THE INTERNATIONAL JOURNAL OF
SCIENCE & TECHNOLEDGE

Finite Element Analysis of Connecting Rod Using Aluminum Alloy-2014

Hitesh Vasudev
Assistant Professor, Punjab Institute of Technology, Hoshiarpur, Punjab, India
Amanjot Singh
Student, Punjab Institute of Technology, Hoshiarpur, Punjab, India
Harminder Singh
CAE Engineer, Futuretech Design, India

Abstract:
Aluminum alloys are used for various mechanical components where corrosion resistance, high thermal conductivity, toughness and light weight construction is required to reduce the inertia forces. In the present study aluminum alloy-2014 has been selected for the connecting rod as a material. Structural analysis was done by using NX- 7.5 software with NASTRAN solver in advanced simulation module. At the extreme ends, bearing pressure and piston load calculations were made by linear static analysis. Results of linear static analysis have shown that the maximum stress value i.e 142.28 MPa which is well below the ultimate tensile strength value i.e, 220 MPa of Aluminum alloy-2014. Further a negligible deformation of 0.179 mm at the piston end had occurred. By using aluminum alloys optimization and analysis should be done in future research work for connecting rod due to it's light weight construction, high ultimate tensile strength and toughness.

Keywords: *Connecting rod, FEA, Aluminum alloy-2014, Linear Static Analysis*

1. Introduction

Connecting rod is a major mechanical machine element of engine which is subjected to variable load. In order to increase it's strength various aluminum alloys are used due to their several features like corrosion resistance, Lightweight construction and toughness. Research of the connecting rod had started by webster in 1983. Now a days, Optimization and analysis of connecting rod is done by various researchers by using different materials. Alloys have introduced a new face to the society for the various applications in the field of engineering. In the present study finite element analysis of connecting rod using aluminum alloy-2014 was done. Aluminum alloy-2014 has been choosen for the study, which possess a features of high ultimate tensile strength. Dynamic load analysis and optimization of connecting rod was studied and their results showed that by using aluminium alloys weight and cost is reduced for a production forged steel connecting rod[1]. Design of the connecting rod of internal combustion engine, a topology optimization approach had been studied and their results showed that the modeling of connecting rod and finite element Analysis has been presented. Topology optimization were analyzed to the connecting rod and according to the results, it can be concluded that the weight of optimized design is 11.7% lighter and maximum stress also predicted lower than the initial design of connecting rod.[2] Study of Optimization and Cost Reduction by Finite Element Analysis of Connecting Rod Using Aluminum Composite had shown weight reduction of connecting rod[3]. Study of Analysis of Connecting Rod Using Analytical and Finite Element Method had shown that the minimum stresses among all loading conditions, were found at crank end cap as well as at piston end. So the material can be reduced from those portions, thereby reducing material cost[4]. Study of Optimizing the Design of Connecting Rod under Static and Fatigue Loading had shown reduction in the resulting inertial and centrifugal force which improves static and fatigue results [5].

2. Methodology

Methodology includes the preparation of 3-D MODEL OF CONNECTING ROD .3-D model is followed by meshing and the analysis of connecting rod

2.1. Modelling and Analysis

3-D model of connecting rod was developed in NX- 7.5 software with idealizing i.e removing of fillets and round faces for proper meshing. The Compressive and tensile forces on the piston pin bearing and crank pin bearing have been considered. Then, using Finite Element Analysis software NX- Advnced Simulation And using NASTRAN as Solver, analysis is done to determine the minimum and maximum stress values in the connecting rod. Crank end is fixed with pin constraint, which helps in free rotation but there is no linear movement of connecting rod. At the piston end, bearing constraint was applied and the user defined constraint defines the degree of freedom of connecting rod.

