is often referred to as **finite element analysis** (**FEA**).

BASIC CONCEPTS:

The subdivision of a whole domain into simpler parts has several advantages:[2]

- Accurate representation of complex geometry
- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects.

A typical work out of the method involves (1) dividing the domain of the problem into a collection of sub domains, with each sub domain represented by a set of element equations to the original problem, followed by (2) systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

In the first step above, the element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). To explain the approximation in this process, FEM is commonly introduced as a special case of Galerkin method. The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero. In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

- a set of algebraic equations for steady state problems,
- a set of ordinary differential equations for transient problems.

These equation sets are the element equations. They are linear if the underlying PDE is linear, and vice versa. Algebraic equation sets that arise in the steady state problems are solved using numerical linear algebra methods, while ordinary differential equation sets that arise in the transient problems are solved by numerical integration using standard techniques such as Euler's method or the Runge-Kutta method.

In next step above, a global system of equations is generated from the element equations through a transformation of coordinates from the sub domains' local nodes to the domain's global nodes. This spatial transformation includes appropriate orientation adjustments as applied in relation to the reference coordinate system. The process is often carried out by FEM software using coordinate data generated from the sub domains.

FEM is best understood from its practical application, known as **finite element analysis (FEA)**. FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analysing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high fidelity situations. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

STATIC ANALYSIS OF EXISTING LATHE CHUCK:

Properties	
Volume	1.7118e-005 m^3
Mass	0.13438 kg

MESH

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multi physics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation. Creating the most appropriate mesh is the foundation of engineering simulations. ANSYS Meshing is aware of the type of solutions that will be used in the project and has the appropriate criteria to create the best suited mesh. ANSYS Meshing is automatically integrated with each solver within the ANSYS Workbench environment. For a quick analysis or for the new and infrequent user, a usable mesh can be created with one click of the mouse. ANSYS Meshing chooses the most appropriate options based on the analysis type and the geometry of the model. Especially convenient is the

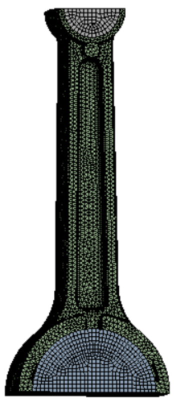
International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 04 | April - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

ability of ANSYS Meshing to automatically take advantage of the available cores in the computer to use parallel processing and thus significantly reduce the time to create a mesh. Parallel meshing is available without any additional cost or license requirements.

Statistics	
Nodes	182432
Elements	105828

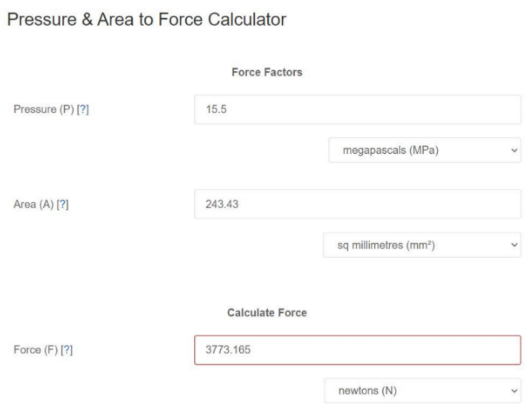
After meshing of 2 wheeler connecting rod the Number of nodes 182432 and Elements 105828

Boundary condition



RESULT :

TOTAL DEFORMATION AND EQUIVALENT STRESS


Pressure & Area to Force Calculator

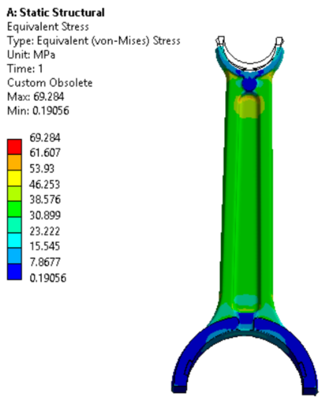
Force Factors

Pressure (P) [?] 15.5 megapascals (MPa)

