ISSN: 0974-2115
www.jchps.com Journal of Chemical and Pharmaceutical Sciences

Design optimization of connecting rod in heavy commercial vehicles

V.M. Illayaraja Muthaiyaa[1*], T. Parameshwaran Pillai[2], G.Elatharasan[1]
[1]Department of Mechanical Engineering, University College of Engineering, Pattukkottai, Tamil Nadu
[2]Department of Mechanical Engineering, University College of Engineering, Thiruchirappalli, Tamil Nadu
*Corresponding author: Email:susheeraj@gmail.com; Mobile: +91 - 9894957585

ABSTRACT

The objective of this project is to design and reduce the weight of an automotive connecting rod. As a startup strategy to achieve weight reduction and to lower the manufacturing cost, Austempered Ductile Iron (ADI) material is chosen in place of forged steel. In the first part of the thesis, the loads acting on the connecting rod were calculated for the maximum speed of the crankshaft. Static analyses were performed on the existing connecting rod under tensile and compressive loads. Design space was finalized based on the results of the static analysis. In the next part of the study, several optimization iterations were performed with different parameters. Optimal design was finalized based on stress, displacement and natural frequency. Fatigue life was estimated for both existing and the optimized connecting rod.

KEY WORDS: Optimization, automotive, vehicles.

1. INTRODUCTION

Connecting rod is an integral part of IC engines that is used to connect piston and crankshaft. It is a critical component that should be designed for safe operation. As today's world is going towards cost reduction of parts, it is logical that any reduction in weight and manufacturing cost will be of great impact.

1.1. Finite element analysis of connecting rod:

1.1.1. Preprocessing phase: First step is importing the CAD geometry and cleaning up the geometry. Geometry cleanup includes removing unnecessary features, repairing the surfaces that are not connected properly, surfaces that have free edges etc.

a) Discretize the solution domain into finite elements, that is to subdivide the component into nodes and elements.
b) Establishing element level equilibrium equations.
c) Assembling the equations to form global equation.
d) Boundary conditions are applied for different load cases.

1.1.2. Solution phase: Solve a set of linear or nonlinear algebraic equation simultaneously to obtain nodal results, such as displacement and stress values at different nodes.

2. METHODOLOGY

2.1. Finite element model of connecting rod geometric cleanup: Geometric cleanup was performed on the CAD model of connecting rod. The free surface edges were joined with minimum tolerance of 0.01mm. Intricate edges of the surfaces were removed for simplification of mesh without compromising on the accuracy and the original geometry of the connecting rod. Washer split (a circular edge created near the hole for proper meshing along the hole) was created for bolt holes in connecting rod cap for even distribution of distribution of loads. The geometry was checked for a) Free edges, b) Duplicate surfaces, c) Small fillets, d) Intersection of parts, e) Small fillets.

Very small features increases intricacy which makes meshing difficult and also increases the time for convergence of solution.

2.2. Meshing: The element length was assigned as 2 mm considering the minimum surface length. Connecting rod is a component which has complex geometry and uneven shape. So the model cannot be represented with either 2D shell elements or 1D beam elements and hence a 3-dimensional model was used. To reduce the complexity 3-D tetra mesh was used for connecting rod body and cap. It is a well-known fact that tetrahedrons are constant stress elements and usage of this would result in a highly stiff behaviour. Hexahedrons are better suited for structural analyses. But because of the connecting rod's complex shape, it is tedious and time consuming to build an FE model of the connecting rod with hexahedrons. Hence tetrahedrons were used. The problem of constant stress was taken care by the usage of quadratic tetrahedrons (with mid side nodes), whose stress behaviour is linear. To build a 3-D meshed model, the model was meshed with 2D elements first to create an enclosed volume, which was then filled with tetras. The bolts were modeled using 1-D beam element to reduce complexity. The shank portion was modeled using beam element and was connected to the nodes of the connecting rod using Rigid RBE2 elements. Bush pressure was considered in small end or piston end of the connecting rod. For modeling bush, 2-D shell elements were used. Mass of the connecting rod is 1.72Kg.

