International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 2, April 2013

Research Paper

STRESS ANALYSIS OF ORTHOTROPIC AND ISOTROPIC CONNECTING ROD USING FINITE ELEMENT METHOD

Ramanpreet Singh[1]*

*Corresponding Author: **Ramanpreet Singh,** ✉ ramanpreet.gurdutta@hotmail.com*

In this study simulation is conducted on a model of connecting rod of a single cylinder four stroke engine. The main objective of this paper is to develop a new insight for the use of composite material in connecting rods. Finite element analysis was done to compare the conventional isotropic material and the orthotropic Composite Material. Modeling of connecting rod was done using software CATIA V5 and for stress analysis it was imported to MSC. PATRAN. Linear static analysis was carried out for both isotropic material and orthotropic composite material with mesh TET4 to obtain the stress results. Comparison of both the material was done, keeping the boundary conditions "samefor" both materials. For future research, the same analysis can be done with the MESH TET10 and the same can be compared to obtain varied results.

Keywords: Connecting rod, Finite Element Analysis (FEA), Orthotropic material, Isotropic material, Patran analysis, CATIA V5 modeling

INTRODUCTION

Connecting rod also known as conrod is used to connect the piston to crankshaft. It forms a simple mechanism that converts linear motion into rotary motion. These rods are subjected to the alternating loads of order 10^8 to 10^9 cycles. These alternating cycles are of high compressive and high tensile loads due to gas pressure and inertia respectively. The maximum stress occurs at the small end of the bearing due to piston thrust. This is the reason connecting rods are designed as struts.

Pal *et al.* (2012) have done finite element analysis for tensile and compressive stresses. The study considered two cases for each case. In first case load was applied on the crank end and in other case load was applied at the piston end. Ultimately results were compared for optimization purpose. On comparison it was found that weight of connecting rod was reduced by 0.477 g, due to which shear force

[1] Lovely Professional University (LPU), Phagwara, Punjab, India.

was also reduced. Also, stress was maximum at piston end so to reduce stresses, material was increased. Shaari *et al.* (2010) presented the optimization technique to minimize mass of connecting rod and reduce the cost of production. Modeling of the connecting rod was done in SOLID WORKS and analysis for the same in PATRAN. Firstly crank end was fixed and secondly pin end was fixed for same axial load in tension and compression. The study analyzes TET4 and TET10 mesh type under the same boundary conditions. It was concluded that maximum stresses are much lower than the initial design and also connecting rod becomes 11.7% lighter. Kumar Sudershn *et al.* (2012) performed modeling and analysis of two wheeler connecting rod. For the analysis three materials viz. aluminum boron carbide, aluminum and carbon steel were taken, and results were compared on the basis of selected parameters. The results showed the percentage increase in stiffness of aluminum boron carbide was more, percentage reduction in weight is same for aluminum 360 and aluminum boron carbide. Kumar *et al.* (2012) also performed the optimization of the connecting rod parameters using CAE. The study shows that on modification of different parameters of connecting rod there is improvement in the existing results. Stress which was maximum at the pin end can be reduced by increasing the material near the pin end. Weight was also reduced by 0.004 kg which was not significant but there is reduction in inertia forces. Thomas George Tony *et al.* (2011) performed the analysis to improve the fatigue life cycles of connecting rod. Connecting rod was modeled in CATIA and its static analysis was done using ANSYS to understand the fatigue locations in the connecting rod. It was concluded that shot peening significantly improve fatigue life of the connecting rod. Pathade *et al.* (2012) analyzed the two most critical areas of the connecting rod. Specified dimensioned connecting rod was modeled in PROE which was later imported to ANSYS. In their problem statement three different loads were applied at pin end whereas the crank end was fixed. When theoretical and FEA results were compared, it was found that stresses were maximum at the small end.Ranjbarkohan *et al.* (2011) studied a case of high rate of damage of connecting rod of Nissan Z24 vehicles in Iran. The study was divided into two parts,First was the kinematic and kinetic analysis and the other was static analysis. In kinetic and kinematic analysis MSCADAMS software and experimental data was used to obtain a combustion chamber pressure. In static analysis connecting rod was modeled in SOLIDWORKS and its analysis was done using ANSYS. It was concluded that tensile stress was maximum at the pin end, Maximum pressure was obtained at the pin end and rod and for future fatigue analysis was proposed. CIOATÃ *et al.* (2010) performed static analysis of the connecting rod. Connecting rod was modeled in Autodesk Inventor and for its analysis they had used ANSYS. Comparison of the deformation by the conventional and FEM was done and it was found that there was adeformation of 0.073 by conventional method and 0.036 mm by FEM. In the current study two materials were used for the connecting rod one is Conventional and and the other is Composite Material (E-Glass/Epoxy). Comparison showed the varied results which are elaborated as under.

