Feasibility Analysis and Structural evaluation of Connecting rod.

Balasaheb Vikhe[1], Sagar Walhekar[2]

[1]Balasaheb Vikhe, Dept. of Mechanical Engineering, SVIT Nashik, Maharashtra, India.
[2]Sagar Walhekar, Dept. of Mechanical Engineering, SVIT Nashik, Maharashtra, India.

***

Abstract - This paper contain to check feasibility of process changed from forging to casting and Structural evaluation of connecting rod with Aluminium alloy(Al6063-T6) material. The main objective finding out effective design of connecting rod with minimum cost and weight. Conventionally material used for connecting rod is stainless steel through the forging process, as this method provides low productivity and higher production cost. The 3D model is prepared by using Pro-E creo4.0 and discretization is prepared by using Hypermesh while FEM is solved by using Optistruct Hypermesh13.0.

***Key Words*—Connecting rod, CAD, FEA, Static analysis, Modal Analysis**

1. INTRODUCTION

The Connecting rod are used generally used in all IC engines acting as an integral part between the piston and crankshaft. It is transfers motion from piston to crankshaft and convert the piston linear motion to crankshaft rotary motion. While connecting rod small end is connected to piston and bigger end is connected to the crankshaft.

Stainless steel Connecting rods generally manufacturing by Forging process. Disadvantages of using steel is that the material is extremely heavy, Costly, manufacturing process time consuming, higher production cost which consumes more power.

There are two forces acting on connecting rod are buckling load due to gas pressure and lateral bending due to inertia forces. Connecting rod must be withstand a cyclic loading during high compressive loads due to combustion and high tensile loads due to inertia.

A connecting rod can be of two types H-beam or I-beam or a combination of both depending on application.

2. METHODOLOGY

The objectives involved are:-

- 2.1 CAD Modeling
- 2.2 Finite Element Meshing
- 2.3 Boundary Conditions

2.1 CAD Modeling

The Fig.1 shows representation of Connecting rod. The CAD Model of I section connecting rod specification is Length-100mm Piston end dia-14mm, Crankshaft end dia- 20mm and thickness 10mm.

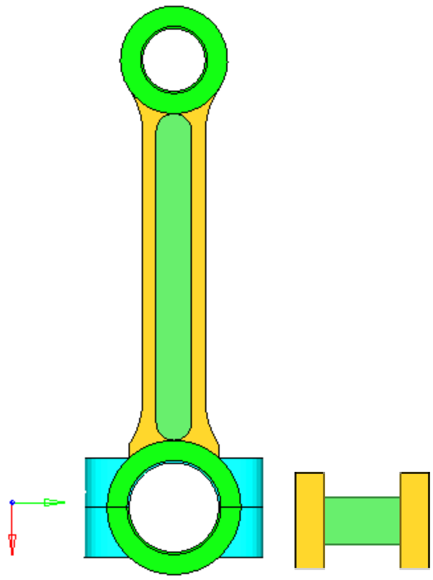

Fig.-1: Schematic Diagram of Connecting rod

2.2 Finite Element Meshing

The cad data in .stp format is imported in Hypermesh for the preparation of FE model. Then geometry cleanup was done by using options like 'geom. Cleanup' and 'defeature' to modify the geometry data and prepapre it for meshing operation. Mesh model is prepared by using Hypermesh 13.0. 8-node Hex 3D solid elements are used to model of Connecting rod.

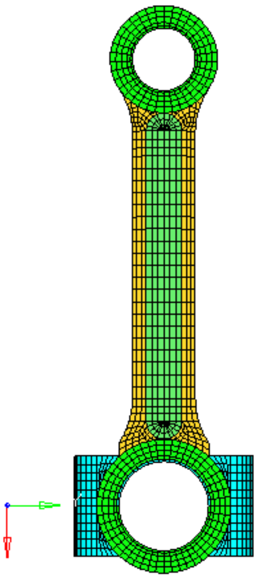

Fig.-2: FEM Model

The element size selected for meshing is 2mm. Connecting rod model is meshed with about 10713 nodes 8176 elements.

2.3 Loading Conditions

Connecting rod small end is connected to piston and bigger end is connected to the crankshaft. There are two forces acting on connecting rod are buckling load due to gas pressure and lateral bending due to inertia forces. Connecting rod must be withstand a cyclic loading during high compressive loads due to combustion and high tensile loads due to inertia.