2.3. Element Quality: Accuracy of the results is directly proportional to element quality. Hence the quality criteria were followed strictly. The following measures are considered for mesh quality:

- Mesh flow lines are maintained parallel to the edges for appropriate expression of the shape to capture the boundaries of surfaces.

- Washer was modeled are bolt locations for even distribution of loads.
- Even numbers of elements are maintained around holes.
- Right angled triangles are used along closed curves for better expression of shape.

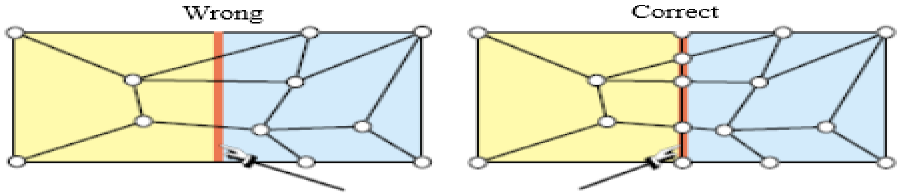

Figure.1.Shows the mesh flow along the surface. It is always better to have mesh boundaries bounding the surface i.e. mesh pattern must be followed

Along with these measures, the quality criteria of individual elements were also maintained. Though the quality of the 3D elements can be checked, it cannot be edited as per the quality criteria. So it is better to have a good quality 2D mesh before converting it to 3D. As mentioned earlier, triangular elements were used for 2D meshing. The ideal shape of triangular element is an equilateral triangle. Elements are maintained to have following quality criteria.

2.4. Warp angle: Warp angle is out of plane angle. It is defined as the angle between normals to two planes formed by splitting the quad element along diagonals. Maximum angle out of the two possibilities is warp angle. Warp angle is not applicable for triangular elements.

2.5. Aspect: It is defined as the ratio of maximum element edge length to minimum element edge length.

2.6. Skew: Skew for triangular element is found as 90-α, that is the minimum angle between the lines from each node to the opposing mid-side and the two adjacent mid-sides at each node in the element.

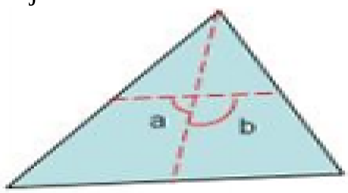

Fig.2. Skew of Tria

2.7. Chord deviation; Chord deviation helps in determining how well curvatures have been modeled. It is defined as distance between mid-node of element edge to curved surface.

Table.1. Shows the ideal and acceptable values for element quality for the solver used in the project

Quality criteria	Ideal value	Acceptable value
Warping	0°	< 5°
Aspect ratio	1	< 5
Skew	0	< 60°
Jacobian	1	>.7
Distortion	1	.7
Chord deviation	0	.1
Minimum angle	60°	>20°
Maximum angle	60°	<120°

Minimum and Maximum angle – An ideal triangular element should be an equilateral triangle. Table shows the ideal and acceptable values for element quality for the solver used in this project.

2.8. To rectify the elements that fail due to the quality criteria following methods are used

2.8.1. Manual Adjustment: This is done by translating the nodes manually or re meshing in the poor mesh region. This method consumes lot of time.

2.8.2. Drag node: All these quality criteria listed above are set in the quality index. Then nodes of failing elements are dragged which shows the effect on elements instantly.

2.8.3. Free edges: Any single triangular element will have three free edges. Free edge is an indication of unconnected nodes. The connectivity of nodes was checked, after checking the quality of elements. Tetra mesh cannot be created without connectivity of nodes, because for tetra mesh generation, an enclosed volume of elements is needed. Sometimes, connectivity issues might result in stiffness matrix singularities and the solution might not even converge. Forces are transferred through nodes, so when nodes are not connected forces will not be transferred. This leads to inaccurate results.

2.8.4. Duplicate elements: During operations like reflect, translate and while deleting and creating new meshes, duplicate elements may be created. These duplicate elements do not cause any error during the analysis but increases stiffness of the model and results in lesser displacement and stress. The mesh was checked for duplicate elements and deleted if any.

2.8.5. Duplicate nodes: Duplicate nodes are sometimes created when nodes are not connected but the distance between them is very small. This may also result because of operations such as copy, translate, and reflect etc. Some software will not solve the problem if duplicate node exists.

