ISSN: 2582-7561

International Journal For Academic Research & Development

Vol. 3 (2021)

ISSUE 4

(Multidisciplinary)

E-mail Id: editor@iifard.org

url: www.iifard.org/about-journal/

INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT
INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT
https://iifard.org/web/AWABbgJhBD0=/IJARD
www.iifard.org

ISSN: 2582-7561

IJARD

Volume 3, Issue 4
2021

Published by

International Institute For Academic Research and Development
Email: trustforacademic@gmail.com, editor@iifard.org

www.iifard.org

INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT

INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT

https://iifard.org/web/AWABbgJhBD0=/IJARD
www.iifard.org

ISSN: 2582-7561

About Journal - I J A R D

International Journal for Academic, Research and Development (IJARD) is an autonomous, peer reviewed online journal. It mainly serves as universal discussion identified with building training, distributed at present quarterly. IJARD welcomes researchers and academicians to submit their original research work which meets the journal criteria of significance and scientific excellence. IJARD is scholarly online open access and peer received journals emphasizing on research studies and application in the field of Science, Engineering and Management. Researchers are requested to submit their original articles online for a peer review and analysis before its publication. The editorial board encourages academicians, research scholars and partitions to publish their articles related to science, engineering and management and relevant fields.

IJARD publishes its journals in full open access format. The scientific community and the general public can unlimitedly and immediately access all content published in our journals for free as soon as it is published on the Internet. Therefore, IJARD needs to defray its editorial and production costs by collecting article processing charges from authors' institutes for research funding bodies. IJARD is committed to keep its open access publication charges at a minimum level.

We believe that immediate, worldwide, barrier-free, open access to the full text of research articles is in the best interests of the scientific community. We also intend to build mutually beneficial and long-lasting relationships with our authors and always provide them full support throughout the publishing and the post-publishing processes.

IJARD invites contribution in the following categories:

Original research work.
Review articles, comprehensive review on a topic.
Survey on a topic
Self-contained articles on ongoing research.
Technical Notes.

IJARD is committed to publishing only original work including research which has neither been published elsewhere, nor is under review elsewhere. All manuscripts that are found to have been plagiarized from a manuscript by other authors, whether published or unpublished, will incur plagiarism sanctions. Conference papers are very welcome; however it is imperative that these papers are significantly revised and updated. Actually, such papers have a better chance of acceptance if their merits were already prescreened at high quality conferences.

Manuscript should be submitted at https://iifard.org/web/AmMJZlY1V24=/IJARD
All correspondence should be made to editor@iifard.org trustforacademic@gmail.com
Phone no./WhatsApp: +918005351780, +67581509800

Welcome to International Journal for Academic, Research and Development ! ! !

INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT

INTERNATIONAL JOURNAL FOR ACADEMIC RESEARCH AND DEVELOPMENT

https://iifard.org/web/AWABbgJhBD0=/IJARD

www.iifard.org

ISSN: 2582-7561

Indexed with

DRJI

Scope Database
Journal indexing & Citation Analysis

ADVANCED SCIENCE INDEX

zenodo

ASOS indeks

OpenAIRE

Google Scholar

ISSN
INTERNATIONAL
STANDARD
SERIAL
NUMBER
INTERNATIONAL CENTRE

International Journal for Academic Research and Development
ISSN 2582-7561(Online)© I J A R D
Vol. 3 Issue 4
2021

Fatigue Analysis of I-section and H- Section Connecting Rod using ANSYS Workbench

Ahamad Sadique *
shabaansari8855@gmail.com
Department of Mechanical Engineering, Bansal Institute of Engineering & Technology, Lucknow, India
Kapoor Anil
raj321anil@gmail.com
Department of Mechanical Engineering, Bansal Institute of Engineering & Technology, Lucknow, India

Abstract: *The connecting rod serves as the connecting point between the crank and piston. Two forces that are produced by mass and fuel combustion are applied to connecting rods. The axial and bending strains are caused by these two forces. A connecting rod must be able to transmit axial compression/tension and bending stresses brought on by the thrust and pull on the piston as well as by centrifugal force. Bending stresses are induced by eccentricity, the crankshaft, case wall deformation, and rotating mass force. Investigated in this work is connecting rod fatigue. ANSYS Workbench 14.0 is used to conduct the investigation. The goal of this study is to identify and analyze the connecting rod technical fault that is impacted by reversal loading its service life.*

The results indicate that with fully reversal loading, one can expect a longer life for the connecting rod and also identify any potential weak areas. Additionally, 10^8 load cycles were added to the permitted number when fully reversal loading was used. The findings could potentially lead to changes in the connecting rod's design, some believe.

Keywords: Connecting Rod, Fatigue analysis, critical points, Finite Element Analysis, ANSYS Workbench.

1. Introduction

Connecting rods are commonly employed in a variety of engines to transport the piston's push to the crankshaft, resulting in the conversion of the piston's reciprocating motion to the crankshaft's rotating motion. It is made up of three parts: the crank end, the shank portion, and the pin end. To allow proper bearing installation, crank-end and pin-end pin holes are drilled. With the aid of a piston pin, one end of the connecting rod is attached to the piston. The other end, which spins with the crankshaft, is split to allow it to be fastened around it. The two halves are held together by two or four bolts, depending on the size of the large end. The forces created by mass and fuel combustion act on connecting rods. Axial and bending stresses are produced by these two forces [1].

The tractor's connecting rods are generally composed of cast iron by powder metallurgy or forging. The fundamental motivation for using these technologies is to build the components as a whole and achieve great productivity at a low cost [2] while also optimizing the geometry of the connecting rod. Because the engine is designed to perform in a variety of challenging settings, the connecting rod design is intricate. The inertia force and rod mechanism due to acceleration/retardation in a cycle subject the connecting rod to changing pressure [4]. The fatigue analysis and offered a connecting rod design [5]. The rupture caused by fatigue and a way for adjusting the connecting rod design specifications [6]. The way for enhancing the connecting rod design. The Finite Element Method (FEM) is a recent method for fatigue study of connecting rods and component lifespan estimate. FEM is capable of creating strain/stress distributions across the component, allowing us to accurately locate critical spots [7]. This approach is particularly beneficial when the component's geometrical form is complicated and the loading circumstances are simple. The relevant component parameters, such as material, cross section conditions, and so on, may be changed in FEM, and component optimization under fatigue cycle loading can be done easily and fast [6]. The examination of a component is accomplished in a virtual environment without the need for a prototype in Computer Aided Design [8], resulting in time and cost savings. Because fatigue is the most common cause of connecting rod failure, the used ANSYS software to perform FE analysis on the U650 Tractor connecting rod and

*Corresponding Author
found that the critical point under reverse loading (compressive and tensile) is 46. This value can be enhanced by lowering the stress concentration coefficient in order to improve the connecting rod's fatigue life [9].

The FE analysis was performed on the ANSYS workbench in this study, and the impacts of connecting rod design parameters were explored to increase performance under cyclic loadings. It has been discovered that lowering the stress concentration coefficient and changing certain of the connecting rod design parameters will enhance fatigue life.

2. FE Modeling Of The Connecting Rod

This chapter addresses the creation, simplifications, and correctness of the connecting rod models utilised in Finite Element Analysis. The creation of meshes and their convergence are explored. Also included is the application of loads, including the distribution of loads at the contact region, impacts on load distribution, estimate of pressure constants based on produced force magnitude, application of restraints, and validation of the finite element analysis model.

In many engineering disciplines, FEA is now a crucial step in the design or modelling of a physical phenomenon. Typically, a physical phenomenon involves a number of field variables and happens in a continuum of matter. There are an endless number of possible solutions because the field variables vary from location to location. In the context of this book, a domain is a continuum with a recognised border.

