FINITE ELEMENT ANALYSIS OF CONNECTING ROD USING NASTRAN SOFTWARE

Wisam Abdul-Lateef

University of technology /Electromecchanical Department

Abstract

Connecting rods , are widely used in automotive internal combustion engines .In this paper, the connecting rod model based on finite element analysis (FEA)is proposed. Critical location of maximum stress and suitable materials for the connecting rod has been identified and the contact stress analysis and dynamic behavior of it has been investigated.The structural model of the connecting rod was utilizing the (Solid works) and Steel alloy (C-70) materials. The linear static stress distribution and dynamic analysis are investigated using Finite Element software package (NASTRAN).For TET10 mesh the linear static stress analysis and the critical location was located at node (174743) and the critical location for TET4 was at node (110703).Then it can find that TET10 is able to capture the higher stresses than TET4 for the same global length and size.

Keywords: Finite element method, dynamic analysis, connected rod, Nastran Software.

الخلاصة

ذراع التوصيل يستخدم بـ شكل وا سع في المركبات ذات محركات الاحتراق الداخلي .في هذه الورقة نموذج ذراع التوصيل يـ ـعـ تمد على تحليل العناصر المحددة المفترضة. تم تعين المواقع الحرجة لـ لاجهادات الـ عظمى والمعادن الملائمة لذراع التوصيل وكذلك تحليل الاجهادات الملامسة لذراع لـها تـ م الـ تحـ قق مـ نة. أ سـ تخدم (Solid works) الـ تو صـ يل, الـ سـ لوك ا لـ ديـ نامـ يكي لـ نموذجهيكل ذراع التوصيل وسبيكة الفولاذ (C-70) كمعدن لـه. توزيع الاجهادات الخطي والـ تحـ لـ يل لـ ديـ نامـ يكي تـ م تـ حـ قـ يـ قه بـ برنـ امج العناصر المحددة(NASTRAN) بواسطة الشبكة (TET10) وتحليل الإجهادات الخطية تم تـ ـعـ ين الـ نـ قطة الحرجة في العقدة (174743) وتم تعين الـ نـ قطة الحرجة لـ لـ شـ بكة TET4 فـ ي العقدة (110703) بعد ذلك وجد ان(TET10) له القابلية والقدرة على تحديد وتحمل الاجهادات الأعلى مقارنة بـ(TET4) لنفس الطول العالمي والحجم .

1- INTRODUCTION

In modern automotive internal combustion engines, the connecting rods are most usually made to absorb high impact stresses that occur onto it.Bending stresses appear due to eccentricities, crankshaft, case wall deformation, and rotational mass force Therefore, a connecting rod must be capable of transmitting axial tension, axial compression, and bending stresses caused by the thrust and pull on the piston and by centrifugal force (Afzal and Fatemi, 2003). The connecting rod of the tractors is mostly made of cast iron through the forging or powder metallurgy. The main reason for applying these method is to produce the components integrally and to reach high productivity with the lowest cost (Repgen, 2005). Nevertheless, connecting rod design is complicated because the engine is to work in variably complicated conditions and the load on the rod mechanism is produced not only by pressure but also inertia (Whittaker, 2001). Rasekh et al. (2009) explained study of experimental equation for a Tractor and performed the result using . The maximum stresses in different parts of (MF-285) connecting rod were determined (Asadi et al., 2010). From the analysis, three parts were being considered of the stress distributions which are pin end, rod and crank end.Mirehei et al. (2008) investigated the connecting rod fatigue of universal tractor (U650) was through the ANSYS software application and its lifespan was estimated. The connecting rod behavior affected by fatigue phenomenon due to the cyclic loadings and to consider the results for more savings in time and costs, as two very significant parameters relevant to manufacturing. The results indicate that with fully reverse loading, one can estimate longevity of a connecting rod and also find the critical points that more possibly the crack growth initiate from. Bari et al. (2004) compare FEA of slab with others analytical solution. Slabs are most widely used structural elements that transmit load to the supporting walls and beams and sometimes directly to the columns by shear and torsion. Similarly with various classical mathematical procedures, simple beams were analyzed in which the concrete and the steel reinforcement were represented by two-dimensional triangular finite elements. Conle and Mousseau (1991) used the vehicle simulation and finite element result to generate the fatigue life contours for the chassis component using automotive proving ground load history result combine with the computational techniques. They concluded that the combination of the dynamics modeling, finite element analysis is the practical techniques for the fatigue design of the automotive component. Kim et al. (2002) was studied a method for simulating vehicles dynamic loads, but they add durability. Yang et al. (1992) were meshing finite element modeling. The connecting rod pin end as shown in **Fig.(1)** contains 1012 linear HEXAGONAL elements, 4 linear PENTADACTYL elements, and 1569 grid points. The design goal is to minimize the material volume subject to a constraint on the von Mises stress. This constraint is imposed at each node in the finite element model of the connecting rod head except the nodes at the reentrant corner where the wrist pin leaves the rod. The singularity effects that occur here can be considered by imposing a stress concentration factor, but the interface between the pin end and crank end generally requires complex modeling techniques In this paper, MSC.NASTRAN finite element techniques have been used as a tool to model the mechanical properties of the connecting rod firstly. Three-dimensional linear tetrahedron solid elements (TET10) and tetrahedral elements (TET4) are used for the initial analysis based on the loading conditions. Convergence of stress and strain energy was considered as the criteria to select the mesh size, Developed the structural model and identify the critical locations and predict the dynamic behavior of connecting rod.

2 - THEORETICAL BASIS OF CONNECTING ROD

(Shenoy, 2005) performed the optimization of connecting rod based on experimental results **Fig.(2)** shows the coordinate system of connecting rod at crank end.