OBJECTIVE

The main objective of this study is to replace the conventional material of connecting rod i.e., steel with the Composite material (E-Glass/Epoxy). In this study von misses stresses, deformations and other parameters are ascertained which has been done by doing the FEA of the connecting rod. Linear static analysis was performed on MSC.PATRAN of the connecting rod for the conventional as well as for the E-Glass/Epoxy to get the varied results.

METHODOLOGY

Modeling of Connecting Rod

Connecting rod was modeled with CATIA V5 R10 software. The orthographic views of the connecting rod taken from the drafting tab of the CATIA software are shown in Figure 1. The solid model of the connecting rod drawn from the same software is shown in Figure 2.

Steps to Model Connecting Rod

- Open CATIA interface, from start menu in mechanical design select part design.

Figure 1: Orthographic Projections of Connecting Rod

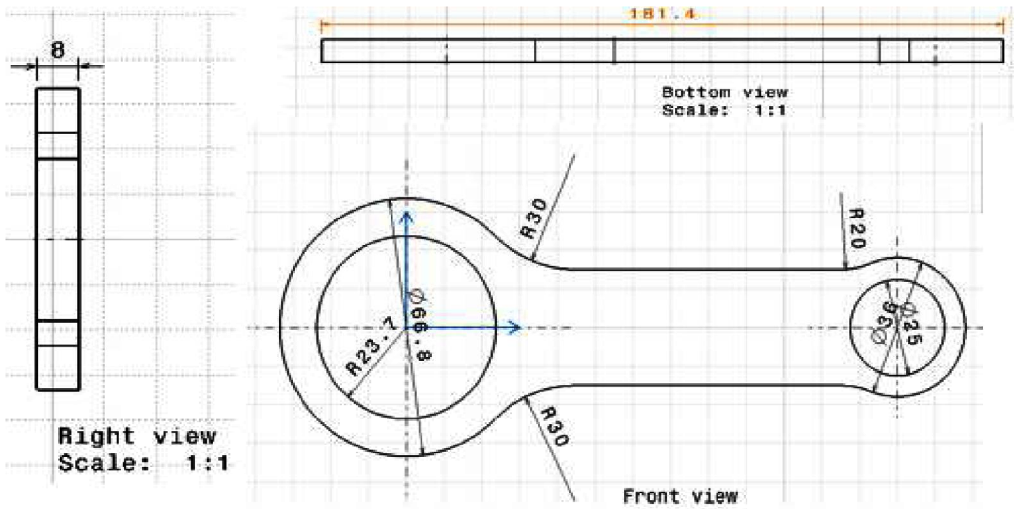
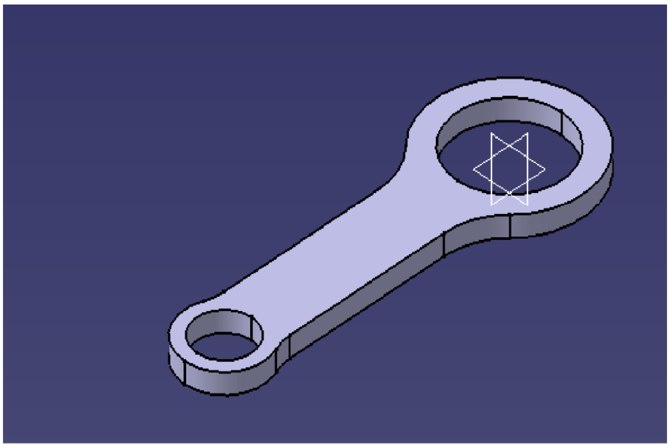


Figure 2: Solid Model of the Connecting Rod



- Choose X-Y plane as the basic drawing area.
- Open sketcher, draw the sketch of the basic connecting rod.
- Trim the parts that are not required.
- Join all the parts.
- Apply the pad command and give the thickness as per the requirement.
- Solid Model is ready. Now save the file in IGES format which will enable easy import to any analysis software.
- For getting the orthographic views on drawing sheet, choose drafting from the start menu.
- In the drafting interface there appear three views which are used here.