3. Finite Element Modeling of I Section Connecting Rod

This section discusses the production, simplifications, and correctness of the I section connecting rod modelling used for Finite Element Analysis. The creation of meshes and their convergence are explored. Additionally, the application of loads is discussed, including the distribution of loads at the contact region, the influences on load distribution, the estimation of pressure constants based on the magnitude of the generated force, the application of restraints, and the validation of the Finite Element Analysis model.

4. Modeling of I Section Connecting Rod

A collection of uniform guidelines for computer and mathematical modelling of three-dimensional solids is known as modelling or solid modelling. The emphasis on physical authenticity in solid modelling sets it apart from comparable fields like computer and geometric modelling graphics. The foundation of computer-aided design is built on the principles of geometric and solid modelling, which also generally facilitate the production, animation, interchange, annotation, visualization, and interrogation, of digital models of real-world things.

Any solid representation scheme is a way to learn about the class of semi-analytic subsets of Euclidean space based on presumptive mathematical features. This implies that every representation is just a distinct data structure that organizes the same geometric and topological facts in a different way. In terms of a finite number of operations on a collection of primitives, all representation schemes are arranged. As a result, each given representation's modelling space is limited, and it's possible that no single representation scheme will adequately describe all different kinds of solids. For instance, except in extremely simple circumstances, solids formed by combinations of regularized Boolean operations cannot always be represented as the sweep of a primitive travelling down a space trajectory. Modern geometric modelling systems are compelled to maintain multiple solid representation schemes and to enable the effective conversion between representation schemes.

Figure 1 displays the connecting rod's geometrical specifications. The Finite Element analysis of the connecting rod for both the I and H section used the material parameters of the connecting rod as indicated in Table 1.

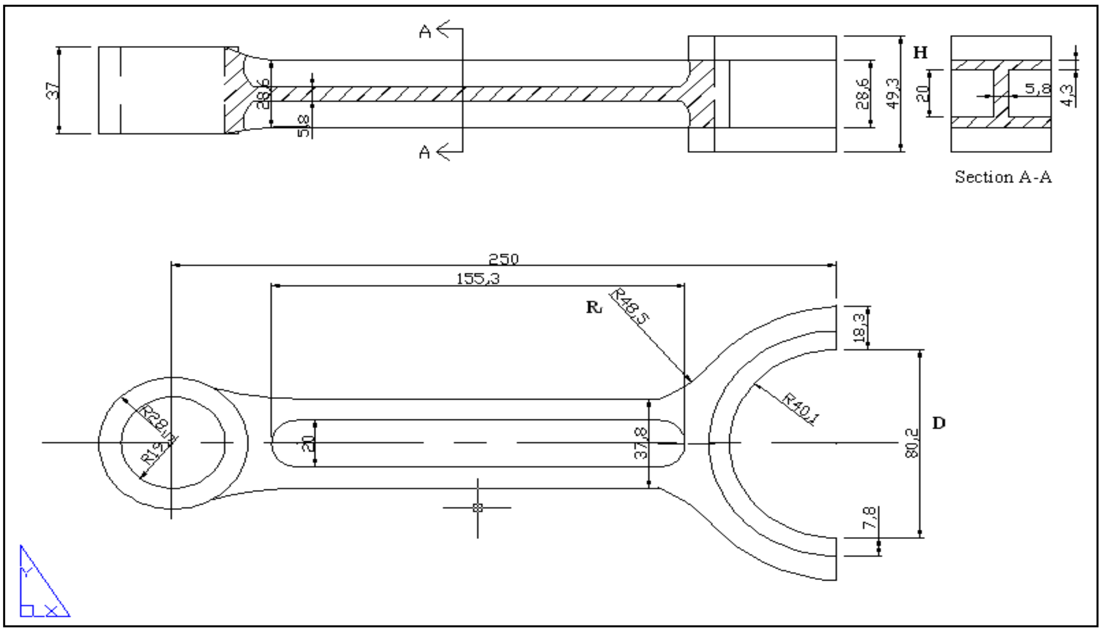

Figure 1. The I section connecting rod's geometry

Using Pro-E Designing Software, a geometrical model of an I section connecting rod was created, as illustrated in Figure 2. I section connecting rod weighs 1705 gm, according to a weighing scale. The weight difference between the component utilised by M. Omid et al. [9] and the solid model used for FEA is less than 1%. This demonstrates how accurate the solid model is.

Table 1. The connecting rod properties used for Finite Element Analysis

Input Parameters	**Values**
Yield Strength	483 MPa
Shear Modulus	79 GPa
Brinell Strength	229-269 HB
Tensional strength	621 MPa
Young's Modulus	207 GPa
Correction Coefficient	0.8
Poison Ratio	0.3
Connecting Rod Material	C-70 Alloy steel
Density	7.7 Mg/m^3

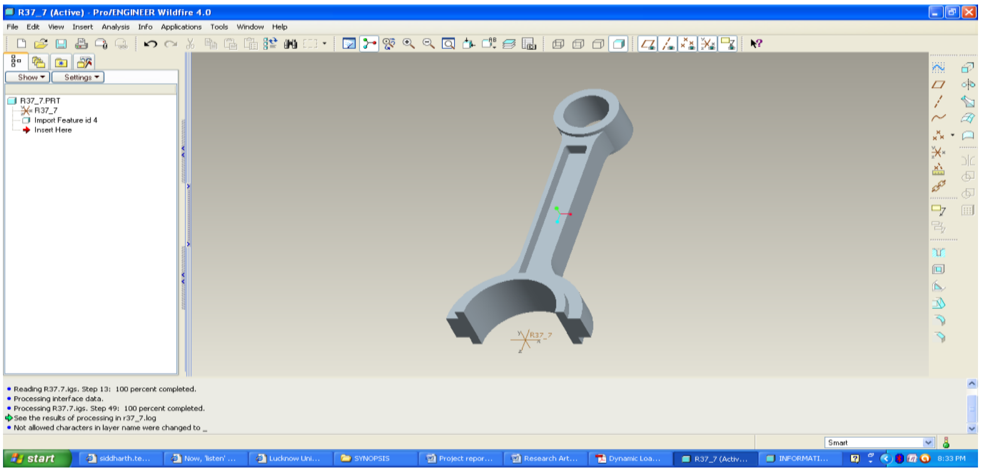

Figure 2. I section Connecting Rod Geometrical Model

By contrasting the regions along the web and on each side of the axis of symmetry perpendicular to its length, the degree of non-symmetry in the shank area for an I section connecting rod was roughly 5%. This loss of symmetry was a production variation that wasn't intentional. Since it is symmetric, the connecting rod has been modelled as such.

4.1 Mesh Generation Of I Section

Mesh generation is the process of producing a polygonal or polyhedral mesh that closely reflects a geometric domain. The phrase "grid generation" is frequently used synonymously. Rendering to physical simulations and computer screens like FEA and CFD are typical uses. Although the input model form might vary widely, STL, B-rep, CAD, and NURBS are frequent sources (file format). With contributions from engineering, computer science, and mathematics, the topic is very interdisciplinary. Tetrahedral, pyramidal, prismatic, or hexahedral shapes must be used for creating three-dimensional meshes for FEA. Any type of polyhedron may be employed in those that are used for the finite volume approach. Those required for finite difference approaches typically need to be made up of multi-block structured meshes, which are piecewise arrays of hexahedra.

The FE analysis is performed using Pro/E and ANSYS workbench software. Pro/E software was used to create a 3D model of the I section connecting rods, after which an IGES file was produced and imported into the ANSYS workbench Programme. The connecting rods needed to be meshed next. Figure 3. depicted the I section mesh model. As indicated in Figure 4., the 10-node tetragonal elements were employed. Tetragonal elements with a 2 mm element length were used to create the finite element mesh (2262 elements). This element was chosen in order to simplify the geometrical elements of a complex mechanical component and to help us obtain more accurate findings for the fatigue life calculation.