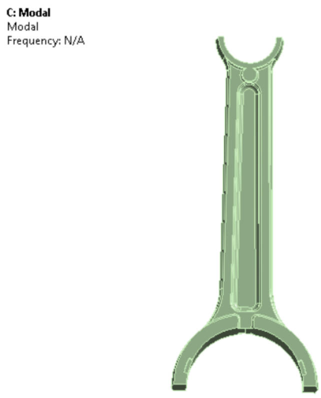
Area (A) [?] 243.43 sq millimetres (mm²)

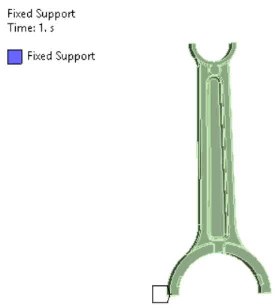
Calculate Force

Force (F) [?] 3773.165 newtons (N)

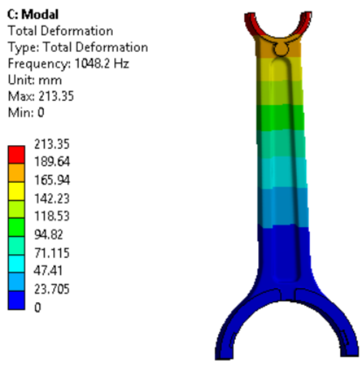


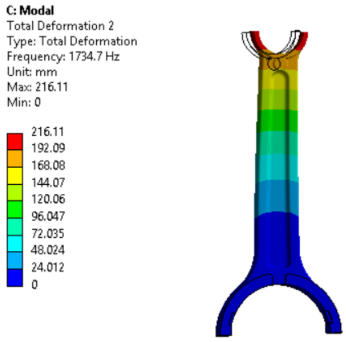
MODEL ANALYSIS AND FIXED SUPPORT

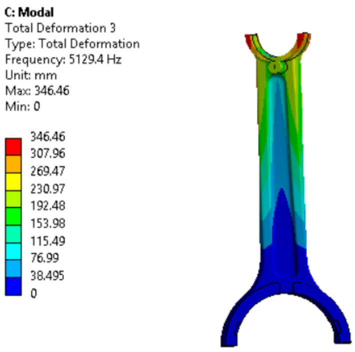


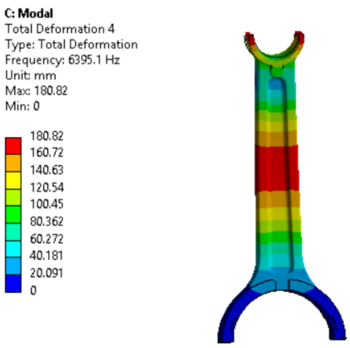


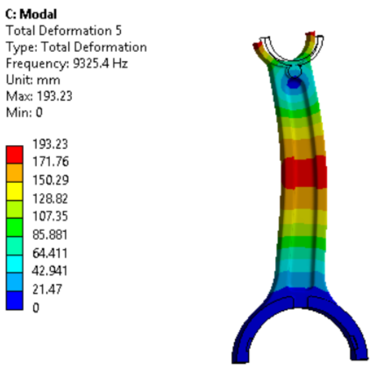
TOTAL DEFORMATION

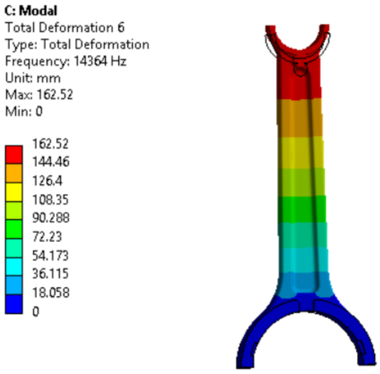












Tabular Data

	Mode	Frequency [Hz]
1	1.	1048.2
2	2.	1734.7
3	3.	5129.4
4	4.	6395.1
5	5.	9325.4
6	6.	14364