Once the connecting rod is modeled, it will be exported from CATIA to MSC.PATRAN.

Finite Element Analysis (FEA) and Comparison of the E-Glass/Epoxy v/s Conventional Steel Connecting Rod

In this study two finite element models were analyzed. One analysis was done for the conventional steel and the other was performed on the E-Glass/Epoxy connecting rod keeping all the parameters same for both the analysis. Table 1 shows the material properties that has been taken for the conventional steel material and Table 2 shows the material properties for the E-Glass/Epoxy material. The basic difference between the two materials is their behavior, E-Glass/Epoxy is orthotropic and the Conventional steel is isotropic. For Linear static analysis the solid model was imported from CATIA to MSC.PATRAN.

Table 1: Material Properties of Conventional Steel

Material Properties	Values
Behavior	Isotropic
Modulus of Elasticity E	2.1e5 MPa
Poisson Ratio μ	0.266
Ultimate Tensile Strength σ_{ut}	1272 MPa
Yield Strength σ_y	1158 MPa
Density ρ	0.00000785 kg/mm^3

Table 2: Material Properties of Composite Material E-Glass/Epoxy

Material Properties	Values
Behaviour	Orthotropic
Modulus of Elasticity E_{xx}	38e3 MPa
Modulus of Elasticity E_{yy}	13e3 MPa
Modulus of Elasticity E_{zz}	13e3 MPa
Poisson Ratio μ_{xy}	0.31
Poisson Ratio μ_{yz}	0.05
Poisson Ratio μ_{zx}	0.31
Modulus of Rigidity G_{xy}	1000 MPa
Modulus of Rigidity G_{yz}	16 MPa
Modulus of Rigidity G_{zx}	60 MPa
Yield Strength σ_y	900 MPa
Density	0.00000185 kg/mm^3

Steps Involved in FEA

Meshing

In the meshing interface of the MSC.PATRAN curve mesh seed should be selected for the curve path and uniform for the linear surface. Type of mesh selected should be solid. In solid mesh there are two type of elements available TET4 and TET10. In the present study TET 4 was taken although results may be improved by using TET10.

Properties

For the first FE model with isotropic behavior material name is given "steel' and the properties given in Table 1 were taken. Element type selected was 3D solid. For the second FE model same element was selected and name was given as otho_E_Glass. Properties for this model was taken from Table 2.

Constraint

Two constraints are used in this analysis

- Displacement Constraint: In both the FE models one end, i.e., the crank end of the connecting rod was fixed and the other end was not allowed to rotate about *x* and *z* axis but free to rotate about *y*-axis. Also the pin end or the small end is allowed to move freely in transnational direction (in *x*-*y*-*z*)
- Load Constraint: In both the FE models no external force is acting on the big end but on the pin or the small end 700 N of compressive force is applied in the *x*-direction whereas there is no external force in the *y* and *z* direction.

Analysis

When all loads and displacement are applied analysis would be last step. After analyzing attach the result with the model. Once the attachment is done for viewing different results select results tab in the PATRAN interface.

Results

In the results interface select the fringe/deformation tab. A pop-up will appear, in the pop-up select the model and the corresponding result parameter. In this step n number of results can be seen for the different input parameters.

RESULTS AND DISCUSSION

In the modeled connecting rod, pin end was analyzed first for both the E-Glass/Epoxy and the Conventional Steel. The complete fringe diagram showing the distribution of von-misses stresses is shown in Figure 3 for Conventional steel and in Figure 4 for E-Glass/Epoxy. Figures 5 and 6 shows the constraint

