

## FATIGUE ANALYSIS IN CONNECTING ROD USING ANSYS

Yogesh. B. Dupare<sup>1</sup>, Raju.B.Tirpude<sup>2</sup> and Akshay.Y.Bharadbhunje<sup>3</sup>

<sup>1</sup> Controller of Exam.Government Polytechnic, Nagpur

<sup>2</sup> LAE,Government Polytechnic, Nagpur

<sup>3</sup> LAE,Government Polytechnic, Nagpur

**ABSTRACT:-** In the present paper a case study of “Fatigue Analysis in Connecting Rod Using ANSYS” is carried out. In this paper, finite element analysis of single cylinder four stroke petrol engine using ANSYS. The work has to find out the stresses at various points on the connecting rod and the portion, which is more susceptible to failure and optimization of connecting rod. To evaluate the magnitude and location stresses in the existing connecting rod. This is of great interest to the auto manufactures that which is the portion of the connecting rod which mainly fails so that they can use various methods of hardening the specific area by using special hardening treatments.

In response to an increasing demand for fuel-economy, more weight reduction techniques have been proposed to create an optimum connecting rod design. The research aims to maximize weight savings in a connecting rod, without sacrificing the structural performances such as bending strength, buckling strength, and torsional stiffness. The focus of fatigue in ANSYS is to provide useful information to the design engineer when fatigue failure may be a concern. Fatigue results can have a convergence attached. A stress-life approach has been adopted for conducting a fatigue analysis. Several options such as accounting for mean stress and loading conditions are available.

**Keywords:** Finite element analysis (FEA), Connecting rod (CR), Computer Aided Design (CAD), Centre of Gravity (CG), Failure Index (FI), Factor of Safety (FOS).

### I. INTRODUCTION

This submission shows the implementation of the FEM software for the assessment of the strength and distortion characteristics of a CR. A combination of axial and bending stresses acts on the rod in operation. The axial stresses are produced due to cylinder gas pressure (compressive only) and the inertia force arising in account of reciprocating action (both tensile as well as compressive), where as bending stresses are caused due to the centrifugal effects. Structural systems of connecting rod can be easily analyzed using Finite Element techniques. Firstly a proper Finite Element Model is developed using CAD software (Pro/Engineer, HyperMesh). Then the Finite element analysis is done to determine the von Mises stresses in the existing CR for the given loading conditions using FEA software ANSYS.

It is estimated that 50-90% of structural failure is due to fatigue, thus there is a need for quality fatigue design tools. However, at this time a fatigue tool is not available which provides both flexibility and usefulness comparable to other types of analysis tools. This is why many designers and analysts use "in-house" fatigue programs which cost much time and money to develop. It is hoped that these designers and analysts, given a proper library of fatigue tools could quickly and accurately conduct a fatigue analysis suited to their needs.

## II. CASE STUDY

### 2.1 FE MODELING OF CONNECTING ROD

In geometric modeling, the graphic image of an object is generated on the graphic screen of the system by inputting three types of commands to the computer. A solid model of the connecting rod was generated using Pro/E Wildfire 3.0.

### 2.2 STRESS ANALYSIS OF CONNECTING ROD

#### 2.2.1 MESH GENERATION

Finite element mesh was generated using quadratic tetrahedral elements with element lengths. First the connecting rod solid model is imported from PRO/Engineer wildfire 3.0 software in ANSYS WORKBENCH 9.0 as shown in fig.2.1. Once you have ensured regular shapes and assigned the appropriate divisions, generating the mesh is easy. Just press the Mesh button in the Mesh Tool, then press [Pick All] in the picker or choose the desired entities. After satisfying above criteria, mesh generation has been done on each surface of component, then check for free edges of elements (equivalence of nodes), Apply quality checks for each surface and editing of meshing if it fails, Go on meshing for adjacent surfaces and check for nodal connectivity, any duplication, Finish the meshing of each surface of components.

The meshing model satisfying all quality requirements is prepared, final meshed model is shown in Fig. 2.2.

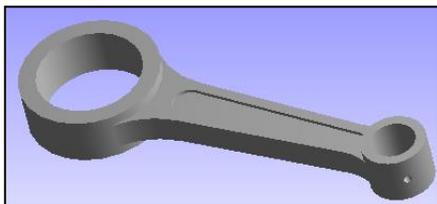


Fig. 2.1 Import model in Ansys Workbench 9.0.

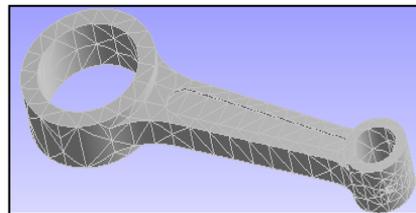


Fig. 2.2 Meshed model of Connecting Rod

#### 2.2.2 BOUNDARY AND LOADING CONDITION

Finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. You can apply loads on the model in a variety of ways in the ANSYS program. The word *loads* in ANSYS terminology includes boundary conditions *and* externally or internally applied forcing functions, as illustrated Fig 2.3.