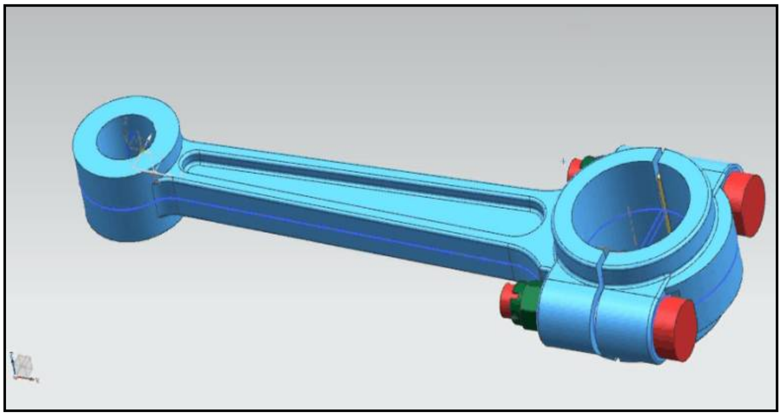

Figure 1:3-D model of connecting rod

2.2. Meshing of Connecting Rod

After the designing model has been established, meshing of connecting rod was done by fracturing the model into small nodes, process is known as meshing.

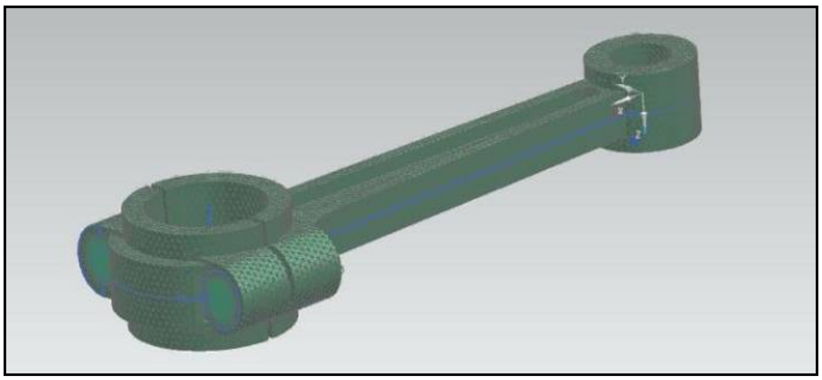

Figure 2: Meshed model of Connecting Rod

2.3. Input Parametres

			Crank End				Piston End					
Crank Angle	**Ang. Velocity**	**Ang. Acc.**	**F_{AX}**	**F_{AY}**	**Resultant**	**Direction**	**F_{BX}**	**F_{BY}**	**Resultant**	**Direction**	**Pressure Constant Force UDL – (MPa)**	
deg	rev/s	rev/s2	N	N	N	deg	N	N	N	deg	Crank End	Pis End
23	-21.3	3401	13579	-3237	13960	-13.4	-17265	2152	17398	-7.1	19.69	45.60
24	-21.2	3548	13477	-3354	13888	-14.0	-17128	2225	17272	-7.4	19.59	45.27
126	14.0	7731	7749	-3934	8691	-26.9	-5451	1688	5707	-17.2	12.26	14.96
360	-22.9	0	-8366	0	8366	0.0	4289	0	4289	0.0		
362	-22.9	296	-8406	-26	8406	0.2	4332	-71	4333	-0.9		
Acceleration at the crank end center is 8,509,792 mm/s2.												

Table 1: Inputs for FEA of connecting rod using dynamic analysis results at crankshaft speed of 4000 rev/min.

2.4. Aluminium Alloy-2014 Properties

Material	**Aluminum Alloy 2014**
Density[Kg/m^3]	2800
Young's Modulus (GPa)	72.4
Tensile Strength, Ultimate (MPa)	220
Shear Strength (MPa)	124
Poisson’s ratio	0.33

Table2: Aluminium Alloy-2014 Properties

3. Results and Discussions

Displacement, elemental stress and elemental strain were calculated on the input parametres, which are discussed in this portion of the this research paper.