The normal pressure (P) on the contact surface is given by

$$P = P_o \cos\theta \tag{1}$$

where

P: is Normal pressure

P_o: is Normal pressure constant

The load is distributed over an angle of 180°. The total resultant load(P_t) given by

$$P_t = \int_{-\frac{\pi}{2}}^{\frac{\pi}{2}} P_o (\cos\theta)^2 rt d\theta = P_o rt\left(\frac{\pi}{2}\right) \tag{2}$$

where

P_t :is total resultant load (Tensile)

r: is small end radial

R :is big end radial

The normal pressure constant (P_o) is calculated from Eq. (3):

$$P_o = P_t / rt\left(\frac{\pi}{2}\right). \tag{3}$$

For compressive loading of the connecting rod, the crank and the piston pin ends are assumed to have a uniformly distributed loading through 180° contact surface. The normal pressure is given by Eq. (4):

$$P = P_o. \tag{4}$$

The total resultant load (P_c) is expressed in Eq. (5):

$$P_c = \int_{-\frac{\pi}{3}}^{\frac{\pi}{3}} P_o (\cos\theta) rt d\theta = P_o rt\sqrt{3}. \tag{5}$$

where

P_c: is total resultant load (compression)

The normal pressure constant (P_o) is then given by Eq. (6):

$$P_o = P_c / rt\sqrt{3}. \tag{6}$$

In this study four Finite Element (FE) models were analyzed. FEA for both tensile and compressive loads were conducted. Two cases were analyzed for each case, Firstly, load applied at the crank end and restrained at the piston pin end, and secondly, load applied at the piston pin end and restrained at the crank end and the axial load was 26.7 KN in both tension and compression (Shenoy, 2004)

2.1- Modeling of the Connecting Rod

Repgen, 2005 performed connecting rod is the one of the most important components in the internal combustion engine. Any unnecessary parts or shape will be removed during modeling the optimized design. A simple three-dimensional model of connecting rod with mass of 0.577 kg was developed using SOLIDWORKS as shown in **Fig.(3)**.The dimension of the connecting rod is depicted in **Fig.(4)** inspired from Shenoy (2004).

2.2- Linear Static Analysis

The FEA can be used to calculate the stress distribution for an entire component or structure. The linear static analysis is considered in this study. (Amin, 2008) was stated that the linear static analysis is one of the most common engineering analyses and represents the most basic of analysis. The term linear means that the displacement or stress is proportion to the applied force. The term static means that the forces do not vary with time. **Fig.(5)** shows the relationship between load and displacement.

Rahman (2007) was stated that, due to the load does not change according to the time, the equation can be written in simple equation which neglecting the inertia forces, damping forces and nonlinearity. Outputs or results of the linear static analysis are displacements, strains, stresses, and reaction forces under the effects of applied loads.

Static analyses derive in Eq. (7):

$$[K]\{U\}=\{f\} \qquad (7)$$

Where

$[K]$: is stiffness matrix (based on the geometry and properties)

$\{u\}$: is vector of displacements

$\{f\}$: is vector of applied forces

3- MATERIAL INFORMATION

Material properties play an important role in the FEA. The material properties are one of the major inputs to perform the optimization. The materials parameters required depend on the analysis methodology being used. The mechanical properties of C-70 steel are listed in **Table 1.**

4- RESULTS

4.1- Finite Element Modeling

Structural analysis performed to create high and low stress region from the input of material, loads, boundary condition and geometry. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry and boundary conditions (Lee et al., 2010). The structural analyses allow stresses and strains to be calculated in FEA, by using the structural model. From the analysis results, the critical areas at the structural model can be analyze and improve it in the optimize design.

Mesh study was performed on the finite element model (FEM)to define the best mesh for the analysis. It is to increase the accuracy result at the same time not high on the cost (CPU processing time). Rahman et al. (2009) b, the mesh convergences were monitored and evaluated during the analysis. The mesh convergence are based on the geometry, model topology and analysis objectives. Due to tetrahedral mesh produce high quality meshing for solid model imported from CAD systems. FEM of connecting rod is developed utilizing the MSC.PATRAN and create the mesh by using the (10) node tetrahedral elements (TET10) and (4)node tetrahedral elements(TET4).

Mesh study was performed on the FE model to ensure sufficiently fines sizes are employed for accuracy of the calculated result depends on the competitive cost (CPU time). Mesh convergence analysis was performed to determine the optimum element size (Grass et al., 2006).

It was preformed iteratively at different mesh global length until the appropriate accuracy obtained. Convergence of the stresses was recorded as the mesh global length refined. During the analysis, the specific variable and the mesh convergence was monitored and evaluated. The mesh convergence is based on the geometry, model topology and analysis objectives. For this analysis, the auto tetrahedral meshing approach is employed for the meshing of the solid region geometry. Tetrahedral (TET) meshing produce high quality meshing for boundary representation most of solids model imported from CAD systems. Since the TET is found to be the best meshing technique, the TET 4 version of the connecting rod was then used for the initial analysis. **Fig.(6)** shows the comparison of TET4 with difference global edge length of 4 mm and 8 mm. **Fig.(7)** shows the results of nodes and elements by using TET4 and TET10 by using the same edge length of 8 mm .

As already discussed before this, a TET10 was used for the solid mesh and tensile loads will be used to analyze the FEM. However, it will be two sub cases which is one part are loaded and the other part restrained. The FEA will analyze on the two cases and produce with the results. From Shenoy (2004), the uniformly distributed tensile load 180^{o}on the inner surfaces of the crank end while pin end will restrain as in **Fig .8 (a).** It is just same when load uniformly distributed on pin end surfaces, the crank end will restrain in all direction. This both cases also work exactly in compressive load. In **Fig .8 (a) and (b),** shows the boundary condition of the connecting rod in three-dimensional FE model with load and constraints.

Fig .(9) shows the von-Mises stresses contour for TET4 and TET10 meshes element at a high load level respectively. The TET10 mesh is presumed to represent a more accurate solution since TET4 meshes which are known to be dreadfully stiff (Rahman et al., 2009). TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function. The TET10 is expected to be able to capture the high stress concentration occurs on connecting rod. Both of the meshes have some distorted elements cause error to the modeling in areas of elevated stress. It can be seen that the TET10 meshes predict higher von-Mises stresses than the TET4. From the **Fig.(9)**, TET4 predicted 49.5MPa while TET10 predicted 2

Variation of stresses in term of von-Mises, Tresca, maximum principal stress and displacement for TET4 and TET10 are tabulated in **Table 2** and **Table** 3 respectively. The comparison was made between these two elements based on stresses concentration and displacement which are in **Fig.(10)** and 12 respectively. It can be seen that, TET10 is always captured the higher stresses and displacement. Thus, support the conclusion of Wang et al. (2004) which, quadratic tetrahedral elements (TET10) are very good and can always be used. For better and accurate results, TET10 was used instead of TET4. This is because, the large degree of freedom (DOF). TET10 have 30 DOF more than TET4 which only have 12 DOF.

4.2- Identification of the Mesh Convergence

presented in **Fig.(12)**,it can be seen that mesh size of 4 mm obtained largest stresses. The smaller size less than 4 mm do not implemented due the limitation of the CPU time and storage capacity of the computer. Hence, the maximum principal stress based on TET10 at(4) mm mesh size is used in the analysis since the stress is higher compared to Von Mises and Tresca principal stress.