The load analysis was carried out to obtain the loads acting on the CR at any given time in the loading cycle and to perform FEA. Most investigators have used static axial loads for the design and analysis of CR. However, lately, some investigators have used inertia loads (axial load varying along the length) during the design process. As a result, FEA was carried out under axial static load with no dynamic/inertia loads. The results of the above mentioned analyses are presented and discussed in this chapter with a view to use them for optimization. Static FEA results showed high stresses in the regions of the transitions to the shank at the crank end and piston pin end, the oil hole, and the cap.



Fig. 2.3 FEA model of the connecting rod with axial compressive load at the piston pin end and crank end restrained.

## 2.3 FATIGUE ANALYSIS

A fatigue analysis can be separated into 3 areas: materials, analysis, and results evaluation. Each area will be discussed in more detail below:

### 2.3.1 MATERIALS

A large part of a fatigue analysis is getting an accurate description of the fatigue material properties. Since fatigue is so empirical, sample fatigue curves are included only for structural steel and aluminum materials. These properties are included as a guide only with intent for the user to provide his/her own fatigue data for more accurate analysis. In the case of assemblies with different materials, each part will use its own fatigue material properties just as it uses its own static properties (like modulus of elasticity).

#### Stress-life Data Options/Features

- Fatigue material data stored as tabular alternating stress vs. life points.
- The ability to define mean stress dependent or multiple r-ratio curves if the data is available.
- Options to have log-log, semi-log, or linear interpolation.
- Ability to graphically view the fatigue material data
- The fatigue data is saved in XML format along with the other static material data.
- Fig. 2.4 shows a user editing SN curves in ANSYS.
- Table 1 is a screen shot showing a user editing fatigue data in ANSYS.

### 2.3.2 ANALYSIS

Fatigue results can be added before or after a stress solution has been performed. To create fatigue results, a fatigue tool must first be inserted into the tree. This can be done through the solution toolbar or through context menus. The details view of the fatigue tool is used to define the various aspects of a fatigue analysis such as loading type, handling of mean stress effects and more. As seen in Fig. 2.5, a graphical representation of the loading and mean stress effects is displayed when a fatigue tool is selected by the user. This can be very useful to help a novice understand the fatigue loading and possible effects of a mean stress.

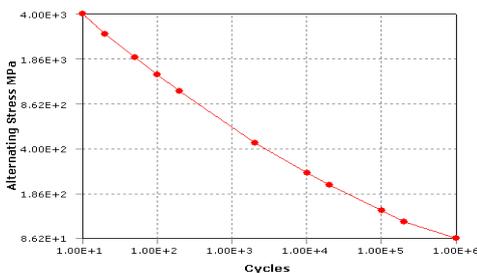


Fig. 2.4 User Editing SN curves in ANSYS.

Table 1. User Editing Fatigue Data in ANSYS.

Cycles	Alternating Stress
10.0	3,999.0 MPa
20.0	2,827.0 MPa
50.0	1,896.0 MPa
100.0	1,413.0 MPa
200.0	1,069.0 MPa
2,000.0	441.0 MPa
10,000.0	262.0 MPa
20,000.0	214.0 MPa
100,000.0	138.0 MPa
200,000.0	114.0 MPa
1,000,000.0	86.2 MPa

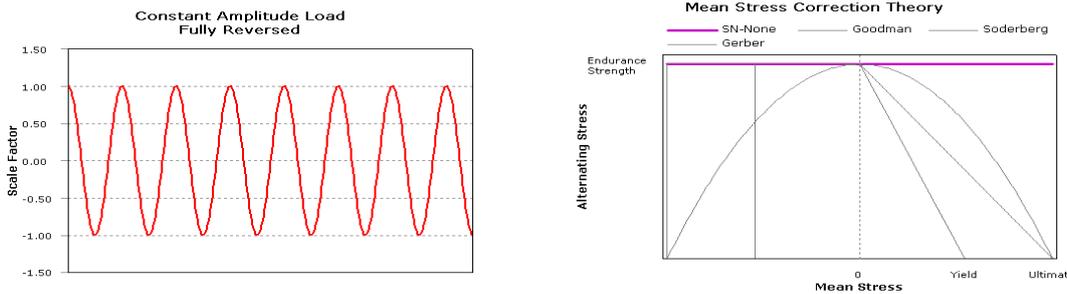


Fig. 2.5 Fatigue tool information page in ANSYS

### 2.3.2.1 LOADING

Fatigue, by definition, is caused by changing the load on a component over time. Thus, unlike the static stress safety tools, which perform calculations for a single stress, fatigue damage occurs when the stress at a point changes over time. ANSYS can perform fatigue calculations for either constant amplitude loading or proportional non-constant amplitude loading. A scale factor can be applied to the base loading if desired. This option, located under the “Loading” section in the details view, is useful to see the effects of different finite element load magnitudes without having to re-run the stress analysis.