A higher order, 10-node, 3-D element is SOLID187. SOLID187 is highly suited to modelling irregular meshes because of its quadratic displacement behaviour. Ten nodes, each with three degrees of freedom—translations in the nodal x, y, and z directions—define the element. The element is capable of great deflection, huge strain, creep, stress stiffening, plasticity, and hyperelasticity. Additionally, it offers the capacity to use mixed formulations to simulate the deformation of fully incompressible hyperelastic materials as well as virtually incompressible elastoplastic materials.

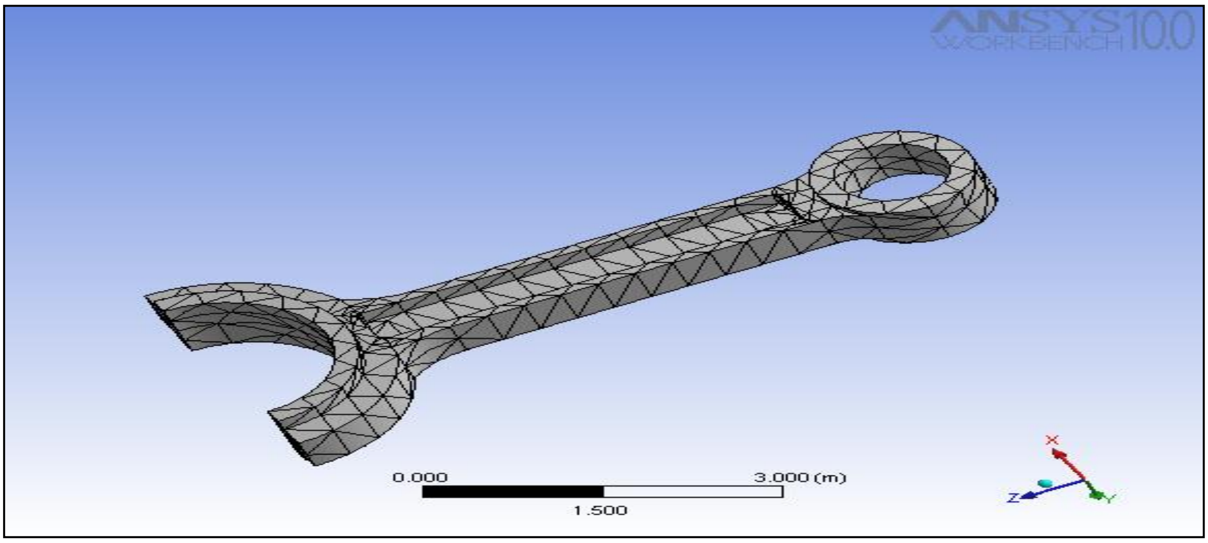

Figure 3. I section Connecting Rod Meshing Model

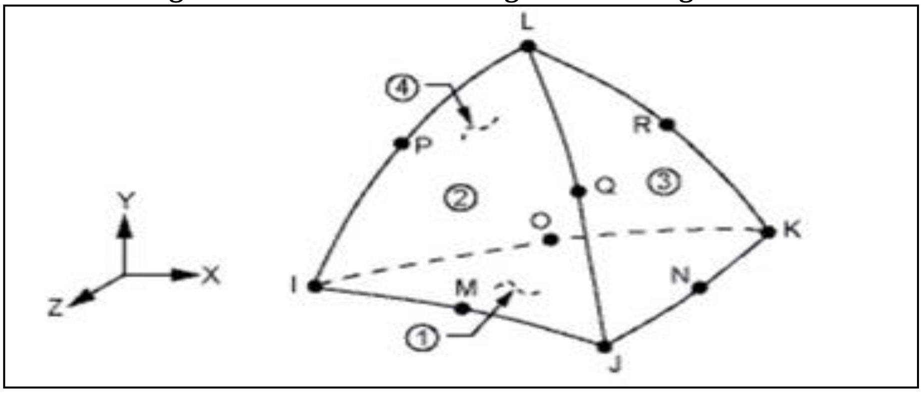

Figure 4. The 10 – node tetragonal elements

4.2 Boundary Conditions For I Section

4.2.1 Loading

The big ends are considered to be under tensile loading, as shown in Figure 5., with a sinusoidal distributed loading throughout the contact surface area, as shown in Figure 1. Based on the findings of an experiment, [10]. On the contact surface, the normal force is determined by:

$$F = F_o \cos \theta \tag{3.1}$$

The total resultant load is given by when the load is distributed over an angle of 180°.

$$F_t = \int_{-\pi/2}^{\pi/2} F_o(\cos^2\theta) rt d\theta = F_o rt\pi/2 \tag{3.2}$$

Figure 8. describes θ and r, t . The normal Force constant F_o is, therefore, given by:

$$F_o = F_t/rt\pi/2 \tag{3.3}$$

Four finite element models were examined in this study. Both compressive and tensile loads underwent FEA. For each case, two cases were examined: one with the other with the load applied at the piston pin end and the load applied at the crank end the crank end restrained and the piston pin end restrained. The axial load in the analysis was 9500 N in both compression and tension. Because the analysis is the stress, linear elastic, displacement, and strain in static analysis are proportional to the size of the load. Therefore, using a proportional scaling factor, the FEA results can be easily applied to different elastic load instances.

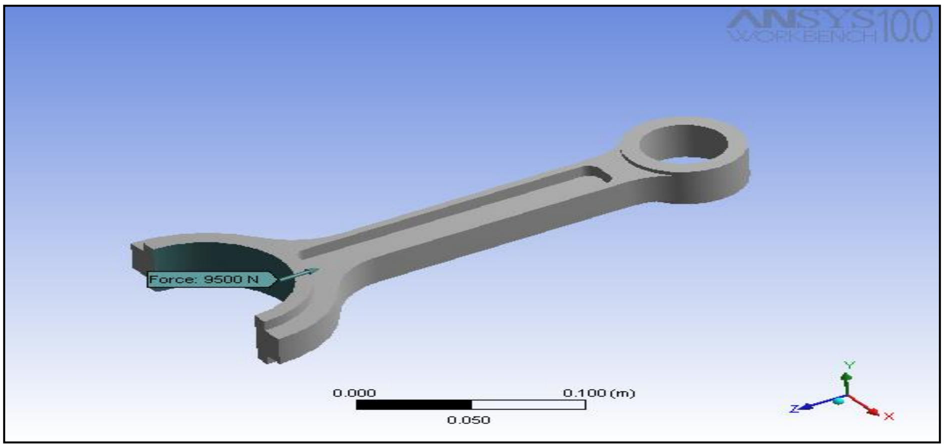

Figure 5. A sinusoidal distributed loading under tensile loading on I section connecting rod

4.2.2 Restraints

Four FEA models were solved, as was already described. A Finite Element Analysis model with a tensile load applied at the big end and a restrained small end is shown in Figure 6. Keep in mind that the piston pin's inner surface is completely constrained on one side (1800). Similar to this, 1200 of the contact surface area is completely restrained when the connecting rod is under an axial compressive load.

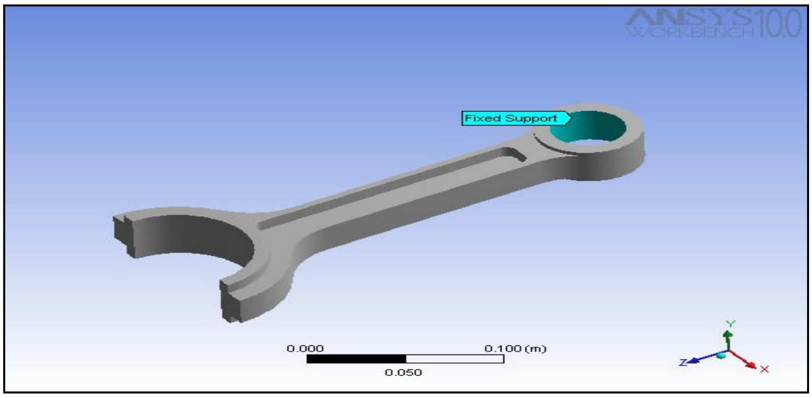

Figure 6. The small end of I section connecting rod

4.2.3 Fatigue analysis of I section connecting rod

The boundary conditions were first specified and applied a tension force. The tension force was then replaced by a compression force exactly of the same size and in the opposite direction, and the problem was once more resolved. The Von Misses stresses were activated throughout each stage of loading, and the crucial sites were identified. Figure 7. depicts the stresses placed on different parts of the H-section connecting rod, and Figure 8. illustrates how cyclic loading causes the critical area to be visible at the stem of the connecting rod close to the bearing or big end roundness.