4.3- Dynamic Analysis of Connecting Rod

The modal analysis is usually used to determine the natural frequencies and mode shapes of a component. It can also be used as the starting point for the frequency response, the transient and random vibration analysis (Rahman, 2007).The finite element analysis codes usually used several mode extraction methods. The Lanczos mode extraction method is used in this study due to it is the recommended method for the medium to large models. In addition to its reliability and efficiency, the Lanczos method supports sparse matrix methods that significantly increase computational speed and reduce the storage space. This method also computes precisely the eigenvalues and eigenvectors. The number of modes was extracted and used to obtain the connecting rod stress histories, which is the most important factor in this analysis. Using this method to obtain the first (5) modes of the connecting rod, which are presented in **Table 4** and the shape of the mode are shown in **Fig.(13)**.

5- CONCLUSION:

A detailed model of connecting rod has been developed using finite element techniques. The tetrahedral element (TET10) and tetrahedral element (TET4) are used for the initial analysis. Sensitivity analysis was performed to determine the optimum element size. It can be seen that TET10 at mesh size 4 capture highest moment levels von Mises stress and for this reason used to dynamic analysis. Smaller mesh size will give more accurate result compare with larger sizes of mesh. However, it should count limitation of (CPU)time and storage capacity of the computer during analysis. The results of the frequency are shown 5 modes and several deformed shapes.

6- REFERENCES

-Afzal A. and Fatemi A. 2003. A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods. SAE International.

-Amin, H.A. 2008. A comparative study of 2 CAD integrated FE-programs using the linear static analysis Master’s Thesis.Halmstad University, Sweden.

-Asadi, M.R., Rasekh, M., Golmohammadi, A., Jafari, A., Kheiralipour, K. and Borghei, A.M. 2010. Optimization of connecting rod of MF-285 tractor. Journal of Agricultural Technology. 6(4): 649-662.

-Bari, M.W., Rahman, M.M., Ali, M.M.Y. and Awall, M.R. 2004. Finite Element Analysis of Slab and A Comparative Study With Others Analytical Solution. Seventh International Conference on Concrete Technology in Developing Countries (7th ICCT), pp. 265-272, Kuala Lumpur.

-Conle, F.A and Mousseau C.W. 1991. Using vehicle dynamic simulation and finite element result to generate fatigue life contours for chassis component. International Journal of Fatigue. 13(3): 195 - 205.

-Grass, H., Krempaszky, C. and Werner, E. 2006. 3-D FEM-simulation of hot forming processes for the production of a connecting rod. Computational Materials Science.36: 480-489.

-Kim, H.S., Yim, H.J. and Kim, C.M. 2002. Computational durability prediction of body structure in prototype vehicles. International Journal of Automotive Technology. 3(4): 129-136.

-Lee, M.K., Lee, H., Lee, T.S. and Jang, H. 2010. Buckling sensitivity of a connecting rod to the shank sectional area reduction, Materials and Design, 31: 2796–2803.

-Mirehei, A., Zadeh, H.M., Jafari, A. and Omid. M. 2008. Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS).Journal of Agricultural Technology.4(2): 21-27.

-Rahman, M.M. 2007. Finite element based durability assessment for a new free piston linear engine. PhD Thesis. Universiti Kebangsaan Malaysia, Malaysia
.

-Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M. and Bakar, R.A. 2009 Assessment of cylinder block for two stroke free piston linear engine using random loading. American Journal of Applied Sciences, 6(4): 726-735. -Rahman, M. M., Kadirgama, K., Noor, M. M., Rejab, M. R. M. and Kesulai, S. A. 2009b. Fatigue life prediction of lower suspension arm using strain-life approach, European Journal of Scientific Research.30(3): 437-450.

-Rasekh, M., Asadi, M.R., Jafari, A. and Kheiralipour, K. 2009. Obtaining maximum stresses in different parts of tractor (MF-285) connecting rods using finite element method. Journal of Basic and Applied Sciences.3(2): 1438-1449.

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

-Shenoy, P.S. 2004. Dynamic load analysis and optimization of connecting rod. Master's Thesis. University of Toledo, USA.

-Shenoy, P.S and Fatemi, A. 2005. Connecting rod optimization for weight and cost reduction, SAE Technical Paper, Paper No. 2005-01-0987.

-Wang, E., Nelson, T. and Rauch, R. 2004. Back to Elements - Tetrahedra vs. Hexahedra, CAD-FEM GmbH, Germany.

-Whittaker D. (2001). The competition for automotive connecting rod markets, Metal Powder Report 56(5): 32-37.

-Yang, R.J., Dewhirst, D.L., Allison, J.E. and Lee, A. 1992. Shape optimization of connecting rod pin end using a generic model. Finite Elements in Analysis and Design.11: 257-264.

Table 1: Mechanical properties of C-70 Steel.

Properties	**C-70 Steel**
Tensile strength,σ_{UTS}(MPa)	965.8
Yield strength(0.2% offset), σ_{YS}(MPa)	573.7
Young's modulus, E (GPa)	211.5

Table 2: Variation of stresses concentration at the critical location of the connecting rod for TET4 mesh

Table 3: Variation of stresses concentration at the critical location of the connecting rod for TET10 mesh

Mesh Size (mm)	No. Node	No. Element	von Mises (MPa)	Tresca (MPa)	Max principal Stress (MPa)	Displacement (mm)
16	3012	10575	22.4	22.7	22.5	0.060
12	4596	16882	30.1	30.6	30.5	0.111
8	9717	39520	49.5	50.1	49.6	0.145
4	25185	109950	67.6	68.8	67.8	0.201

Mesh Size (mm)	No. Node	No. Element	von Mises (MPa)	Tresca (MPa)	max principa stress (MPa)	Displacement (mm)
16	19410	10823	107.0	111.0	113.0	0.351
12	29698	17018	142.0	145.0	146.0	0.475
8	65200	39752	261.0	284.0	325.0	0.604
4	174743	110703	301.0	324.0	355.0	0.850