- **Constant amplitude, proportional loading:** This is the classic, “back of the envelope” calculation. Loading is of constant amplitude because only 1 set of finite element stress results along with a loading ratio is required to calculate the alternating and mean stress.
- **Non-constant amplitude, proportional loading:** In this case, again only 1 set of results are needed, however instead of using a single load ratio to calculate the alternating and mean stress, the load ratio varies over time.

Several sample load histories can be found in the “Load Histories” directory under the “Engineering Data” folder. Setting the loading type to “History Data” in the fatigue tool details view specifies non-constant amplitude loading. Several analysis options are available for nonconstant amplitude loading, when referring to fig. 2.6.

Import geometry and apply boundary conditions. Apply loading corresponding to the maximum developed load of 4319 N as already done, shown in Fig.2.3. Insert fatigue tool. Specify fully reversed loading to create alternating stress cycles. Specify that this is a stress-life fatigue analysis. No mean stress theory needs to be specified since no mean stress will exist (fully reversed loading). Specify that Von-Mises stress (Fig. 2.7) will be used to compare against fatigue material data.

### 2.3.2.2 LOAD EFFECTS

Fatigue material tests are usually conducted in a uniaxial loading under a fixed or zero mean stress state. It is cost-prohibitive to conduct experiments that capture all mean stress, loading, and surface conditions. Thus, empirical relations are available if the fatigue data is not. Specify a modification factor since material data represents a polished specimen and the in-service component is cast. Perform stress and fatigue calculations (Solve command in context menu). Plot factor of safety for a design life of 1,000,000 cycles. Find the sensitivity of available life with respect to loading. Determine multiaxial stress state (uniaxial, shear, biaxial, or mixed) at critical life location by inserting “biaxiality indicator” into fatigue tool. The stress state near the critical location is not far from uniaxial, which gives an added measure of confidence since the material properties are uniaxial.

- **Mean Stress correction.** If the loading is other than fully reversed, a mean stress exists and should be accounted for. Methods for handling mean stress effects can be found in the “Options” section. If

experimental data at different mean stresses or r-ratio's exist, mean stress can be accounted for directly through interpolation between material curves. If experimental data is not available, several empirical options may be chosen including Gerber, Goodman and Soderberg theories which use static material properties (yield stress, tensile strength) along with S-N data to account for any mean stress when referring to fig.2.5.

- **Multiaxial Stress Correction.** Experimental test data is uniaxial whereas stresses are usually multiaxial. At some point stress must be converted from a multiaxial stress state to a uniaxial one. Von-Mises, Max shear, Maximum principal stress, or any of the component stresses can be used as the uniaxial stress value in stress analysis result. In addition, a "signed" Von-Mises stress may be chosen where the Von-Mises stress takes the sign of the largest absolute principal stress. This is useful to identify any compressive mean stresses since several of the mean stress theories treat positive and negative mean stresses differently. Setting the "Stress Component" is done in the Options section in the fatigue tool detail view.

### 2.3.3 RESULTS OUTPUT

Several results for evaluating fatigue are available to the user. Some are contour plots of a specific result over the model while others give information about the most damaged point in the model (or the most damaged point in the scope of the result). Outputs include fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity as shown in Fig. 2.7 through Fig. 2.11.

- A contour plot of available life over the model. This result can be over the whole model or scoped to a given part or surface. This result contour plot shows the available life for the given fatigue analysis. If loading is of constant amplitude, this represents the number of cycles until the part will fail due to fatigue. If loading is non-constant, this represents the number of loading blocks until failure. Thus if the given load history represents one month of loading and the life was found to be 120, the expected model life would be 120 months.
- A contour plot of the fatigue damage at a given design life. Fatigue damage is defined as the design life divided by the available life. This result may be scoped. The default design life may be set through the Control Panel (Table 3).
- A contour plot of the factor of safety with respect to a fatigue failure at a given design life. The maximum FS reported is 15. Like damage and life, this result may be scoped. This calculation is iterative for nonconstant amplitude loading and may substantially increase solve time (Table 3).
- A stress biaxiality contour plot over the model. As mentioned previously, material properties are uniaxial but stress results are usually multiaxial. This result gives the user some idea of the stress state over the model and how to interpret the results. Biaxiality indication is defined as the principal stress smaller in magnitude divided by the larger principal stress with the principal stress nearest zero ignored. A biaxiality of zero corresponds to uniaxial stress, a value of  $-1$  corresponds to pure shear, and a value of  $0.97$  corresponds to a pure biaxial state (Table 3). From the sample biaxiality plot shown below, most of the model is under a pure shear or uniaxial stress. This is expected since a simple torque has been applied at the top of the model. When using the biaxiality plot along with the safety factor plot above, it can be seen that the most damaged point occurs at a point of nearly pure shear. Thus it would be desirable to use S-N data collected through torsional loading if available. Of course collecting experimental data under different loading conditions is cost prohibitive and not often done.
- A fatigue sensitivity plot. This plot shows how the fatigue results change as a function of the loading at the critical location on the model. This result may be scoped to parts or surfaces. Sensitivity may be found for life, damage, or factory of safety. The user may set the number of fill points as well as the load variation limits.