Figure 3: Von-Mises Stresses of Isotropic Conventional Stee

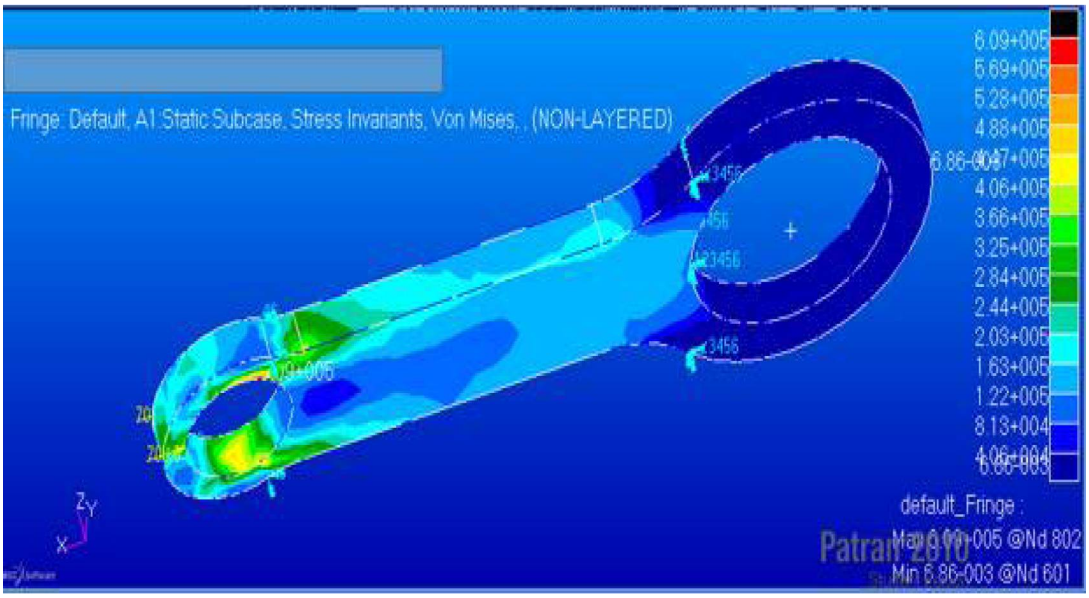


Figure 4: Von-Mises Stresses of Orthotropic E-Glass/Epoxy

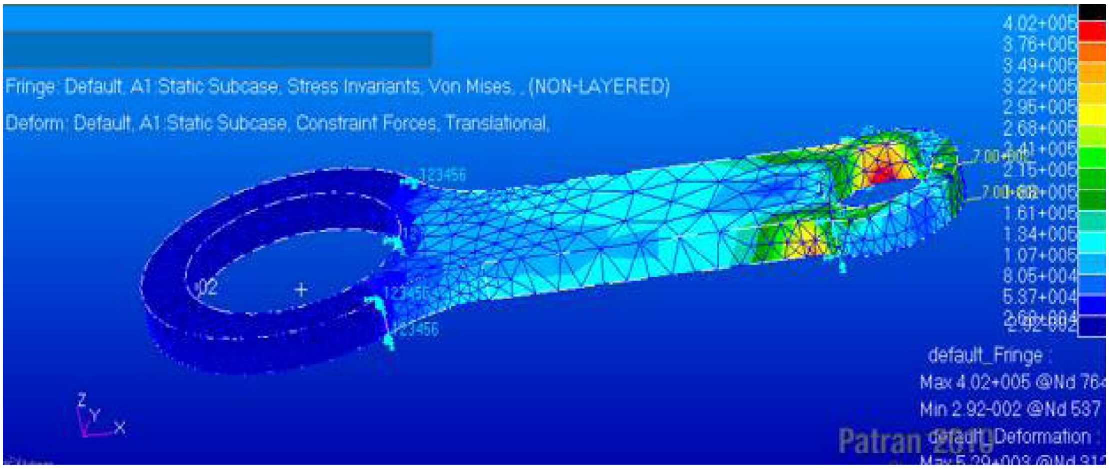