Table 4: The results of dynamic analysis

Mode	Frequency (Hz)
1	19.11
2	61.21
3	142.31
4	210.24
5	221.30

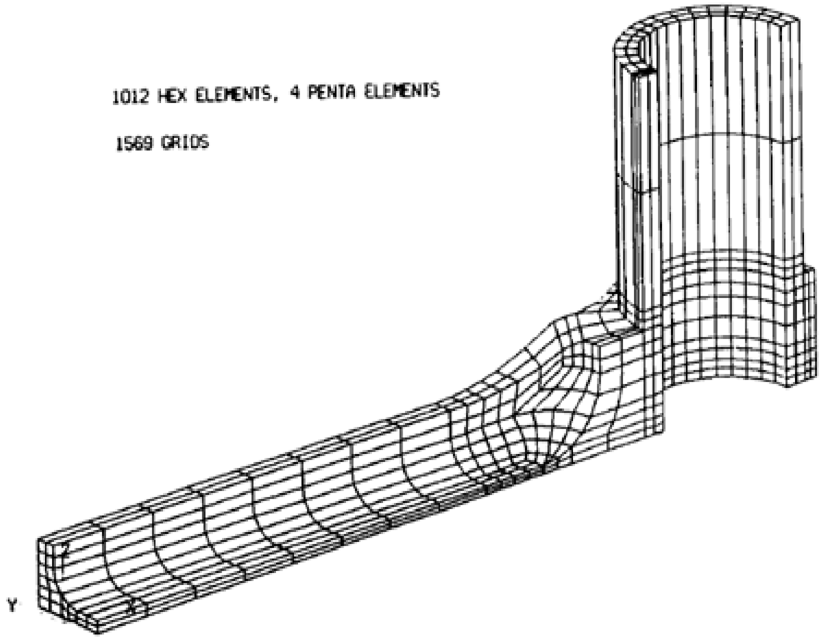


Fig 1: Finite element mesh of connecting rod pin end by (Yang et al., 1992)

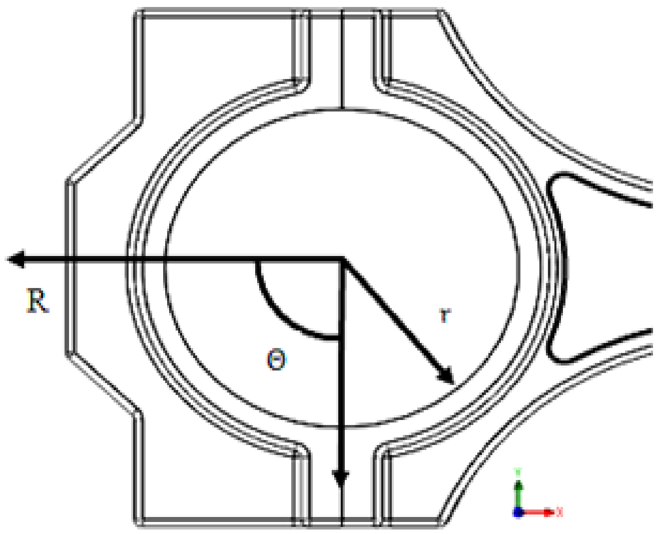


Fig 2: Coordinate system of connecting rod.

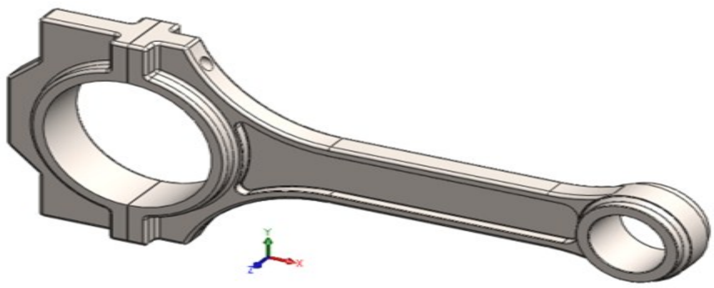


Fig 3: Isometric view of 3D modeling of connecting rod

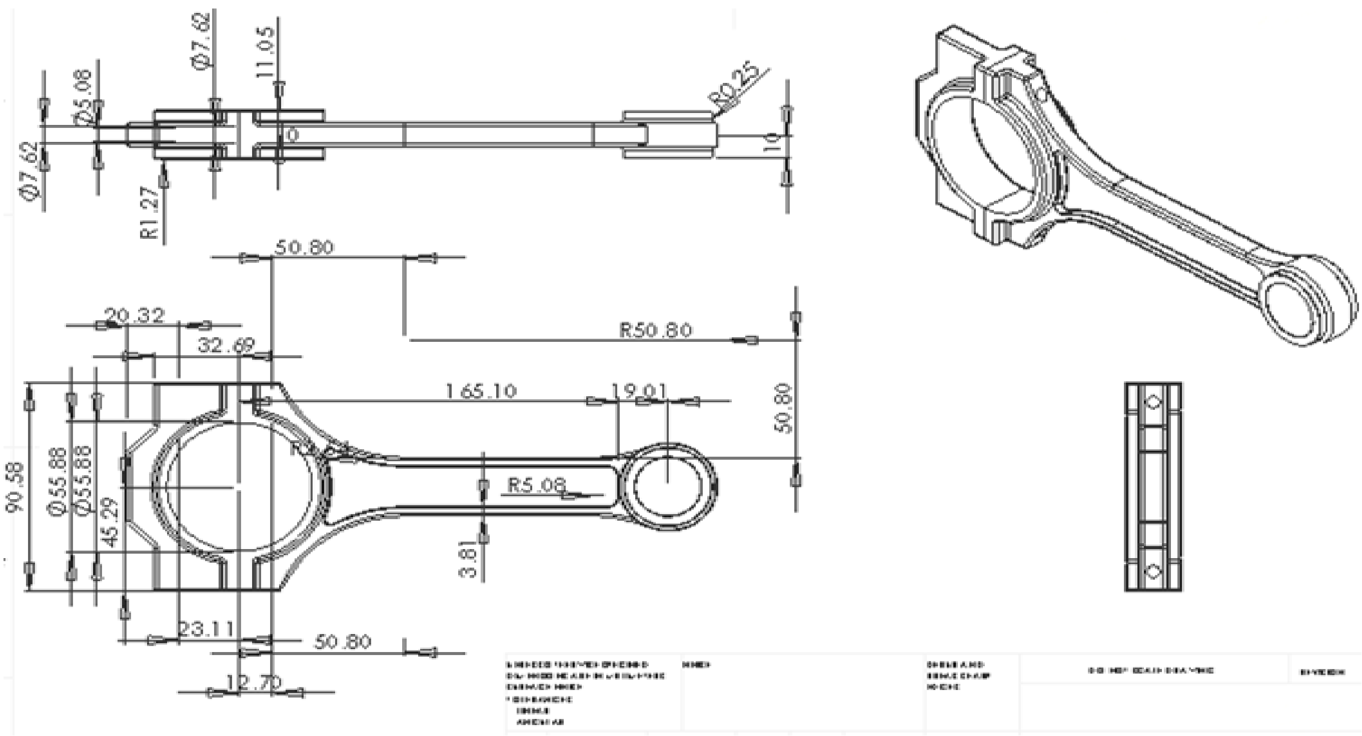


Fig 4: 3D modeling of initial connecting rod.