3.1. Displacement

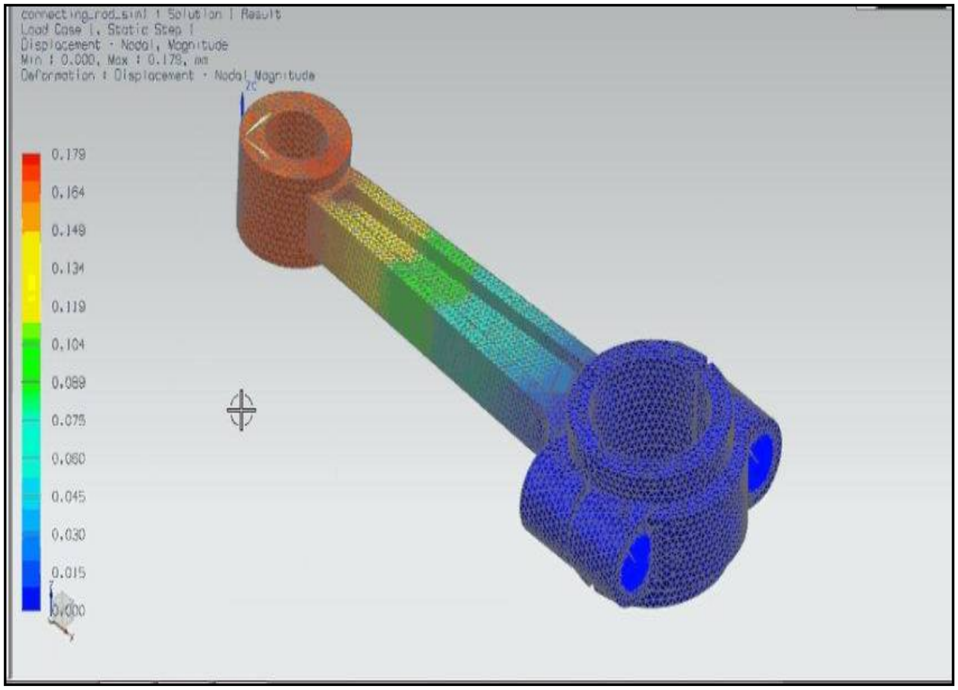

Figure 3: Displacement result of connecting rod

Figure shows results of Aluminum Alloy 2014 after analysis- Displacement is minimum at crank end, but near to maximum at piston end. Further a negligible deformation of 0.179 mm at the piston end had occurred.

3.2. Equivalent Stress

Equivalent Stresses are minimum at both the ends and moderate at shank. Results of linear static analysis have shown that the maximum stress value, i.e 142.28 MPa which is well below the ultimate tensile strength value i.e 220 MPa of Aluminum alloy-2014.