2.8.6. Material properties: The analysis was performed with an assumption that the connecting rod operates within the elastic limit, so linear elastic material model was used with isotropic properties. MAT1 property card which is generally used for structural analysis was used. For existing connecting rod, forged steel is used and for optimized connecting rod, ADI is used.

Table 2. Materials used in Fe Analysis

Model	Material	Young's Modulus N/mm^2	Tensile strength N/mm^2	Yield strength N/mm^2	Density kg/m^3
Existing	C70S6	210000	850	545	7850
Optimized	ADI	170000	950	750	7100

2.9. Loads and boundary conditions: As discussed, by experimental studies, the compressive load was found to be distributed over 120° of the contact surface area and the tensile load distributed over 180° of the contact surface area. There are totally 4 load cases considered for the static analysis:

a) Constraining the crank pin end for all degrees of freedom and applying a compressive force distributed over 120º at piston pin end.

b) Constraining the piston pin end for all degrees of freedom and applying a compressive load distributed over 120º at crank pin end.

c) Constraining the crank end for all degrees of freedom and applying a tensile load distributed over 180º at piston pin end.

d) Constraining the piston pin end for all degrees of freedom and applying a tensile load distributed over 180º at crank pin end. Bush pressure was considered in all the load cases.

As explained earlier, in the compressive load case, there are two subcases. This is because load at crank end and piston end are different. So the effect of compressive load at crank end and at piston end can be observed with the help of sub cases. In compressive load, load at piston end is more than the load at crank end. The compressive load is distributed over 120° of the surface area.

In the tensile load case, the force due to inertia plays a main role. Though tensile load is less when compared to compressive load, stresses due to tensile load at the connecting rod cap is critical. This is because the connecting rod cap and body are fastened with the help of bolts. The tensile load at crank end is more than that in the piston end because; the mass at the crank end is more than the mass at the piston end. The tensile load is distributed over 180° of the surface area.

2.10. Fatigue life estimation: Fatigue is a process which causes a premature failure or damage of a component subjected to repeated loading. There are basically two types of fatigue failures: High cycle fatigue and low cycle fatigue. Connecting rod mainly fails due to fatigue and hence fatigue life was estimated for existing connecting rod design and for optimized design to ensure that it passes required minimum number of loading cycles.

2.11. Fatigue life for existing design: Material used for existing connecting rod is forged steel.

Ultimate tensile strength σ_u = 850 Mpa, Young's modulus, E= 210 Gpa, Fatigue ductility coefficient factor, ε_f = 2.52.

2.12. Fatigue life for optimized design: Material used for optimized design is ADI

Ultimate tensile strength σ_u = 950 Mpa, Young's modulus, E= 170 Gpa, Fatigue ductility coefficient factor, ε_f= .2127

3. RESULTS AND DISCUSSION

3.1. Static analysis of existing model: Linear static analysis was performed on the existing connecting rod to observe the effect of compressive and tensile load.

3.2. Compressive load: The displacement does not exceed .18 mm during compressive load. Stresses are more at the contact surface of the gudgeon pin and small end. Stresses are more at the shank portion of the connecting rod.

3.3. Tensile Load: In tensile load the displacement was more at the connecting rod cap. This is because the cap and connecting rod body is connecting using bolt. Displacement due to crank end constrained which is less when compared to piston end constrained. This is because the inertia force at big end is higher than the inertia force at small end. Though stresses are quite high, it only localized stresses which will not have any significance.

Stress due to compressive force is higher than stress due to tensile force. Though tensile force is less, it is important because stresses are more at the connecting rod cap near the bolt.

3.4. Modal analysis of existing design: Modal analysis was done on connecting rod to find the natural frequency and the mode shapes of connecting rod. It was found that the first mode shape is bending about x-axis. The first or fundamental natural frequency of the existing model of connecting rod was found to be 502.5 Hz. The natural frequencies of existing design of connecting rod for the first 3 modes are

Table.3. Frequencies of first 3 mode shapes

1st mode	502.8 Hz	Bending about x-axis
2nd mode	960.5 Hz	Bending about z-axis
3rd mode	1639 Hz	Torsion along y-axis

3.5. Optimization:

3.5.1. Displacement: The three optimized designs are compared with the existing. It is clear from the analysis results that the model I deforms more when compared to the models II and III. The displacement plots shown are that of the case where the crank end is constrained and the load is applied at the piston end.