Figure 5: Translational Forces *x* Component Conventional Steel

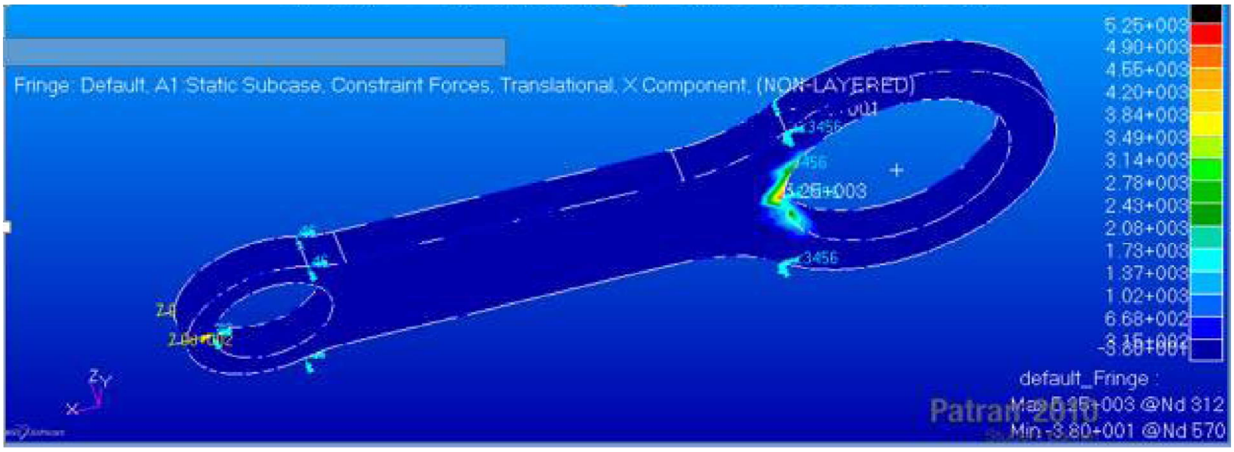


Figure 6: Translational Forces *x* Component E-Glass/Epoxy

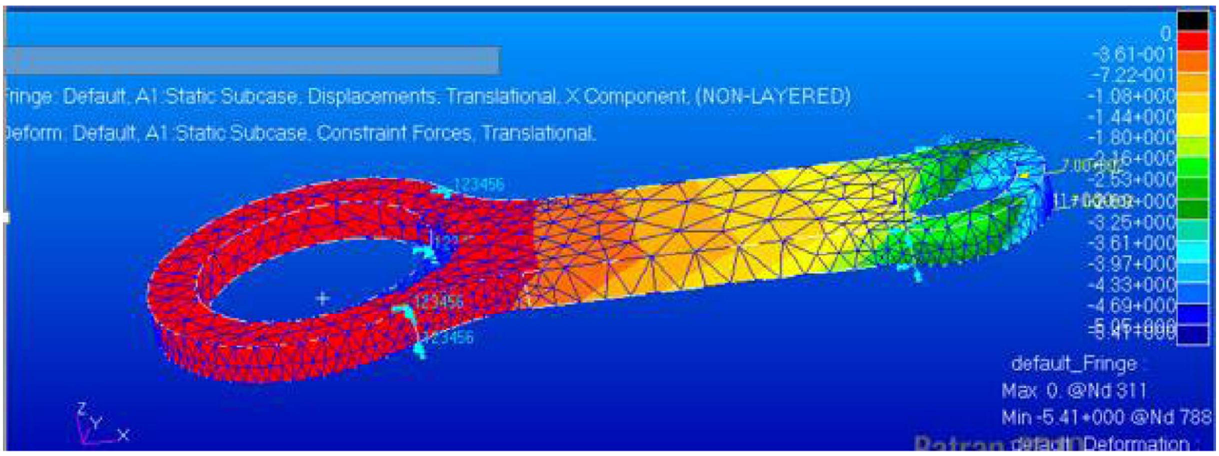