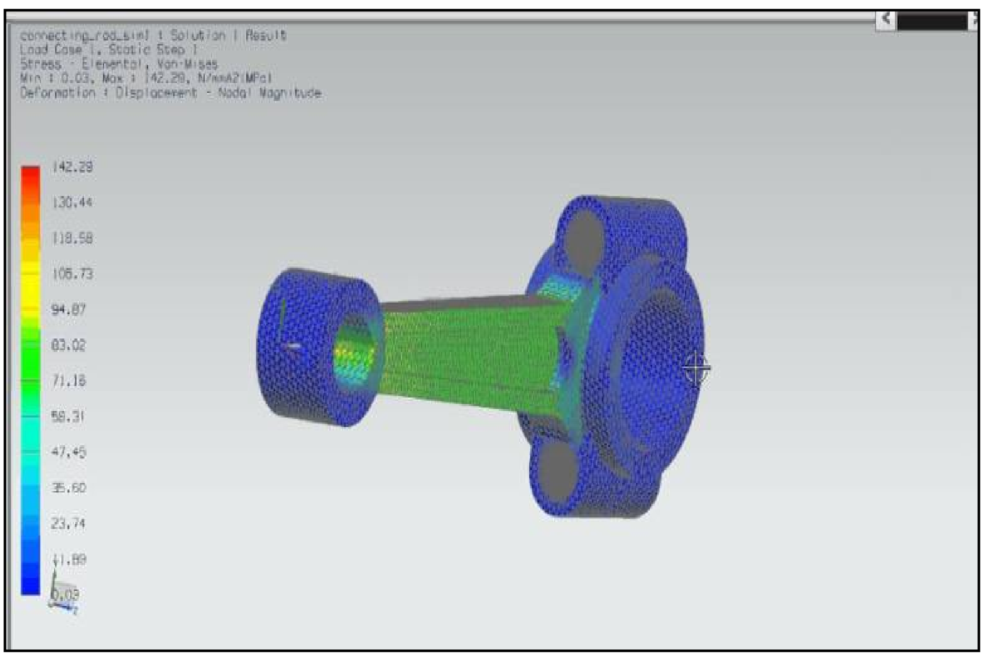

Figure 4: Elemental Stress

3.3. Elemental Elastic Strain

When an external stress is applied to a body, the body tends to pull itself apart. This induces strain in the body.

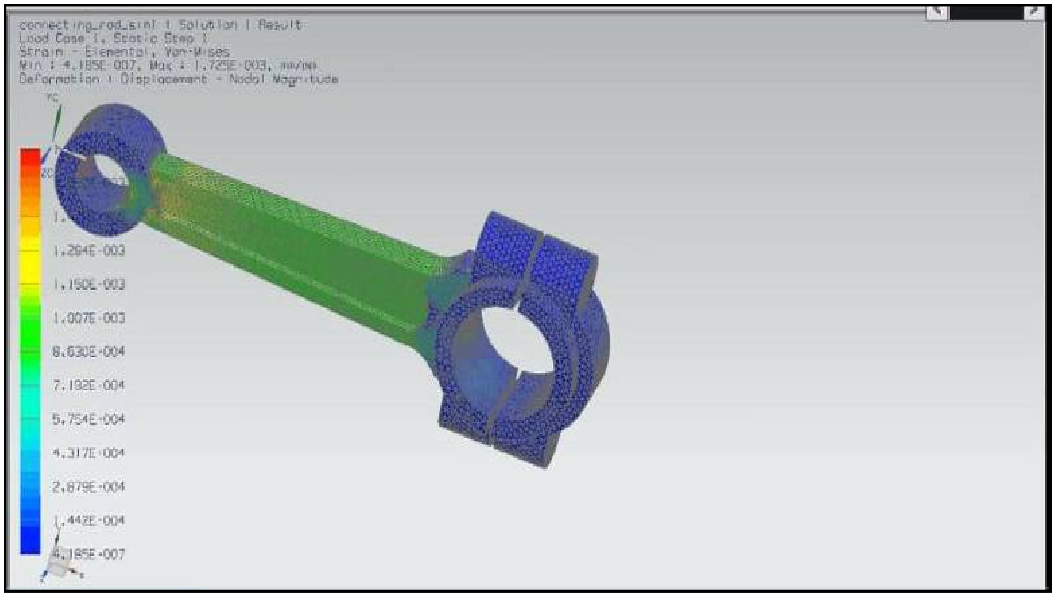

Figure 5: Elemental Strain

3.4. Results for Static Loading

PARAMETERS	RESULTS	
	MINIMUM	MAXIMUM
DISPLACEMENT (mm)	0.000	0.179
ELEMENTAL STRESS (MPa)	0.03	142.28
ELEMENTAL STRAIN (mm/mm)	4.185e.007	1.725e.003

Table 3: Maximum and Minimum Values of Stress Parameters

4. Conclusion

At the extreme ends, bearing pressure and piston load calculations were made by linear static analysis. Results of linear static analysis have shown that the maximum stress value, i.e 142.28 MPa which is well below the ultimate tensile strength value, i.e 220 MPa of Aluminum alloy-2014. Further a negligible deformation of 0.179 mm at the piston end had occurred. By using aluminum alloys optimization and analysis should be done in future research work for connecting rod due to it’s light weight construction, high ultimate tensile strength and toughness.