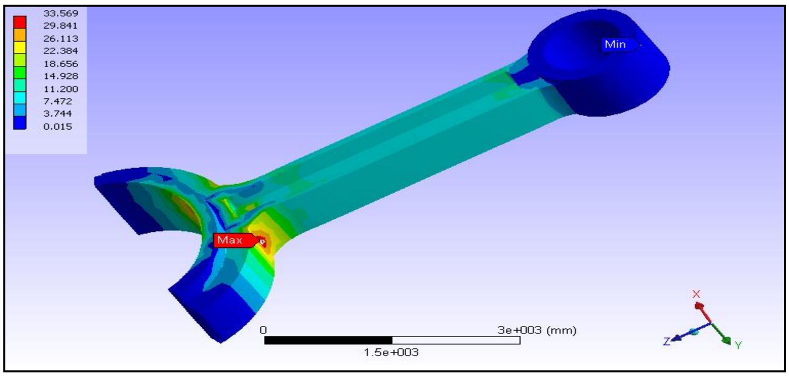

Figure 7. I section connection rod Fatigue Analysis

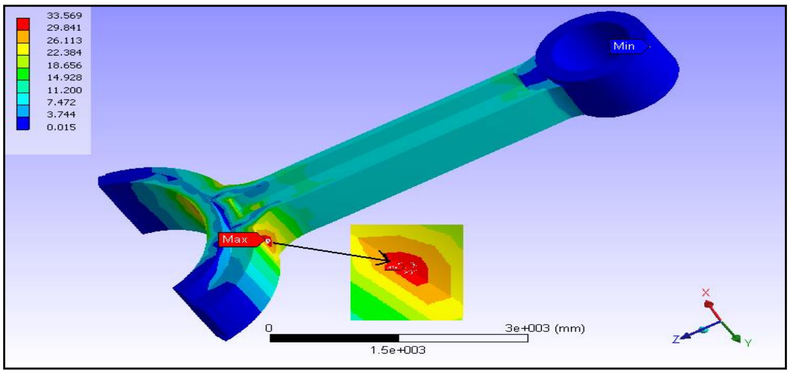

Figure 8. I section Connecting Rod Critical point

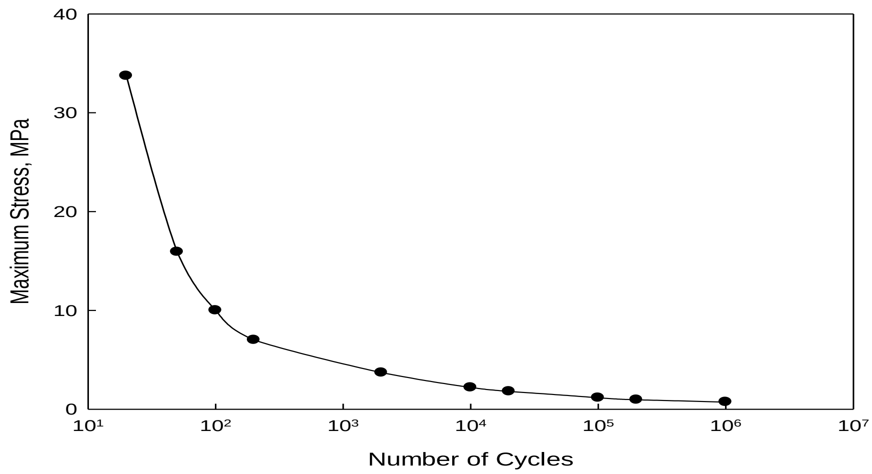

Figure 9. I section connecting rod subjected to a fatigue loading

These important spots were identified after which they were chosen as the areas to be investigated for fatigue. The S-N data gathered from the fatigue test of the particular alloy into the programme should import is illustrated in Figure 9., filling in the fatigue parameter blanks. After applying 10^6 force cycles to the model, a partial consumption rate was obtained, showing the ratio of the applied force cycles to the permissible ones for each node. We can change specific geometrical dimensions while maintaining the same values for all other dimensions towards the big end in order to enhance and maximise the fatigue life of a connecting rod.

5. FE Modeling of H Section Connecting Rod

This section discusses the production, simplifications, and correctness of the H section connecting rod modelling used for Finite Element Analysis. The creation of meshes and their convergence are explored. There is also discussion of the application of loads, the factors that affect load distribution, the use of restraints, the calculation of pressure constants based on the size of the resultant force, the validation of the Finite Element Analysis model and notably the distribution of loads at the contact region.

5.1 Modeling of H Section Connecting Rod

Figure 10. displays the H section connecting rod's geometrical dimensions. The FE analysis of the connecting rod for H section used the material parameters of the connecting rod as indicated in Table 1.

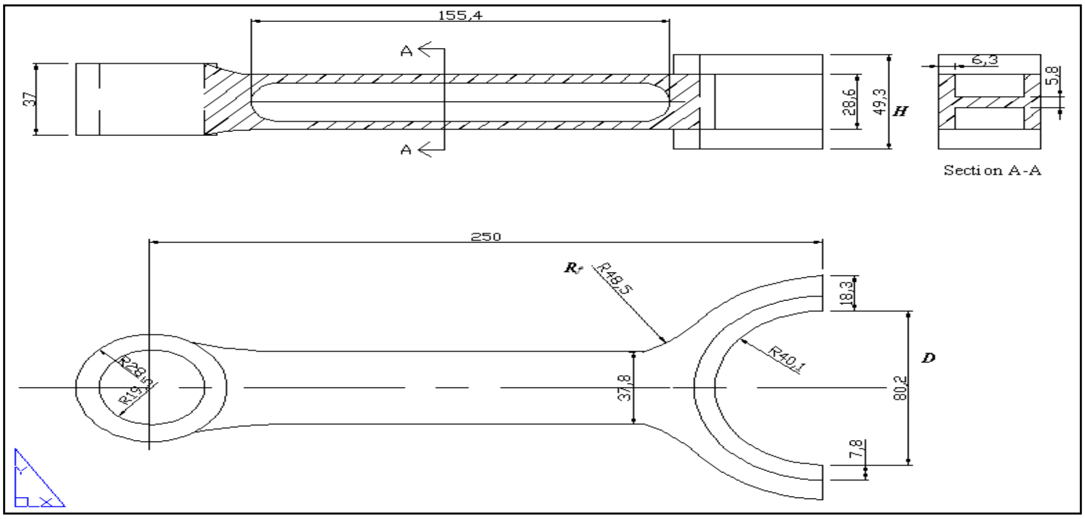

Figure 3.10: The H section connecting rod's geometry [9]

Using Pro-E Designing Software, a geometrical model of a H section connecting rod was created, as seen in Figure 11. The weight of the connecting rod for an H-section is 1705 grammes, according to a scale. Less than 1% of the weight of the component utilised by M. Omid et al [9] differs from the weight of the solid model used for Finite Element Analysis. This demonstrates how accurate the solid model is. The connecting rod with the H section has been modelled while maintaining a constant mass for both the H and I section. Using Pro-E Designing Software, a geometrical model of a H section connecting rod was created, as illustrated in Figure 20.