TOPOLOGY OPTIMIZATION

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

Basic Theor

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

Three optimization methods correspond to the three stages of the product design process, namely the detailed design, basic design and conceptual design. Size optimization keeps the structural shape and topology structure invariant, to optimize the various parameters of structure, such as thickness, section size of beam, materials' properties; shape optimization maintains the topology structure, to change the boundary of structure and shape, seek the most suitable structure boundary situation and shape; topology optimization is to find the optimal path of materials' distribution in a continuous domain which meet the displacement and stress conditions in structure, make a certain performance optimal. Thus, compared to size and shape optimization, topology optimization with more freedom degree and greater design space, its greatest feature is under uncertain structural shape, according to the known boundary condition and a given load to determine the reasonable structure, both for the conceptual design of new products and improvement design for existing products, it is the most promising aspect of structural optimization. For continuous structure topology optimization, there are some mature methods like: uniform method, evolutionary structural optimization method, variable density method etc. Uniform method introduced cell structure of micro structure (unit cell) in the elements of the structure, each unit cell has three forms, namely non-material voids (size = 1), isotropic-material entity medium (size = 0) and orthotropic-material opening-hole medium (0 < size < 1). Wherein the distribution of each form will be able to describe the form of topology and the shape of structure; evolutionary structural optimization method believes that stress in any parts of the structure should under the same level in an ideal structure. That means the local material with a low stress state is not fully utilized, so you can delete the material artificially. So gradually remove material which in a low stress state, and then delete the update rate, so optimized structure becomes more uniform. Variable density method is used to conduct optimization in this paper. The basic idea is to introduce a hypothetical material which density is variable and range from 0 to 1.

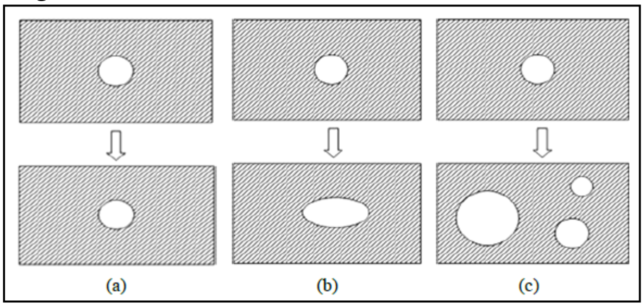


Fig: Three kinds of structure optimization. (a) Size optimization,(b) Shape optimization, (c) topology optimization.

After changing the continuous structure to finite element model, then making per units' density as the design variables, to convert topology optimization problem into the optimal distribution of material, while in order to suppress the intermediate density of the material (material between void and entities), the introduction of a interpolation penalty factor used to describe the relationship between Young's modulus and density of the material which shown as the following formula:

$E = x^p E_0$

where: p—interpolation penalty factor ($p > 1$); E_0—Young's modulus of densified material unit; E—Young's modulus after interpolation. So that the original model with intermediate density will be eliminated or replaced by densified material and will achieve optimal results which close to the entity. Therefore, during topology optimization, variable is relative density of units, then structural topology optimization problem is converted into the optimal distribution of the material.

Process of Topology Optimization

Based on Hyper Works platform topology optimization holder, first, according to the engine mounting position, we establish the three-dimensional geometric model of engine bracket, and then pre-treated in HyperMesh, define design area, objective function and constraints under the optimization panel, finally operate topology optimization which design process is as Figure 2.

Analysis

To establish the geometry model by CATIA, then input the geometry to the ANSY to carry out pre-treatment operations like geometry clean up, meshing, loads, constraints, etc.

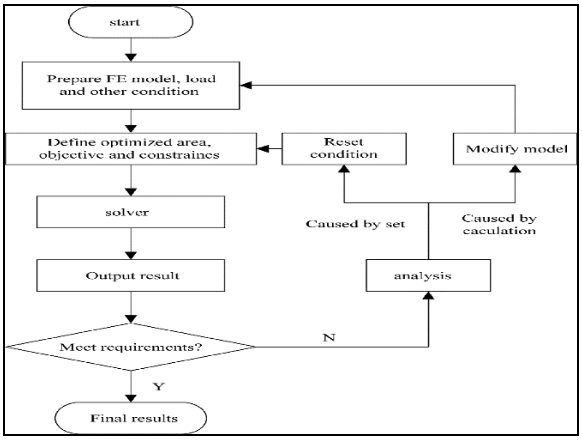


Fig : Process of Topology Optimization Initially we need to collect the information regarding different loads acting on the bracket and the packaging data for fixing design space. The base bracket results from testing and finite element analysis (FEA) point of view for evaluating final optimized design.