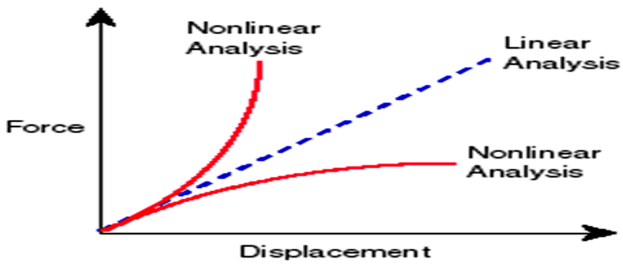
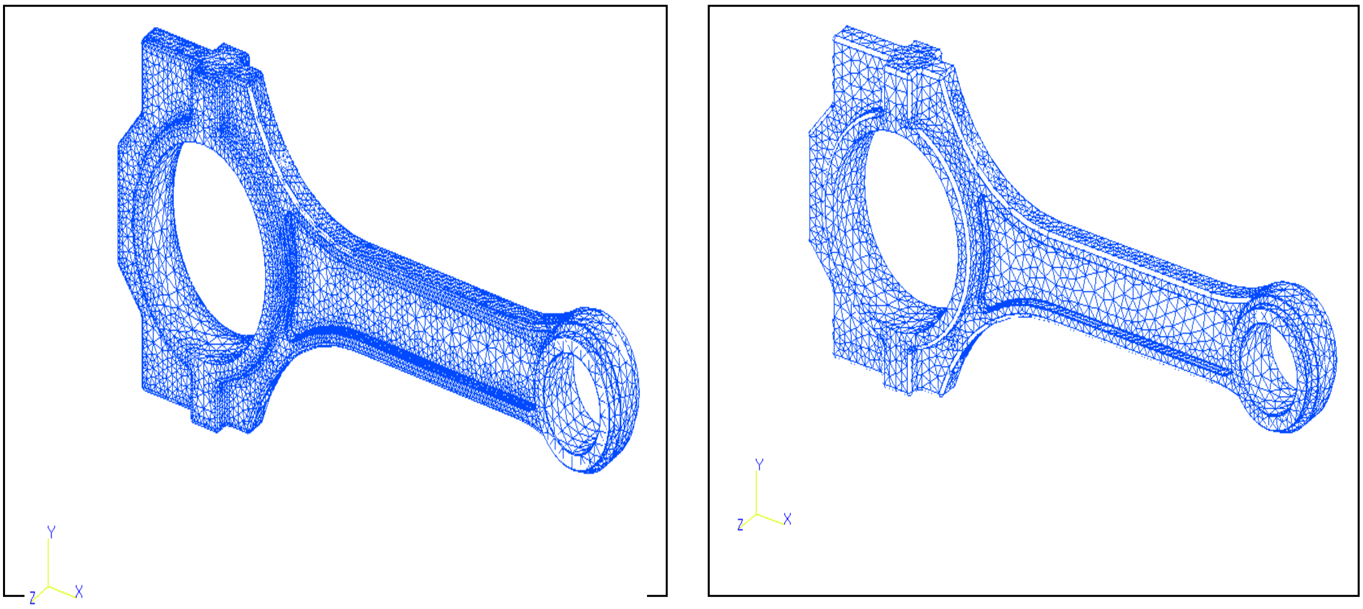
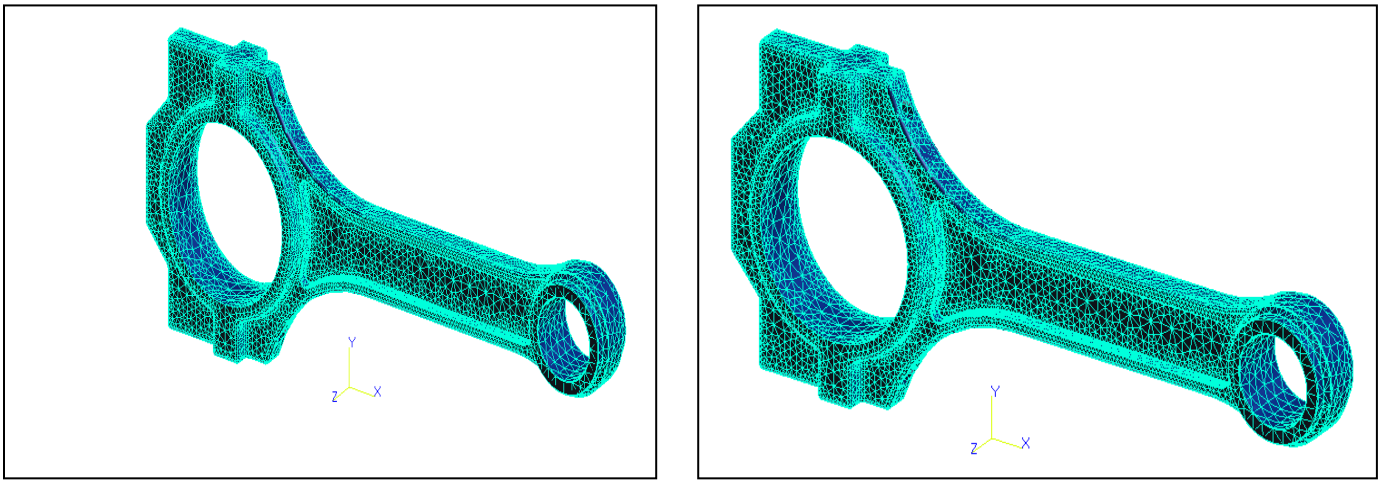