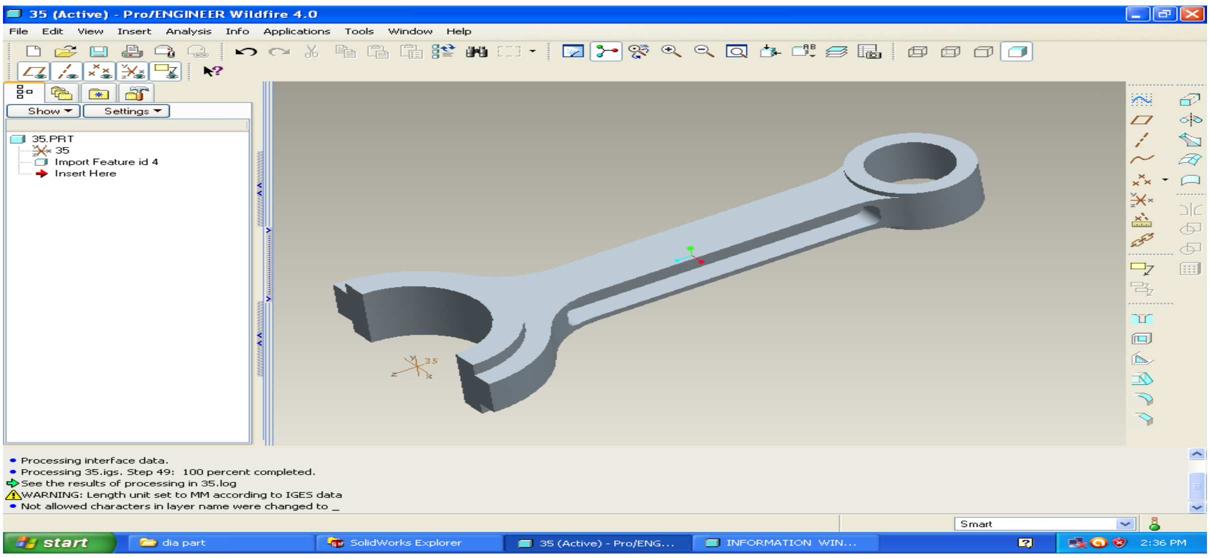

Figure 11. Geometrical Model of H Section Connecting Rod

Comparing the regions on each side of the axis of symmetry perpendicular to the connecting rod

International Journal for Academic Research and Development
ISSN 2582-7561(Online)© I J A R D
Vol. 3 Issue 4
2021

length and along the web, the degree of non-symmetry in the shank area of a H section connecting rod was around 5%. This loss of symmetry was a production variation that wasn't intentional. Since it is symmetric, the connecting rod has been modelled as such.

5.2 MESH GENERATION OF H SECTION

The FE analysis is performed using Pro/E and ANSYS workbench software. Using Pro/E software, a 3-D model of the H section connecting rod was created. An IGES file was then produced and imported into the ANSYS workbench programme. The connecting rods needed to be mesh next. Figure 12. depicts the H section mesh model. As seen in Figure 4, the 10-node tetragonal elements were employed. Tetragonal elements with a 2 mm element length were used to create the finite element mesh (2262 elements). This element was chosen in order to simplify the geometrical elements of a complex mechanical component and to help us obtain more accurate findings for the fatigue life calculation.

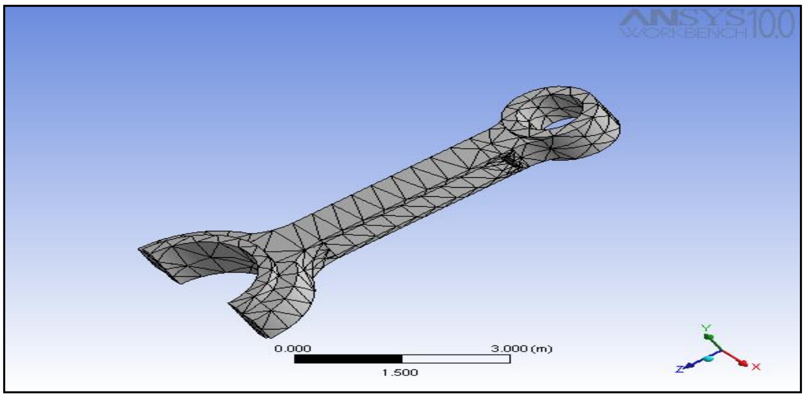

Figure 12. H section Connecting Rod Geometrical Model

5.3 Boundary Conditions of H Section

5.3.1 Loading

The big ends are considered to be under tensile tension, as shown in Figure 13, with a sinusoidal distributed loading throughout the contact surface area. Based on the findings of an experiment [10]. On the contact surface, the normal force is determined by:

Four finite element models were examined in this study. Both compressive and tensile loads underwent FEA. For each case, two cases were examined: one with the load applied at the other with the load applied at the piston pin end the crank end restrained and the crank end and the piston pin end restrained. The axial load in the analysis was 9500 N in both compression and tension.

Because the analysis is linear displacement, the stress, elastic, and strain in static analysis are proportional to the size of the load. Therefore, using a proportional scaling factor, the FEA results can be easily applied to different elastic load instances.

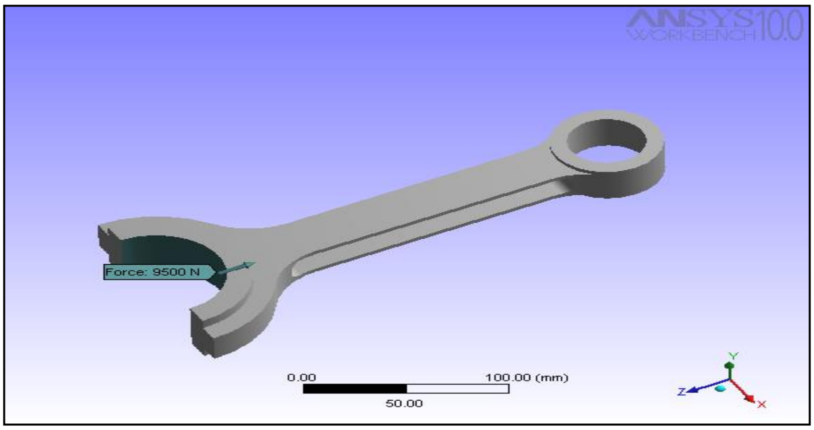

Figure 13. A sinusoidal distributed loading under tensile loading on H section connecting rod

5.3.2 Restraints

Four FEA models were solved, as was already described. A Finite Element Analysis model with a tensile load applied at the big end and a restrained small end is shown in Figure 14. Keep in mind that the piston pin's inner surface is completely constrained on one side (1800. Similar to this, 1200 of the contact surface area is completely restrained when the connecting rod is under an axial compressive load.

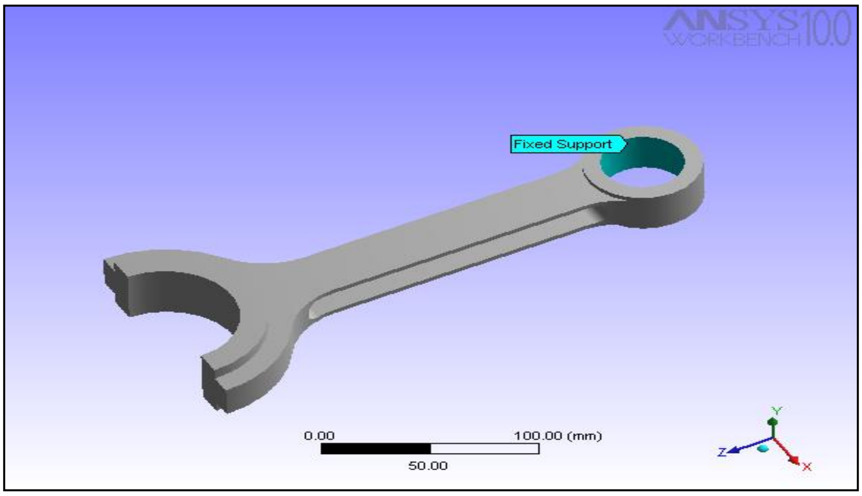

Figure 14. A model in which the small end of H section connecting rod is restrained

6. Fatigue analysis of H section connecting rod

The boundary conditions were first specified and applied a tension force. The tension force was then replaced by a compression force exactly of the same size and in the opposite direction, and the problem was once more resolved. The Von Misses stresses were activated throughout each stage of loading, and the crucial sites were identified.