Figure 7: Displacement *x* Component for E-Glass/Epoxy

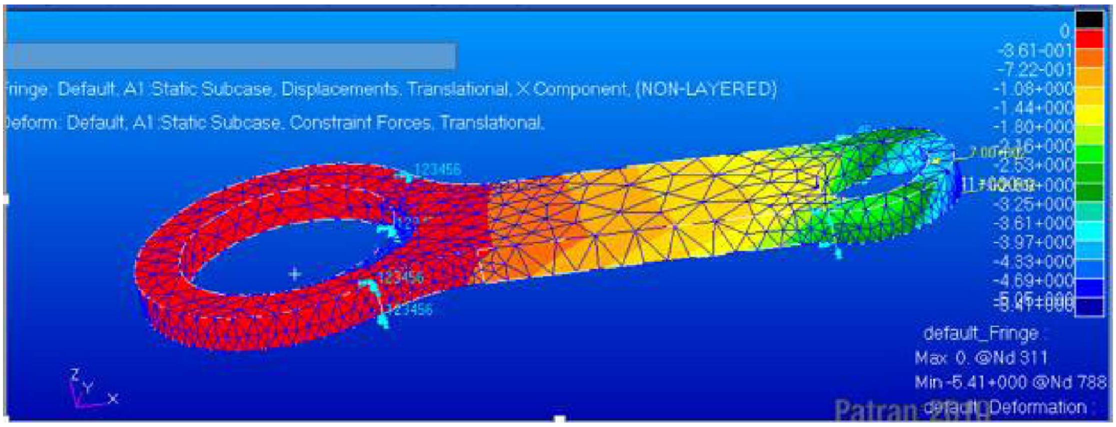
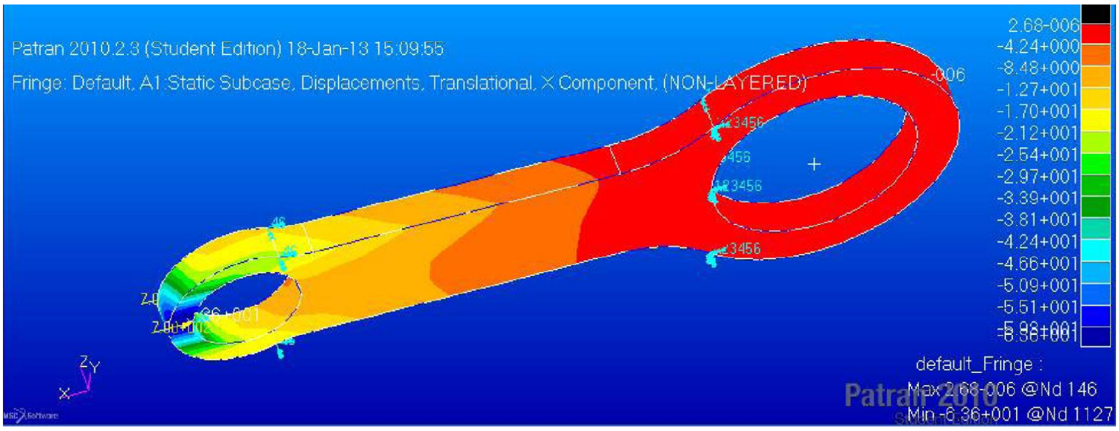
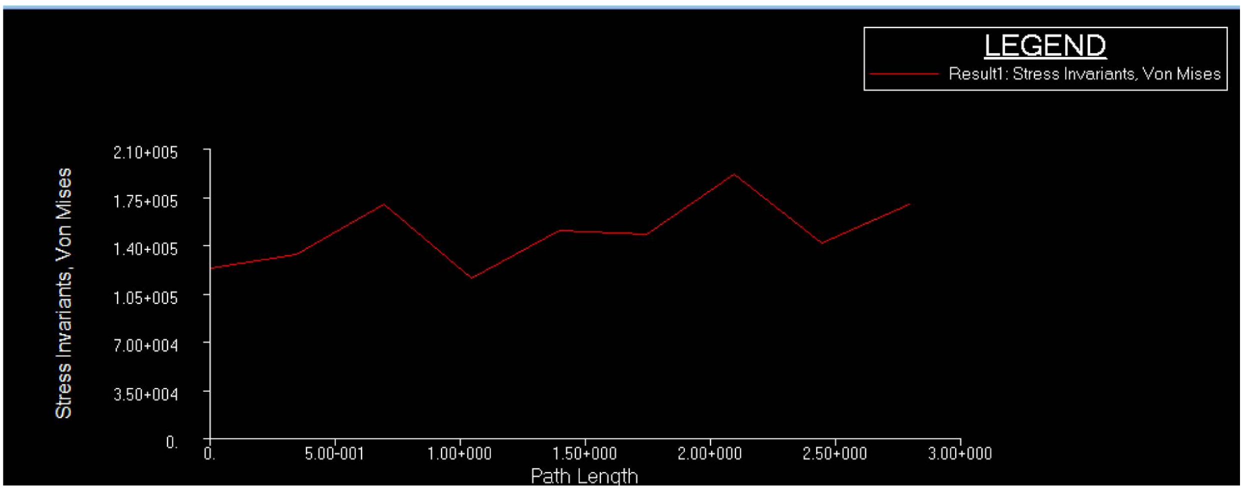