3.5.2. Stress distribution: Stress distribution in the various designs under compressive load. Even though the stresses are high in the optimized designs, compared to the existing, they are much below the yield strength of ADI. The existing design is made of forged steel which has yield strength of 545 Mpa, while the optimized design is going to be made out of ADI, which has yield strength of 750 Mpa. This allows having relaxation in the maximum stresses by removing material. Among the three optimized designs, the maximum stress in models II and III are lesser compared to the model I.

3.5.3. Modal analysis: Modal analysis was performed to compare the effect of material removal on the natural frequencies. Natural frequency compares the specific stiffness of the components. Higher the natural frequency, higher the specific stiffness. Here specific stiffness is defined as k/m (stiffness to mass ratio). Existing Design: 502.8 Hz While, Design Model 1: 438.3 Hz; Design Model 2:526.8 Hz; Design Model 3: 485.1 Hz.

Table compares the different parameters such as mass, yield strength, mass reduction, etc. of the various optimization iterations and the existing design.

Table.4. Comparison of existing model to optimized model.

Properties	Existing connecting rod	Optimized connecting rod		
		Design I	**Design II**	**Design III**
Mass (grams) for forged steel	1721	1623	1650	1651
Maas (grams) for ADI	1557	1468	1493	1494
Maximum Displacement (mm)	0.18	0.22	0.20	0.21
Maximum stress (Mpa)	280	545.1	406.1	450.1
Yield Strength (Mpa)	545	750	750	750
Tensile Strength (Mpa)	790	950	950	950
Mass reduction in %	-	5.687%	4.09%	4.07%
Natural frequency (Hz)	502.8	438.3	526.8	485.1

Among the three optimized designs, design I has higher percentage of reduction in weight (5.7%) when compared to designs II and III. But the displacement and the maximum stress is higher in design I. The natural frequency is also less compared to the existing one. Though stress distribution is similar in design II and design III, design III has a higher maximum stress. The percentage of reduction in mass in designs II and III is almost the same. But on comparing the fundamental natural frequency, design II seems to be a better option. Hence design II is selected for further studies.

3.5.4. Fatigue life estimation comparison: Strain for each element was found by assuming life as 10^7 and the critical element was figured out by seeing the maximum strain. Then the actual strain found from the finite element analysis was taken and then the maximum fatigue life was found out. By assuming different life ε-N curve is drawn for the existing model. The fatigue analysis results are based on the material properties which are interpreted with the connecting rod models by taking individual elements from static analysis results of connecting rod. The fatigue analysis revealed that crack initiation occurs at connecting rod cap.

Table.5. Fatigue life estimation of existing model

Fatigue Life Estimation of existing connecting rod		
σ_u	850	Mpa
E	210000	Mpa
b	-0.09	
c	-0.56	
ε_f	0.09531	
$2N_f$	10^7	cycles
σ_f’	1336.27	Mpa
ε_f’	0.2524	
σ_0(critical element)	-121.348	Mpa
$\Delta\varepsilon$ (critical element)	0.00085	
$\Delta\varepsilon$ (from FEA result)	0.000848	
Maximum N_f	$1.5*10^{13}$	cycles

3.5.5. Fatigue life estimation for optimized model: Material used for optimized model is ADI. The critical element was found in similar fashion. Then maximum fatigue life was found out to be the same as the existing model because the crack initiation occurs at connecting rod cap. Crack initiation was found out by finding the element that is strained more. No change is done in connecting rod cap as it is in non-design space. Thus the fatigue crack initiation occurs at the same area as existing model. Though there is no change in fatigue life due to change in design, there will be some changes due to change in material. Also, the fatigue life is more than the required number of cycles. Thus the maximum fatigue life is same for both existing and optimized model.