5. References

i. Pravardhan S. Shenoy and Ali Fatemi,2005,” Connecting Rod Optimization for Weight and Cost Reduction”,SAE International 2005-01-0987.
ii. M.S. Shaari, M.M. Rahman, M.M. Noor, K. Kadirgama Anda.K. Amirruddin,2010” Design of connecting rod of internal combustion engine: a topology optimization approach”, National Conference In Mechanical Engineering Research And Postgraduate Studies (2nd NCMER 2010)3-4 December 2010, Pp. 155-166.
iii. Naveen Lohan, Sandeep Nandal,2014,” Optimization and Cost Reduction by Finite Element Analysis of Connecting Rod Using Aluminum Composite”, International journal of engineering sciences & research technology,pp-17-24.
iv. N.P.Doshi, N.K.Ingole,2013, Analysis of Connecting Rod Using Analytical and Finite Element Method, International Journal of Modern Engineering Research, (IJMER) www.ijmer.com Vol.3, Issue.1, Jan-Feb. 2013 pp-65-68.
v. Om Parkash, Vikas Gupta, Vinod Mittal,2013,” Optimizing the Design of Connecting Rod under Static and Fatigue Loading” International Journal of Research in Management, Science & Technology ,Vol. 1; No. 1, June 2013,pp-39-43.
vi. G.M Sayeed Ahmed, Sirajuddin Elyas Khany, Syed Hamza Shareef,2014” Design, Fabrication and Analysis of a Connecting Rod with Aluminum Alloys andCarbon Fiber, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, Issue 10, October 2014,pp-1-14
vii. Glodováa. I, Liptákb.T, Bockoc. J (2014) “Usage of finite element method for motion and thermal analysis of a specific object in SolidWorks environment”, Procedia Engineering 96, pp 131-135.
viii. John A. ,Nidhi M.B. (2014) “Modelling and Analysis of an Automotive Bumper Used for a Low Passenger Vehicle”, International Journal of Engineering Trends and Technology (IJETT) – Volume 15 Number 7 , pp 344-353.

ix. Quesada. A, Gauchia.A, Álvarez-Caldas C., and Román J.L.S, (2014)" Material Characterization for FEM Simulation of Sheet Metal Stamping Processes", Hindawi Publishing Corporation Advances in Mechanical Engineering Volume 2014.
x. Shinde A., Prof. Thombare D., (2014) "Finite Element Method for Stress Analysis of Passenger Car Floor", Int. Journal of Engineering Research and Applications Vol. 4, Issue 4(Version 8), pp.38-42.
xi. Q. H. Ma, C. Y. Zhang , S. Y. Han and Z. T. Qin (2013) "Research on the Crash Safety of the Car Bumper Base on the Different Standards", International Journal of Security and Its Applications Vol.7, No.6 pp.147-154.
xii. Zhongyao Wu, Jianbo Cao, Qifen Gu, and Tianfeng Zhao,(2012) "Design for Novel Appearance of Mini Electric Car" Atlantis Press, Paris, France, pp 2279-2282.
xiii. NimbalkarV.V, Ghorpade R.R, Kulkarni S.S. (2012) "DIE CASTING DIE DESIGN FOR A HIGH VOLUME AUTOMOTIVE COMPONENT USING CAE TECHNIQUES FOR SIMULATION' International Journal of Advanced Engineering Research and Studies.
xiv. W. M. Wan Muhamad, E. Sujatmika, M.R. Idris, S.A. Syed Ahmad (2012) "An Optimization Analysis on an Automotive Component with Fatigue Constraint Using HyperWorks Software for Environmental Sustainability" International Scholarly and Scientific Research & Innovation, Vol. 6, pp 197-201.