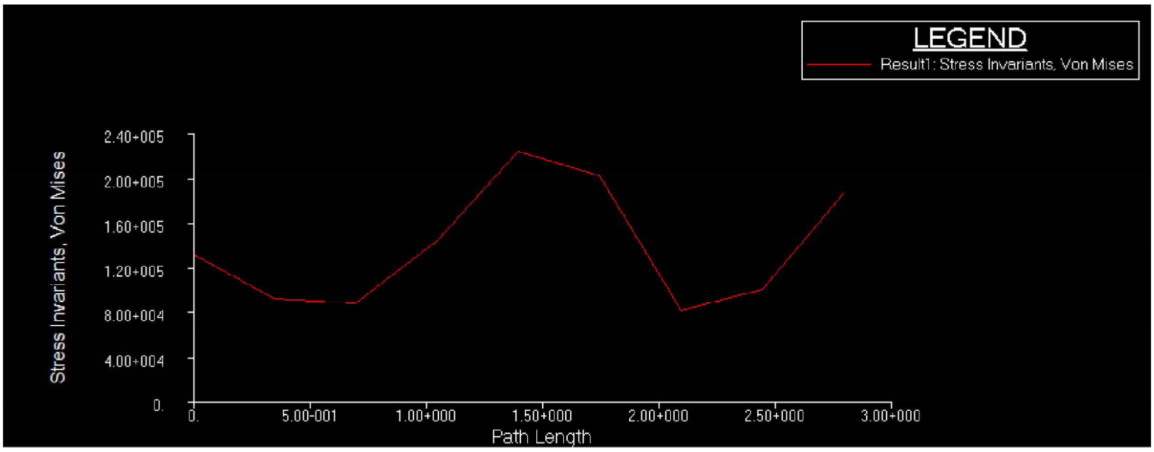
Figure 8: Displacement *x* Component for Conventional Steel



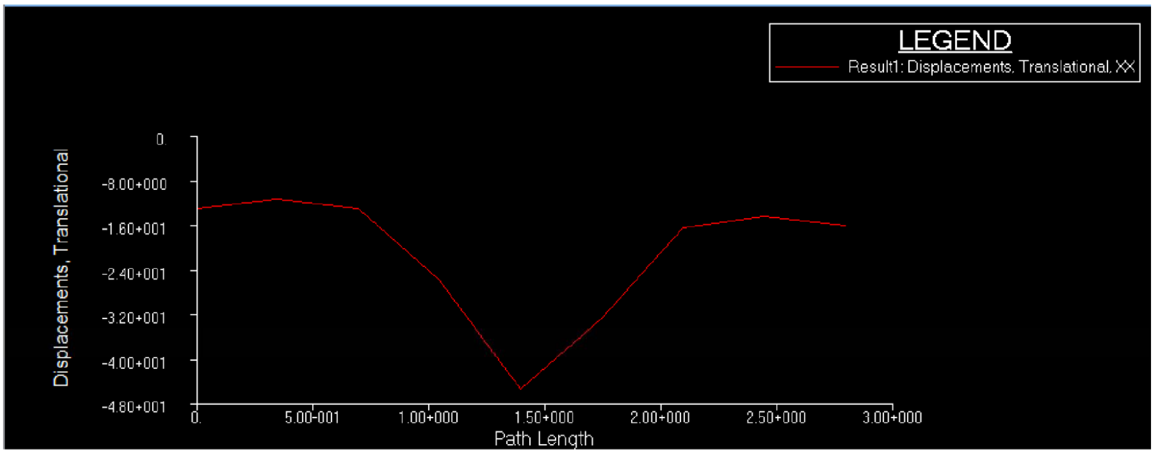
Legend 1: Variation of Von-Mises Stresses (Pin End) for Conventional Steel



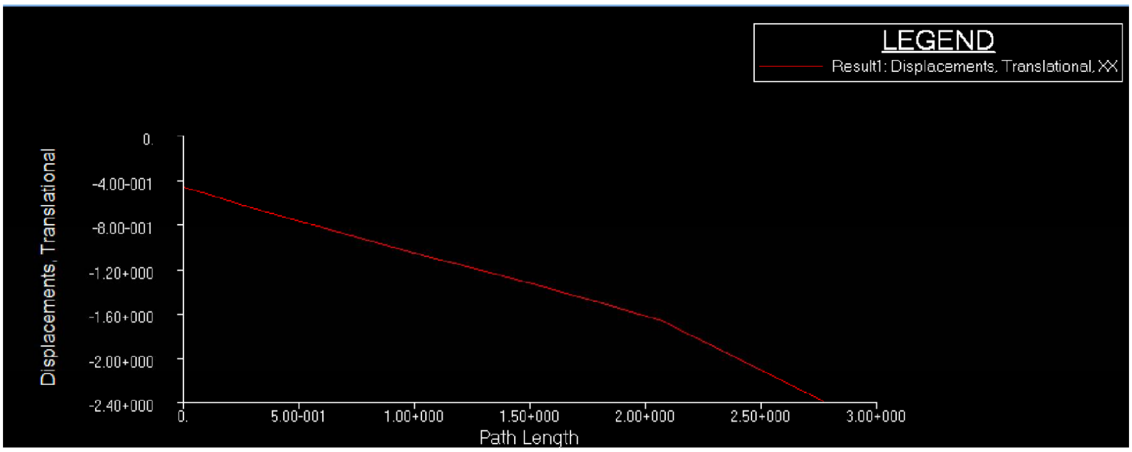
Legend 2: Variation of Von-Mises Stresses (Pin End) for E-Glass/Epoxy