Figure 15. depicts the stresses placed on different parts of an H-section connecting rod, and Figure 16. illustrates how cyclic loading causes the critical area to be seen at the stem of the rod close to the bearing or big end roundness.

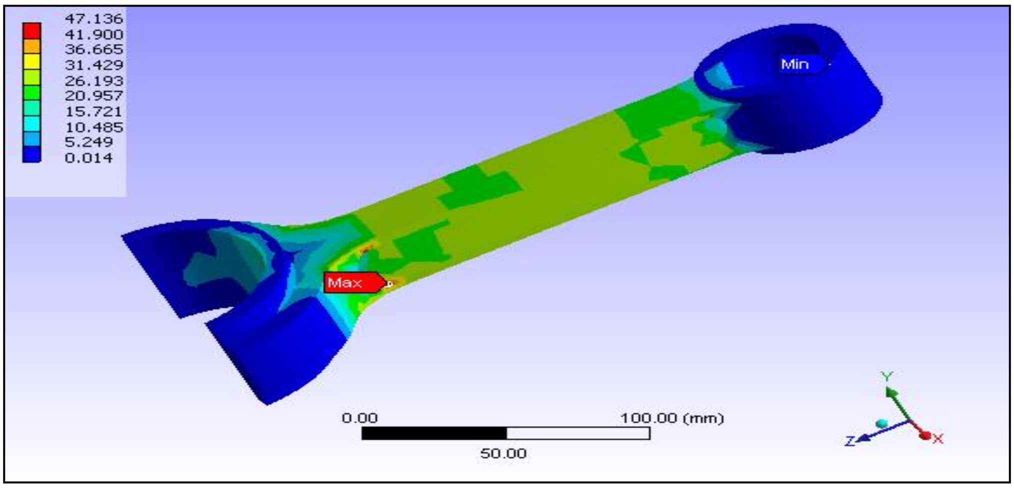

Figure 15. H section connection rod Fatigue Analysis

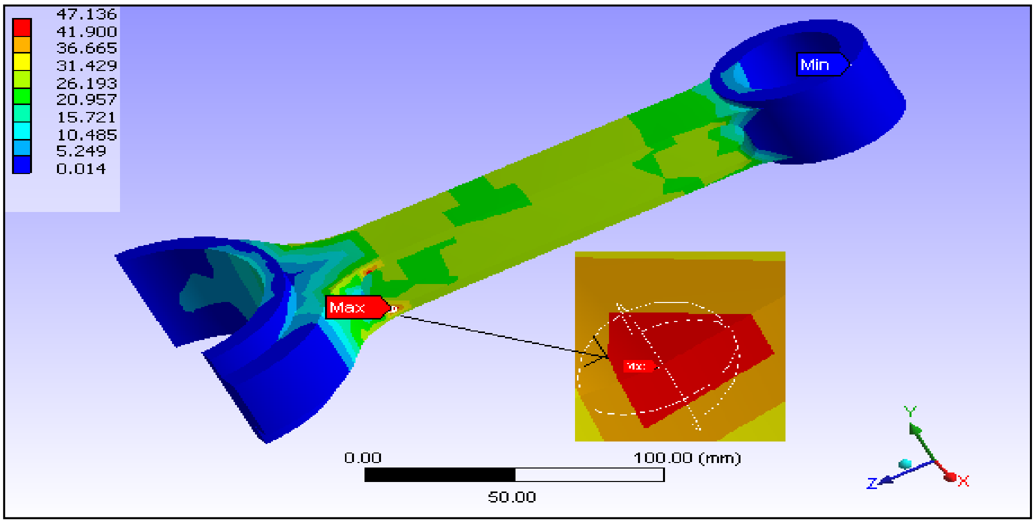

Figure 16. H section Connecting Rod Critical point

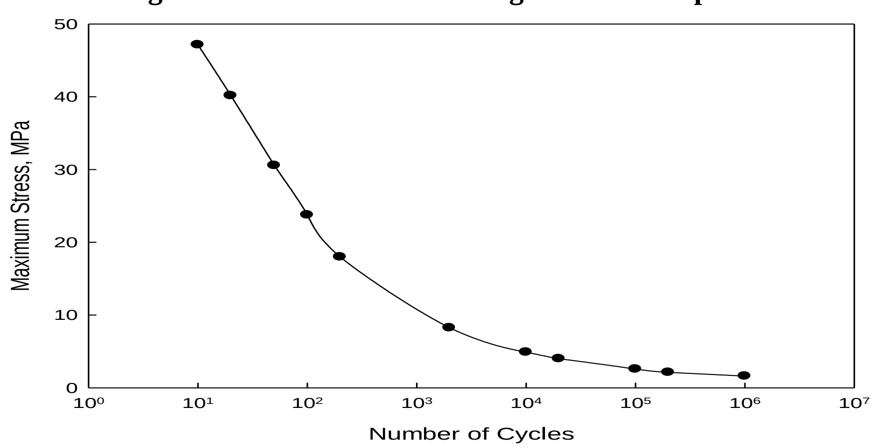

Figure 17. H section connecting rod subjected to a fatigue loading

These important spots were identified after which they were chosen as the areas to be investigated for fatigue. The S-N data gathered from the fatigue test of the particular alloy should import into the programme, filling in the blanks for the fatigue parameter as indicated in Figure 17. After applying 10^6 force cycles to the model, a partial consumption rate was obtained, showing the ratio of the applied force cycles to the permissible ones for each node. We can change specific geometrical dimensions while maintaining the same values for all other dimensions towards the big end in order to enhance and maximise the fatigue life of a connecting rod.

7. Validation of Finite Element Analysis

M.Omid et al. [9] determine the characteristics of the substance employed for linear elastic finite element analysis. The stresses in the shank region halfway down the connecting rod's length were compared under two scenarios of compressive load application in order to validate the Finite Element Analysis model. The big end was first subjected to a 9500 N evenly distributed load while being restrained at the tiny end.

In the current study, the connecting rod's FE analysis was performed using the ANSYS workbench software and validated against results obtained by Omid et al. [10] and displayed in Table 2.

Table 2. Validation of Finite Element Analysis of the connecting rod

Stresses	ANSYS Workbench	ANSYS 10
Maximum Stress in Compression (MPa)	23.33	24.00
Maximum Stress in Tension (MPa)	29.94	29.40

8. Results and Discussion

Near the connecting rod's big end, the I section connecting rod's maximum stress developed is 33.56 MPa. Near the connecting rod's big end, the H section connecting rod's maximum stress is 47.136 MPa. Increasing the number of forces cycles up to 10^6 constantly results in the calculation of the stresses corresponding to the critical points.

The remaining Figures of present work investigates the effects of critical dimensions, such as inner diameter, height of the big end and fillet radius near the big end , on the mass of the connecting rod and stresses generated at critical point. In the current study, it has been examined how fundamental crucial dimensions affect the fatigue life of connecting rods with I and H sections while maintaining the status quo for all other dimensions.

8.1 Results of I Section Connecting Rod

Figure 9 displays the S-N curve for an I section connecting rod based on the results of a fatigue test for the particular alloy, the parameters of which are listed in Table 1. Increasing the number of forces cycles up to 10^6 constantly results in the calculation of the stresses corresponding to the critical points. The basic critical dimensions of I section connecting rod as shown in Figure 18. are D=80.20mm, H= 49.30mm and Rf = 485mm.