Steps in Topology Optimization

The topology optimization consists of the following sequence of steps.

• Define the design space
• Define optimization parameters
• Material removal process and detail design

➢ **Defining the Optimization Parameters**

The aim of topology optimization in this project is to minimize the volume without affecting the bracket stiffness and strength compared to base bracket, so the design objective is taken as to minimize the volume. Following parameters are defined as constrains:

1. Allowable stress limit value is defined as stress constraints from durability point of view
2. Single draw direction is defined as manufacturing constraint.

➢ **Material Removal Process and Detail Design**

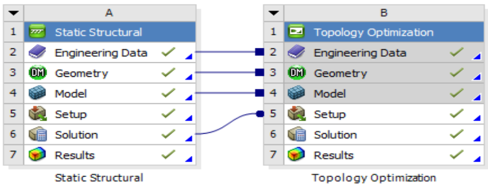
The optimization process took some iteration to remove the unnecessary material from the design space. The output of the topology optimization, an intermediate model which may be called a topology-based model, is constructed by removing unnecessary materials from the rough conceptual model.

- Additive Manufacturing

Without manufacturing constraints, optimized components often have complex geometries. Recent developments in Additive Manufacturing (3D printing) enable the fabrication of these optimized designs without compromise. Topology optimization forms the natural design technology for Additive Manufacturing, as it fully exploits its potential.

- Shape Optimization

Where topology optimization excels at automated concept generation, shape optimization allows for efficient final fine tuning of designs. However, every fabrication technique has its shape/property accuracy limitations, particularly at the micro scale. Our activities on shape optimization concentrate on dealing with fabrication inaccuracy, and optimizing for robustness.



Step 1: Details of material namely copper, steel, grey cast iron, composite material, fluid domain material is defined in engineering data. i.e. ANSYS default material is structural steel.

Step 2: Import of geometry created in any CAD software namely CATIA, PRO E, SOLIDWORK, INVENTOR etc. in geometry section. If any correction is to be made it can be created in geometry section in Design modeller or space claim.

Step 3: In model section after import of component

- Material is assigned to component as per existing material
- Connection is checked in contact region i.e. bonded, frictionless, frictional, no separation etc. for multi body components.
- Meshing or discretization is performed i.e. to break components in small pieces (elements) as per size i.e. preferably tetra mesh and hexahedral mesh for 3D geometry and for 2 D quad or tria are generally preferred.

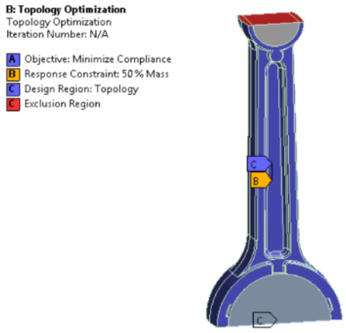
Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

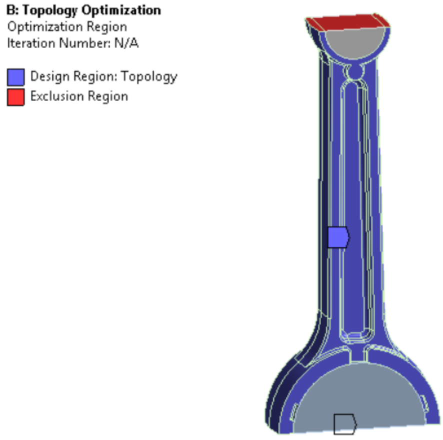
Step 5: Now problem is well defined and solve option is selected to obtain the solution in the form of equivalent stress, strain, energy, reaction force etc.

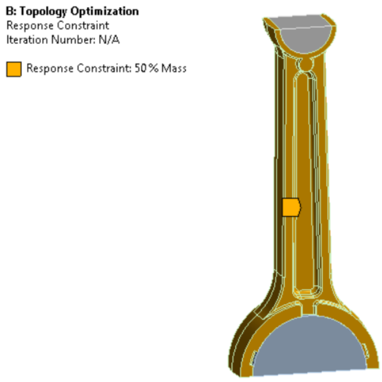
Step 6: Topology optimization tool is selected from ANSYS list view and drag and dropped in solution section of static structural so that it takes its all boundary condition, geometry, all details and perform topology optimization on selected component.