Legend 3: Displacement Variation at Pin End for Conventional Steel



Legend 4: Displacement Variation at Pin End for E-Glass/Epoxy



translational forces in X-direction for steel and E-Glass/Epoxy respectively. On comparing the results shown in Figures 3 and 4 it becomes clear that E-Glass/Epoxy should be used in place of conventional steel as the maximum stress is reduced largely when material used was E-Glass/Epoxy. There is reduction of 33.99% of stresses when material used was E-Glass/Epoxy. The variation of Von-Misses stresses with the path length at the pin end for conventional steel is shown in Legend 1 and the same for E-Glass/Epoxy is shown in Legend 2. Figures 7 and 8 shows the displacement of the E-Glass/Epoxy and Conventional steel respectively. It can be seen that when material for connecting rod was E-Glass/Epoxy displacement observed reduction by 0.0026%. Although the % reduction seen in "*x* component-displacement" is small but it will affect its life. Lastly, Legends 3 and 4 were used to differentiate the displacement variation at the pin end for the conventional steel and for the E-Glass/Epoxy respectively.

CONCLUSION

Stress analysis of the connecting rod was done using FEM (Finite Element Method). Two Different materials were used for the "Conrod" and later the variation in the results were compared. Materials selected for comparison were E-Glass/Epoxy and the Conventional Steel. Finite element analysis was done using MSC.PATRAN. The results obtained during the analysis were in compliance with the existing studies.

- On comparing the von-misses stresses in the two materials it was found that there is reduction of 33.99% of stresses when convention steel was replaced with the orthotropic E-Glass/Epoxy. For connecting rod it is suggested to replace Conventional steel with E-Glass/Epoxy.
- When the Displacement *x* component was compared, again there was reduction of 0.026% displacement when material used was E-Glass/Epoxy.

The current analysis of connecting rod was done by considering mesh TET4 and to get varied or improved results mesh TET10 can be used and the results of the mesh types can be compared.

REFERENCES

1. CIOATĂ G V and Kiss I (2010), "Computer Aided Design of the Connecting Rod", *Faculty of Technical Sciences Novi Sad, Machine Design*.
2. http://www.mscsoftware.com/Academia/Learn/NastranPatran-tutorials.aspx
3. Kumar S K, Reddy T K and Hussain A S (2012), "Modeling and Analysis of Two Wheeler Connecting Rod", *I. J. Mer.*, Vol. 2, No. 5, pp. 3367-3371.
4. Kumar A, Grover K and Budania B (2012), "Optimization of Connecting Rod Parameters Using CAE Tools", *I. J. LTET.*, Vol. 1, No. 3.
5. Pal S and Kumar S (2012), "Design Evaluation and Optimization of Connecting Rod Parameters Using FEM", *I. J. Emr.*, Vol 2, No. 6, pp. 21-25.
6. Pathade C V, Patle B and Ingale N A (2012), "Stress Analysis of IC Engine Connecting Rod by FEM", *I. J. EIT.*, Vol. 1, No. 3.

7. Ranjbarkohan M, Asadi R M and Dardashti N B (2011), "Stress Analysis of Connecting Rod of Nissan Z24 Engine by the Finite Elements Method", *Australian Journal of Basic and Applied Sciences*.

8. Shaari M S, Rahman M M, Noor M M, Kadirgama K and Amirruddin A K (2010), "Design of Connecting Rod of Internal Combustion Engine: A Topology Optimization Approach", pp. 155-156, NCMER, Faculty of Mechanical Engineering UMP Pekan, Kuantan, Pahang, Malaysia.

9. Thomas G T, Srikari S and Suman J L M (2011), "Design of Connecting Rod for Heavy Duty Applications Produced by Different Processes for Enhanced Fatigue Life", *SASTECH Journal*, Vol. 10, No. 1.

10. Timoshenko S (1948), *Strength of Materials*, 2^{nd} Edition, D Van Nostrand Company Inc.