Fig 5: Relationship between load and displacement



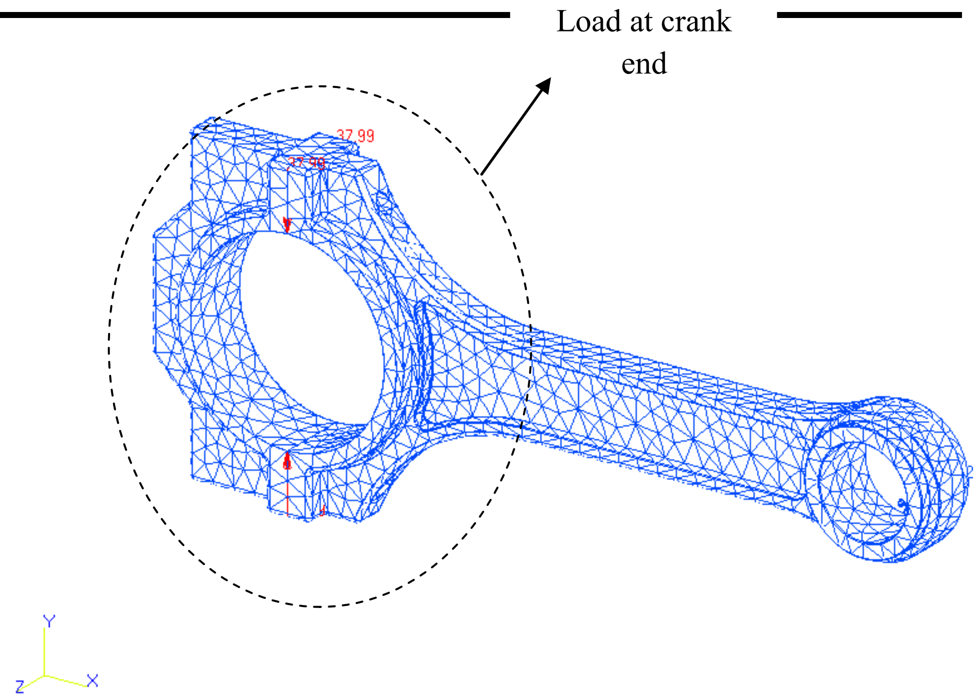
a- 4 mm (25185 nodes and 109950 elements) b-8 mm, (9717 nodes and 39520 elements).

Figure 6: Finite element model with difference mesh size.

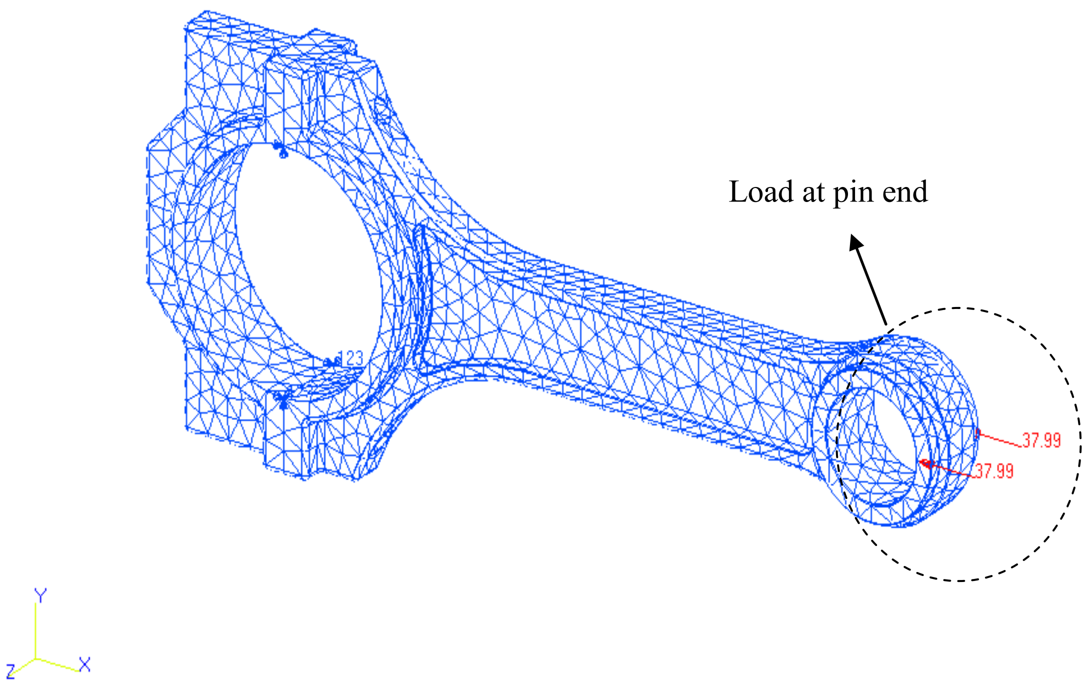


(a)TET4(109950 nodes and 25185 elements). (b)TET 10(110703 nodes and 174743 elements)

Fig 7: FEM using 8 mm mesh size with nodes and element

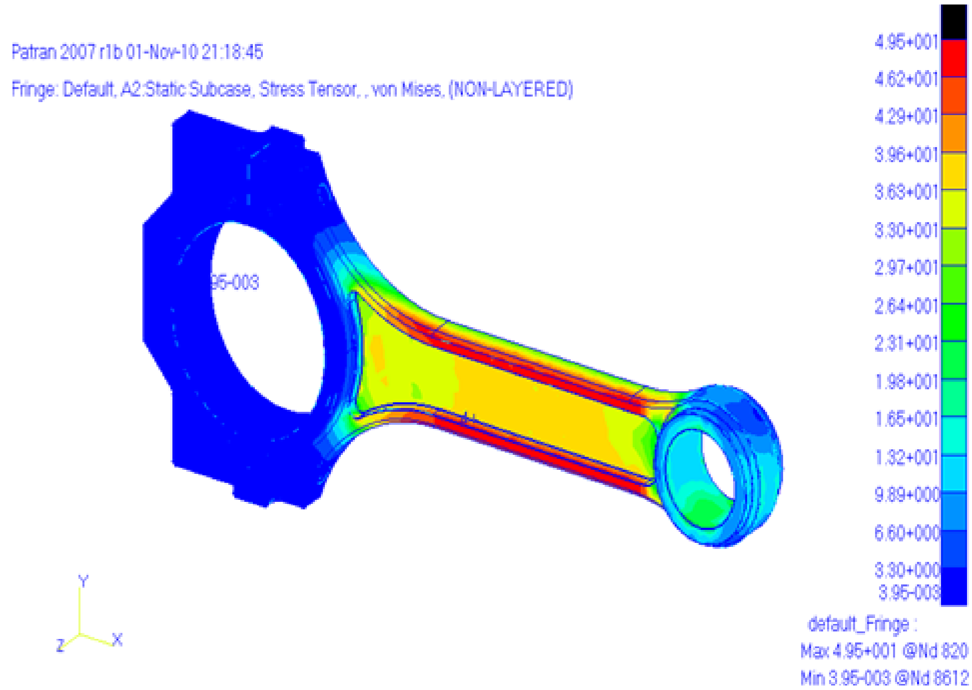


(a) Tensile load at crank end and fixed at pin end.