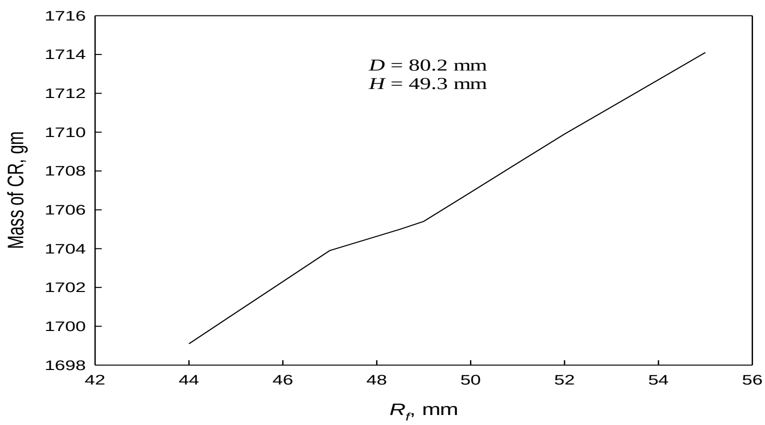

Figure 18. The effect of fillet radius near the big end on the mass of I section connecting rod

Keeping all other dimensions constant, Figure 18. illustrates the impact of the fillet radius near the big end on the mass of the connecting rod. It has been found that the connecting rod's mass and the fillet radius near the big end are proportional. The connecting rod's mass increases as the fillet radius does, and vice versa. Keeping all other dimensions equal, Figure 19. illustrates how the big end fillet radius affects the highest stresses produced at the critical point. The stress at the critical point continuously reduces as fillet radius or fillet roundness grows, then slightly rises after reaching a minimum value and stays constant as fillet radius increases further. It is discovered that the basic radius considered in the finite element analysis is best for the given design. The effect of stress concentration is more pronounced for lower fillet radius values, below the optimum value of, because of a reduction in the thickness of the big end at the critical point.

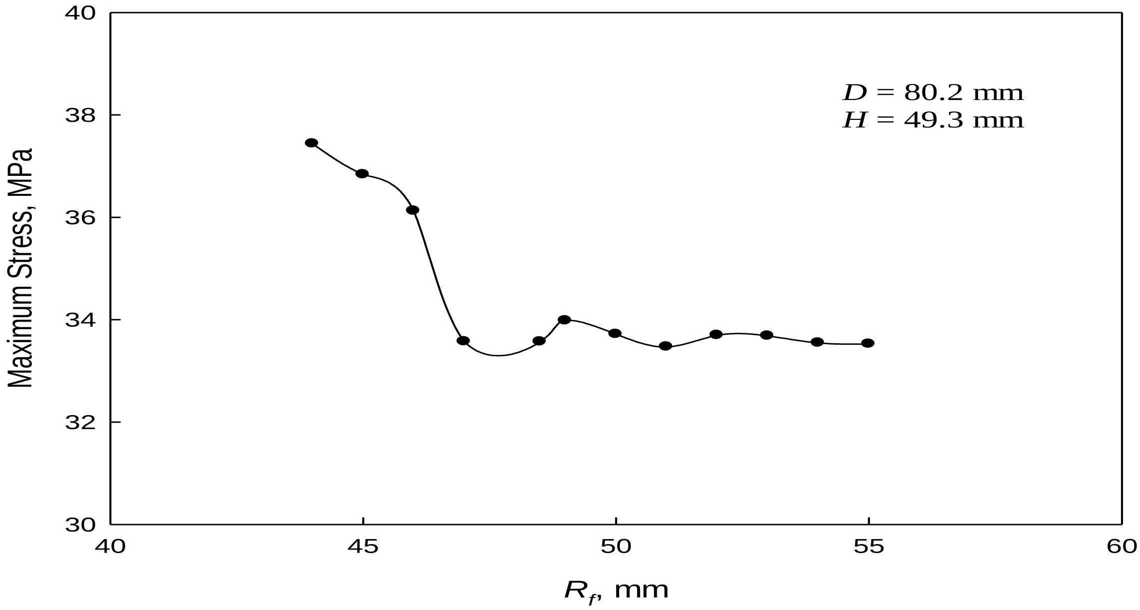

Figure 4.2: Maximum stresses produced at the critical point during cyclic loading and the impact of large end fillet radius

Keeping all other parameters constant, Figure 20. illustrates the impact of big end inner diameter (D) on connecting rod mass. It is discovered that the mass of the connecting rod is determined to be inversely proportional to the inner diameter of the big end and decreases with increase in inner diameter (D), as expected. The impact of huge end inner diameter on the highest stresses generated at the critical point under cyclic loading is depicted in Figure 21. For a fixed fillet radius, the effect of increasing inner diameter results in a minor reduction in the stresses generated at the critical point, and a sharp increase in stresses after reaching the lowest optimal value. The reason is that the section weakens very quickly when the thickness of the connecting rod at the critical point is smaller than the thickness of the remaining circular part of the big end.

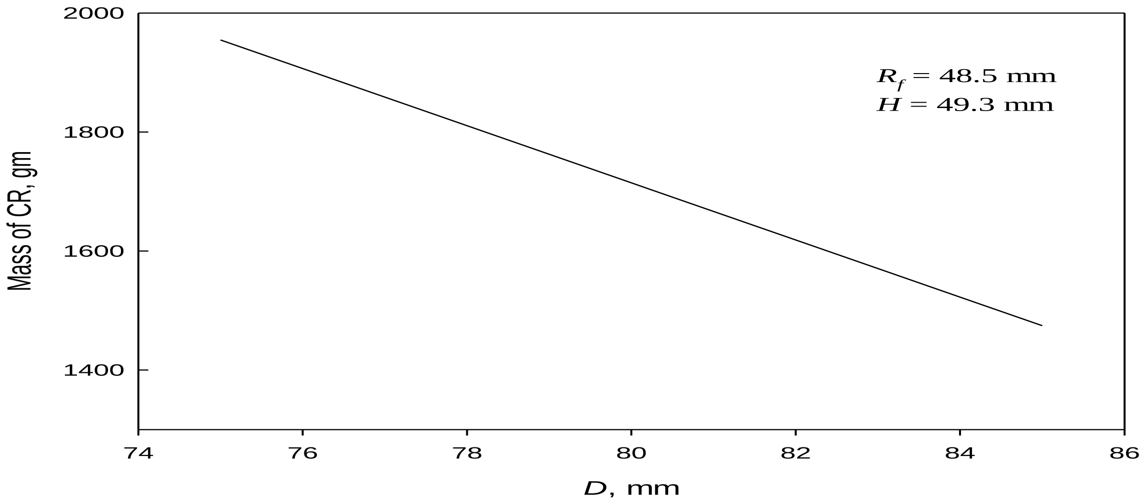

Figure 20. The effect of big end inner diameter (D) on the mass of I section connecting rod