Table.6. Fatigue life estimation of optimized model

Fatigue Life Estimation of existing connecting rod		
σ_u	950	Mpa
E	170000	Mpa
b	-0.09	
c	-0.56	
ε_f	0.09531	
$2N_f$	10^7	cycles
σ_f’	1414.71	Mpa
ε_f’	0.2127	
σ_0(critical element)	-126	Mpa
$\Delta\varepsilon$ (critical element)	0.00405	
$\Delta\varepsilon$ (from FEA result)	0.001	
Maximum N_f	$1.5*10^{13}$	cycles

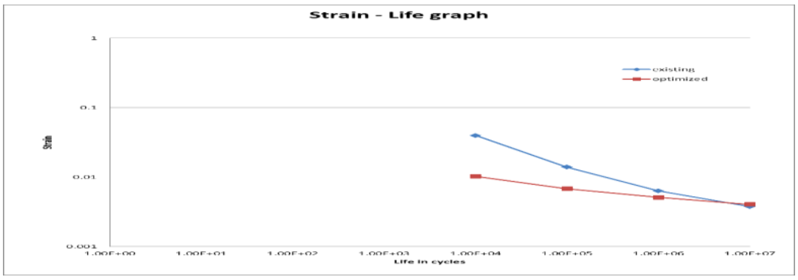

Figure.3. Strain – Life Graph.

This graph shows the comparison of ε-N curve of existing and optimized connecting rod. Both the curves converge between 10^6 and 10^7 cycles. Life of optimized model is more after converging which shows that the optimized model is good at high cycle fatigue. This is because the Austempered Ductile Iron has very good fatigue life for high cycles. This fatigue life estimation is just based on materials, and further improvement on fatigue strength shall be done in manufacturing process.

4. SUMMARY AND CONCLUSION

This project investigated the scope for weight reduction of connecting rod by topology optimization. The connecting rod considered for investigation belonged to a six cylinder engine. The loads acting on the connecting rod at big end and small end were computed using a rigid body dynamics software and were validated using analytical calculations. Linear static analysis was then performed on the connecting rod to predict the structural behaviour. Weight optimal design of the connecting rod was achieved based on the topological optimization trials. Finally, the fatigue life for existing and the optimized connecting rods were estimated.

The conclusions that are drawn from this study are listed below:

- Stress due to tensile load is significant though it is less when compared to compressive stress. The tensile stress was found to be more at critical regions like connecting rod cap, regions near the bolt etc.
- Modal analysis was done to find the natural frequency of the connecting rod which is also an important parameter to be checked.
- Critical areas or high stress areas of connecting rod are found out with the help of the static analysis results. This is very important while performing optimization, to decide which region to remove and which regions to retain.

Weight reduction was achieved with the help of topology optimization and by changing the connecting rod material from forged steel to Austempered Ductile Iron. While performing optimization to reduce weight, the static strength, fatigue strength and the natural frequency were maintained as it is in the existing one. To maintain the interchangeability, the constrained space was marked as non-design space. The optimized design was achieved through the following steps:

- The design variable was defined with manufacturing constraints. Many objectives like minimize volume fraction, minimize compliance and minimize displacement were used to explore various ways to obtain the optimized models.
- In the optimization process, the regions were retained or removed considering the critical areas and the stress distribution of connecting rod.
- Among the several optimized models, three models were selected based on the manufacturing feasibility, for comparison. The best out of these three models was finalized on the basis of stress, displacement, natural frequency and also the manufacturability, that is, with less intricate shape. The optimized model is 4.1% lighter than the present connecting rod. The volume of design space comprised of 22.81% of the whole connecting rod. When the ADI metal was substituted to present forged steel then 13.24% weight reduction can be achieved. Fatigue life estimation was done to compare the fatigue life with the existing model of connecting rod. The optimized model was found to be having required number of life cycles.

REFERENCES

Alex Antoc, Advanced Connecting Rod Design for Mass Optimization, SAE world congress, 2010.

Anton Olason, Methodology for Topology and Shape Optimization in the Design Process, Master’s Thesis, 2010.

Asadi M.R, Optimization of connecting rod of MF-285 tractor, Journal of Agricultural Technology, 2010.

Danielle Visser, A Comparison of Manufacturing Technologies in the Connecting Rod Industry, Thesis, 2008.

Tony George Thomas, Design of connecting rod for heavy duty applications produced by different processes for enhanced fatigue life. SAS Tech Journal, 10 (1), 2011.