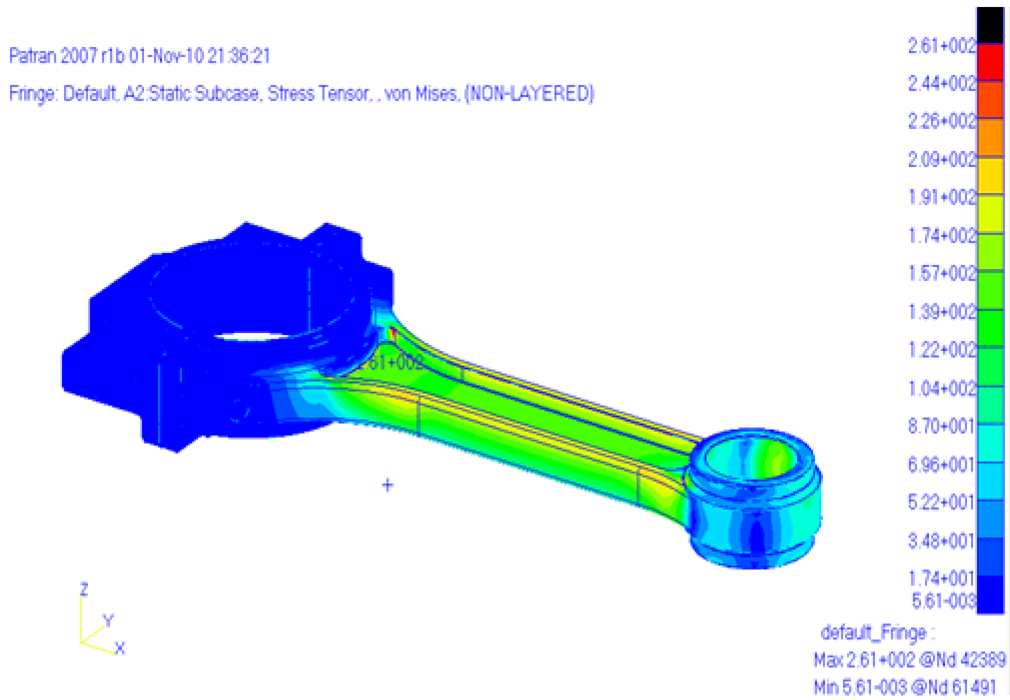


(b)Tensile load at pin end and fixed at crank end.

Fig 8: Boundary condition with tensile load.



(a) TET4(maximum stress is 49.5 MPa at node 820).



(b) TET10(maximum stress is 261.0 MPa at node 24398

Fig 9: von-Mises stresses contour

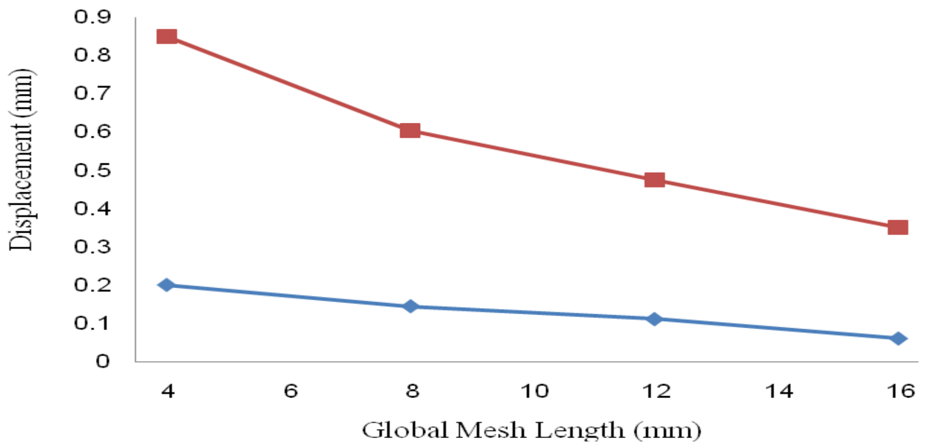
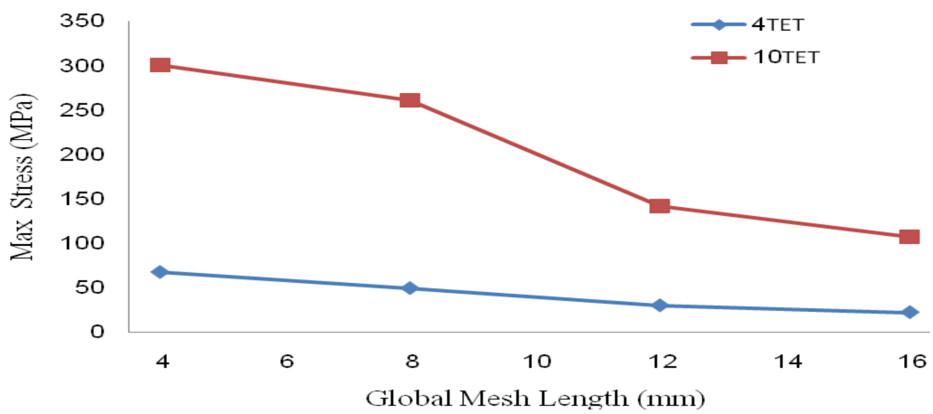
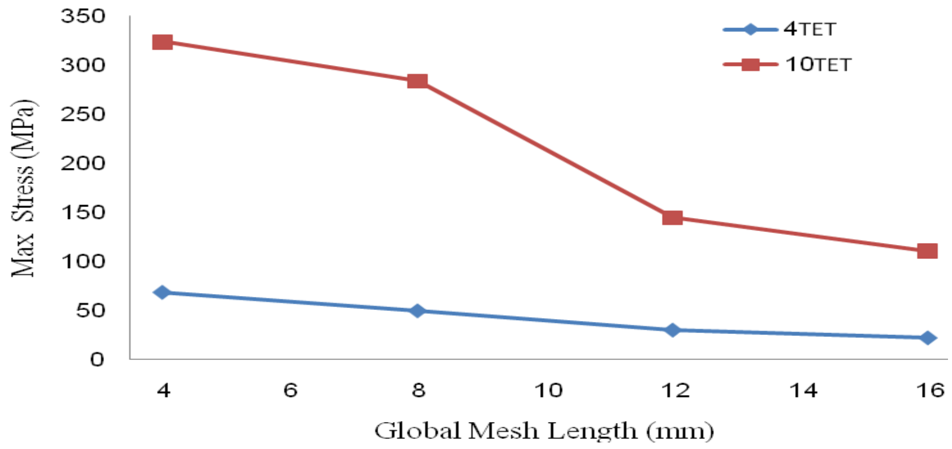