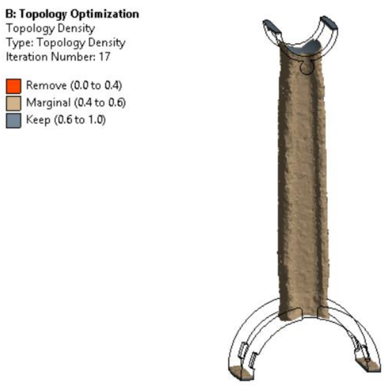
Step 7: Define the topology density and element density section as output. After performing topology algorithm on component, selected parts contain red, brown and grey colour on it. So, red region colour indicates material removal area along with marginal (brown colour) and grey colour to keep material.

Step 8: After selection of specific shape material is removed from component and reanalysis of component is performed to observe the sustainability of existing optimized component under same boundary condition.









References

i. D.Gopinatha,Ch.V.Sushmab, "Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis", 4th International Conference on Materials Processing and Characterization, ScienceDirect.

ii. Mohammed Mohsin Ali Ha , Mohamed Haneef b, "Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At The Big End", 4th International Conference on Materials Processing and Characterization, ScienceDirect.

iii. Amit Kumar, Bhingole P.P. , Dinesh Kumar, "Dynamic Analysis of Bajaj Pulsar 150cc Connecting Rod Using ansys 14.0", Asian Journal of Engineering and Applied Technology

iv. Pravardhan S. Shenoy, Ali Fatemi, "Connecting Rod Optimization for Weight and Cost Reduction", SAE International

v. Anil kumar, Kamaldeep Grover, BalvinderBudania, "Optimization of Connecting Rod Parameters using CAE Tools",

vi. SatishWable, DattatrayS.Galhe, RajkumarL.Mankar, "review on design and optimization of connecting rod"

vii. Mr. Sahil, Mr. JitenSaini, "Static, Fatigue and Modal Analysis of Connecting Rod under Different Loading Conditions", international journal of engineering sciences & research technology,

viii. Sameer NasirMomin , R.J. Gawande, "Investigation of Natural Frequency by FEA Modal Analysis and Experimental Modal Analysis of Connecting Rod", ISSN(Online)

ix. Hitesh Kumar, Vijay Kumar Sharma, "Stress Analysis and Optimization of I. C. Engine Connecting Rod by Finite Element Analysis", International Journal of Engineering Research &Technology (IJERT)

x. Gaba. Peeyush, Sethi APS, Design Evaluation of Connecting Rod using FEM for HighCycle Fatigue Strength, nternational Journal of Research in Advent Technology, Vol.2, No.5, May 2014

xi. B. Kuldeep "A novel composite connecting rod: study on mechanical and dynamic behaviour through experimental and finite element approach" 18 October 2023 https://doi.org/10.1016/j.jcomc.2023.100413

xii. S.Kaliappan "analysis of an innovative connecting rod by using finite element method" 2018

xiii. Aisha Muhammad "Static Analysis and Optimization of a Connecting Rod" DOI:http://dx.doi.org/10.15282/ijets.6.1.2019.1003

xiv. Dilip Verma" Design and Optimization of Connecting Rod" Vol. 4, Issue 1, pp: (38-43), Month: April 2016

xv. Anurag Vijayvergiya" Design and Shape Optimization of Connecting Rod End Bearing through ansys" http://dx.doi.org/10.47392/irjash.2021.261 Vol. 03, Issue 11 November

2. Bruce, K.B., Cardelli, L., Pierce, B.C.: Comparing Object Encodings. In: Abadi, M., Ito, T. (eds.): Theoretical Aspects of Computer Software. Lecture Notes in Computer Science, Vol. 1281. Springer-Verlag, Berlin Heidelberg New York (1997) 415–438

3. van Leeuwen, J. (ed.): Computer Science Today. Recent Trends and Developments. Lecture Notes in Computer Science, Vol. 1000. Springer-Verlag, Berlin Heidelberg New York (1995)

4. Michalewicz, Z.: Genetic Algorithms + Data Structures = Evolution Programs. 3rd edn. Springer-Verlag, Berlin Heidelberg New York (1996)