The following loads on Connecting rod-:

1) Tensile loading

2)Compressive loading

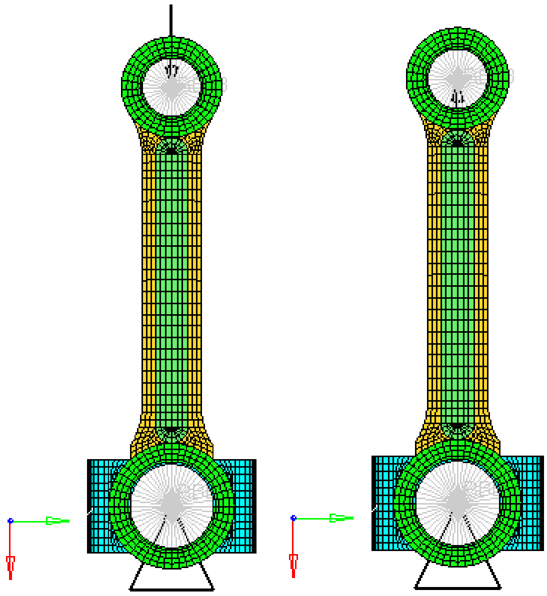

Fig.-3: Compressive loading **Fig.-4**: Tensile loading

3. RESULTS AND DISCUSSION

All machine component analysis, a component must be designed such that the stresses observing during operation will not exceed material limits. The material limits are determined by material properties and some known deformation theories. Analysis has to conclude whether the component is safe or fail comparing the max stress value with yield or ultimate stress.

FEA analysis is to find out the total amount of stresses and displacement, Modal Natural frequencies, Mode shapes of Connecting rod.

3.1 Static Analysis Results

Non-Linear Static analysis used to determine the displacements, stresses, strains and forces in structures or components cause by static loads. The solver used for analysis Optistruct Hypermesh.

3.1.1 Tensile loading

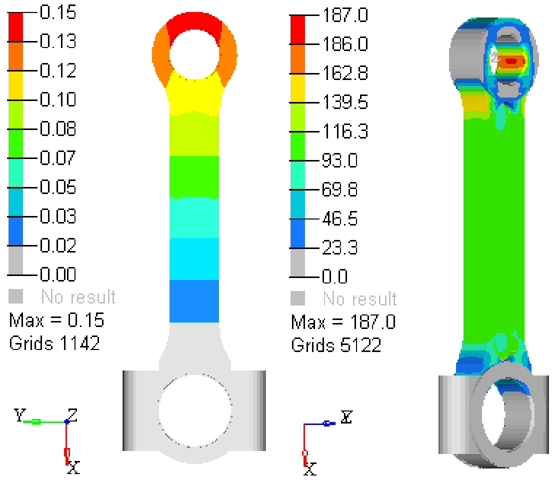

Fig.-5: Tensile loading Contour plot

Results:-

1. Maximum Displacement = 0.15mm.
2. Max. Principal stress = 187MPa.

3.1.2 Compressive loading

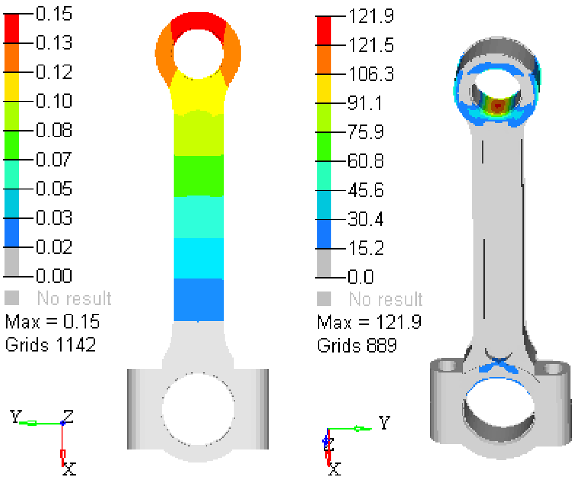

Fig.-6: Compressive loading Contour plot

Results:-

1. Maximum Displacement = 0.15mm.
2. Max. Principal stress = 122MPa.

4. CONCLUSIONS

The Max. Principal stresses observed on Connecting rod during Tensile and Compressive loading is less than the yield strength of material hence, connecting rod design is safe against load.

REFERENCES

[1] Nitin S Gokhale. “Practical Finite Element Analysis.” (2008).

[2] V. B. Bhandari. “ Design of Machine Element,”