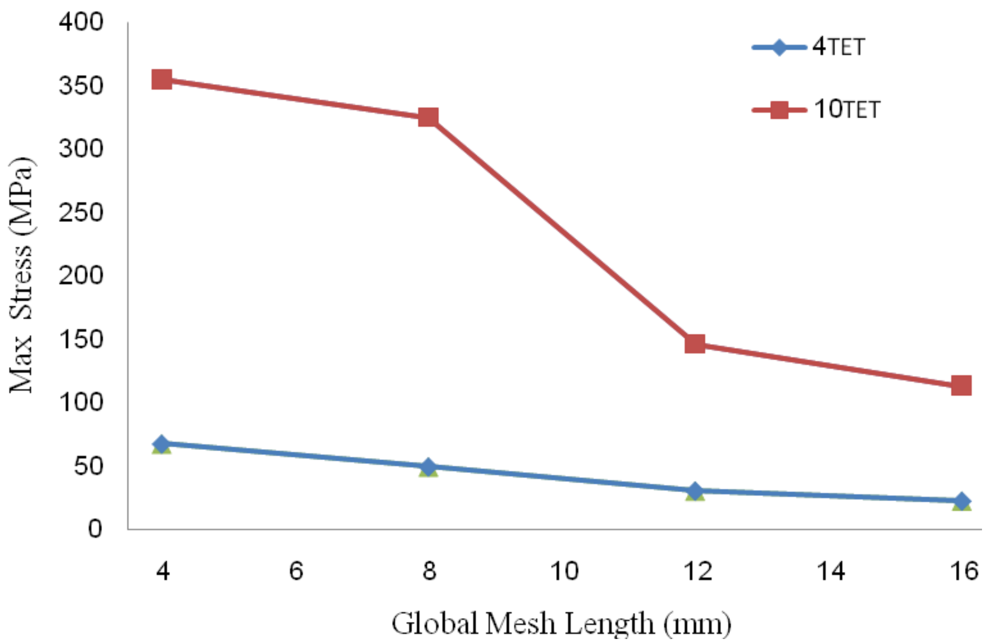
Fig 10: Comparison between TET4 and TET10 on maximum displacement.



(a) Von-Mises principal stress.



(b) Tresca principal stress.



(c) Maximum principal stress.

Fig 11: Comparison of stressesbetweenTET4 and TET10

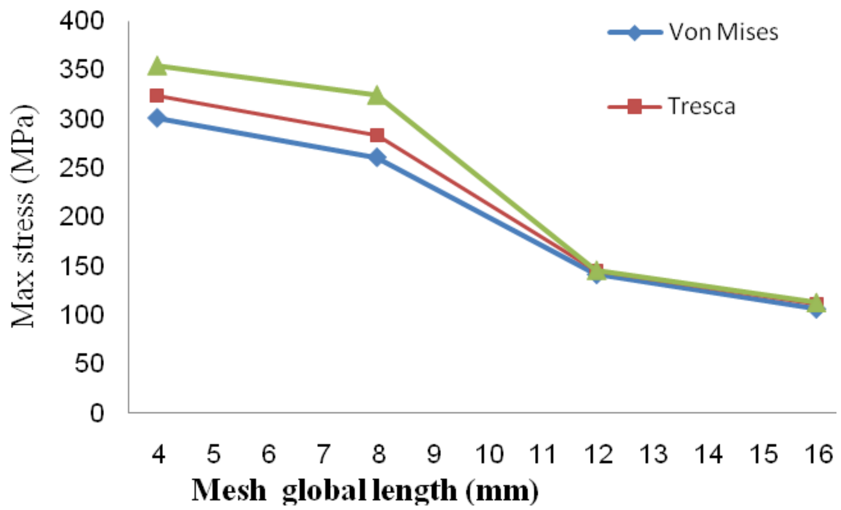
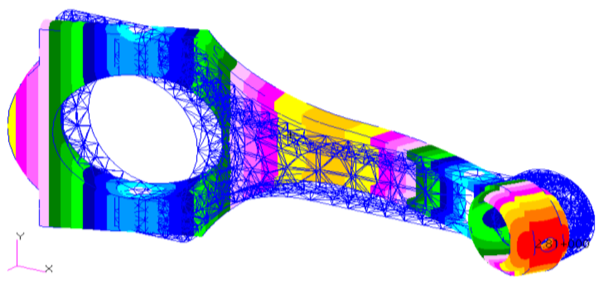
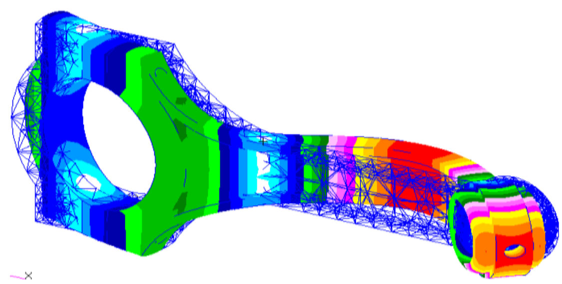


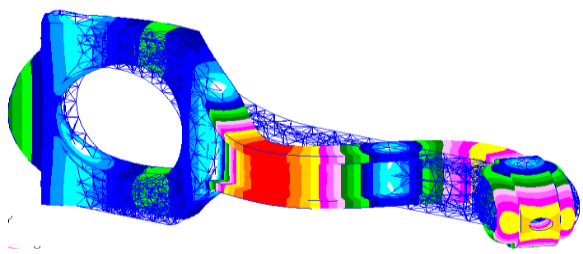
Fig 12: Stresses concentration versus mesh global length for TET10 to check mesh convergence.



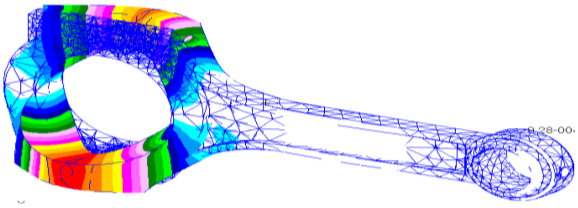
Mode 1, 19.11 Hz



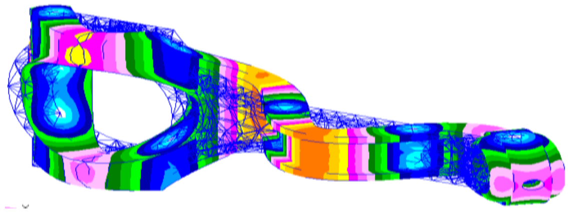
Mode 2, 61.21 Hz



Mode 3, 142.31 Hz



Mode 4, 210.24 Hz



Mode 5, 221.30 Hz

Fig 11: The mode shapes of the connecting rod.