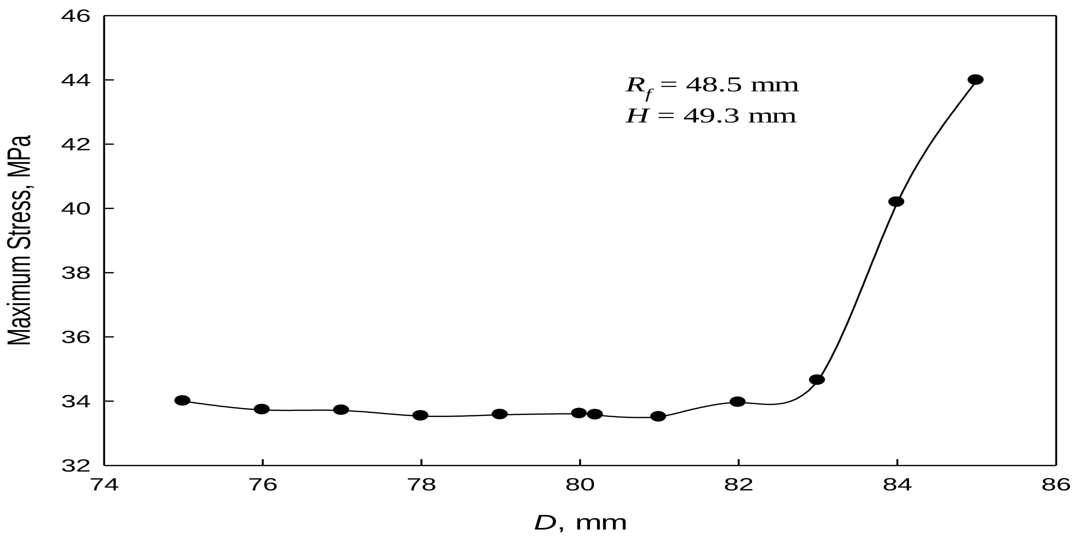

Figure 21. The effect of big end "D" on the $(\sigma_{\max})$ generated at the critical point under the cyclic loading

Keeping all other dimensions constant, Figure 22. illustrates the impact of the large end's height (H) on the mass of the connecting rod. It has been found that the connecting rod's mass and big end height are proportionate. The mass of the connecting rod similarly increases with height, and vice versa. Keeping all other dimensions constant, Figure 23. illustrates how the height (H) of the big end affects the highest stresses produced at the critical point. When the big end height is increased for a certain fillet radius, the stresses produced at the critical point are marginally increased, and after reaching the lowest optimal value, they are also significantly increased.

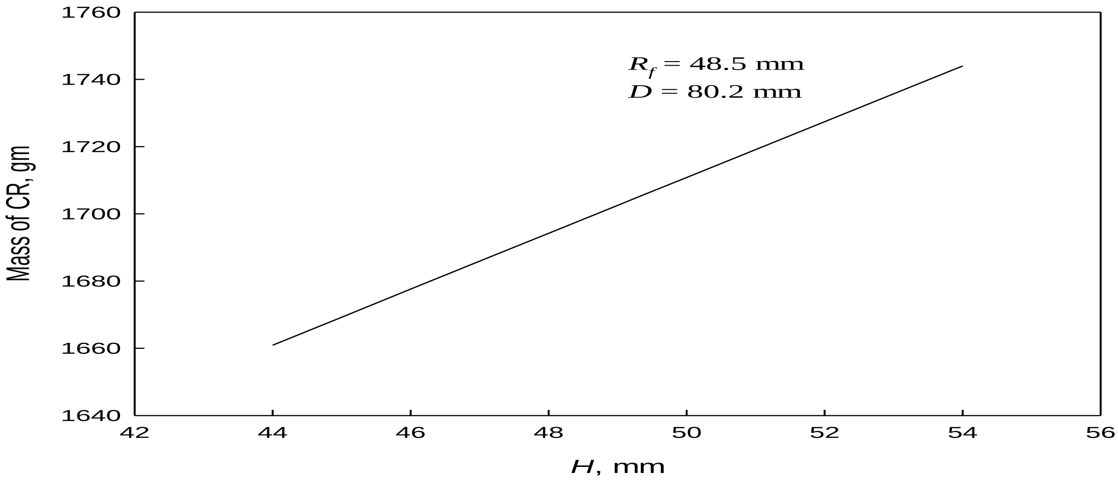

Figure 22. The effect of height (H) of the big end on the mass of I section connecting rod, keeping other dimensions' constant

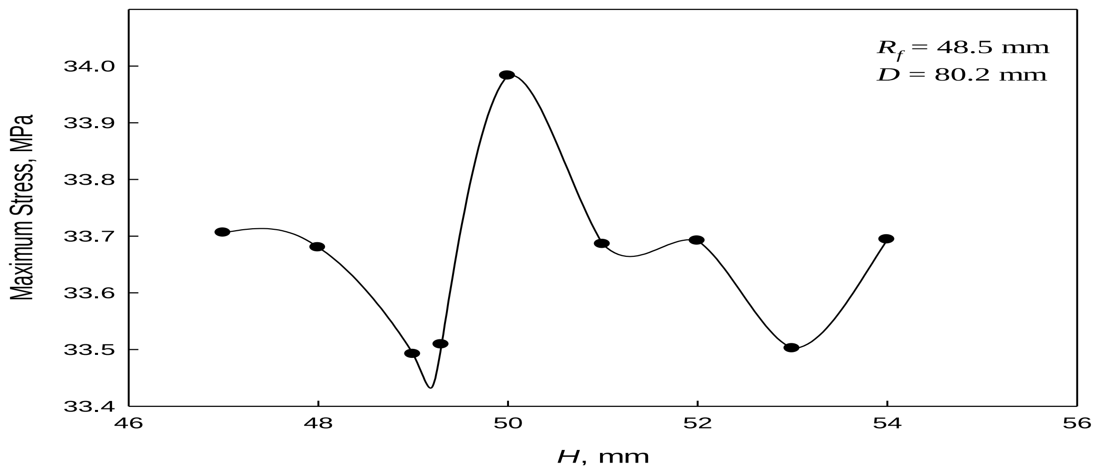

Figure 23. The effect of height (H) of big end on the $(\sigma_{\max})$ generated at the critical point, keeping other dimensions' constant

9. Conclusion

With the help of ANSIS software and the finite element analysis approach, it is possible to study various components from a variety of angles, including fatigue, which saves time and money. The way loadings were defined had a positive impact on the outcomes. They should therefore closely match the actual circumstances. Because the fatigue analysis requires for some static analysis, it is necessary to specify the boundary conditions that are most realistic. In this study, we get to the conclusion that the critical node close to the big end fillet is optimal for the three critical dimensions. The manufacturing sector is the optimum match for this connecting rod design. In the future, I section connecting rods and H section rotating parts will be compared.

References

[1] *Afzal A. and Fatemi A. (2003). A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods. SAE International.*

[2] *Whittaker, D., 2001. The competition for automotive connecting rod markets. J. Metal Powder Report, 56: 32-37.*

[3] *Repgen B. (2005). Optimized Connecting Rods to Enable Higher Engine Performance and Cost Reduction, SAE Technical Paper 980882, pp. 1-5.*

[4] *Augugliaro, G. and Biancolini, M. E. (2000). Optimization of Fatigue Performance of Titanium Connecting*

Rod.
[5] Biancolini, M.E., C. Brutti, E. PennestrÃ and P.P. Valentini, 2003. Dynamic, mechanical efficiency and fatigue analysis of the double cardan homokinetic joint. J. Vehicle Design, 33: 47-65.
[6] Rabb, R., 1996. Fatigue failure of a connecting rod. J. Eng. Failure Anal., 3: 13-28.
[7] Beretta, S., A. Blarasin, M. Endo, T. Giunti and Y. Murakami, 1997. Defect tolerant design of automotive components. J. Fatigue, 19: 319-333.
[8] Lo, S.H.R. and Bevan, A. (2002). Fatigue Analysis of a Plate-with-a-hole Specimen and a Truck Exhaust Bracket Using Computer-Based Approach, Int. j. Eng. Sim. (IJES). 4(2).
[9] M. Omid, S. S. Mohtasebi, S.A. Mireei and F. Mahimoodi (2008) Fatigue Analysis of connecting rod of U650 Tractor in the Finite Element Code ANSYS.
[10] Whittaker, D., 2001. The competition for automotive connecting rod markets. J. Metal Powder Report, 56: 32-37.
[11] Richard C.Rise, Brian N.Leis, Drew V.Nelson, Henry D.Berns, Dan Lingenfelser, M.R. Mitchell, 1988. Fatigue Design Handbook, Society of Automotive Engineers, Inc.