Table 2 Fatigue Definition:

Name	Fatigue Strength Factor	Type	Scale Factor	Analysis Type	Stress Component	Infinite Life
<i>Fatigue Tool</i>	1.0	Fully Reversed	1.0	SN-None	Equivalent (Von-Mises)	$1.0 \times 10^9$

Table 3 Fatigue Results

Name	Figure	Scope	Type	Design Life	Minimum	Maximum	Alert Criteria
<i>Life</i>	None	Model	Life		1,000,000.0	1,000,000.0	None
<i>Damage</i>	None	Model	Damage	$1.0 \times 10^9$	1,000.0	1,000.0	None
<i>Safety Factor</i>	5.21	Model	Safety Factor	$1.0 \times 10^9$	1.13	15.0	None
<i>Biaxiality Indication</i>	5.22	Model	Biaxiality Indication	--	-1.0	0.97	None
<i>Equivalent Alternating Stress</i>	5.23	Model	Equivalent Reversed Stress	--	0.01 Mpa	76.22 Mpa	None

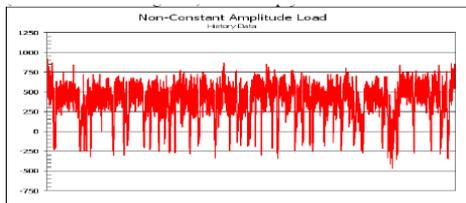


Fig. 2.6 Chart of loading history

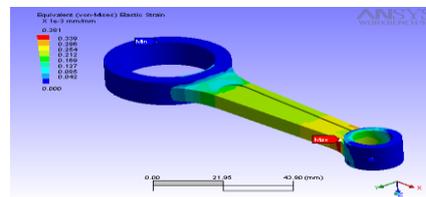


Fig. 2.7 Equivalent (Von-Mises) Elastic Strain.

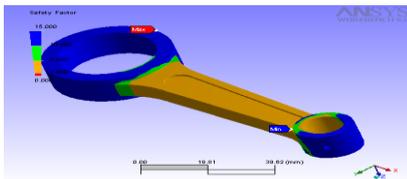


Fig. 2.8 Safety Factor.

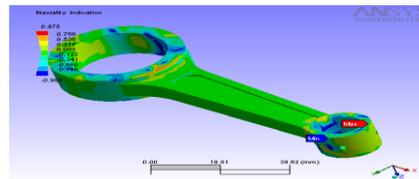


Fig. 2.9 Biaxiality Indication.

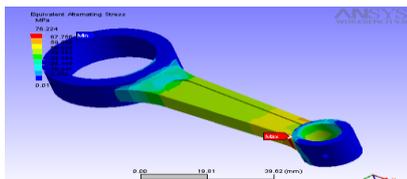


Fig. 2.10 Equivalent Alternating Stress.

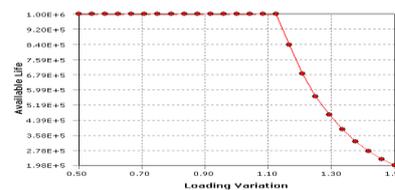


Fig. 2.11 Fatigue Sensitivity

### III. CONCLUSION

1. There is considerable difference in the structural behavior of the connecting rod between axial fatigue loading. The results obtained with the analysis tool are quite comfortable and can be used to optimize the model.
2. The Optimization carried out in analysis gives deep insight by considering optimum parameter for suggestion of modification in the existing connecting rod.
3. Fatigue strength was the most significant factor (design driving factor) in the optimization of this connecting rod.
4. The parameter consideration for optimization are its 20 % reduction in weight of connecting rod, while reducing the weight, the static strength, fatigue strength, and the buckling load factor were taken into account.

5. The optimized geometry is 20% lighter than the current connecting rod. PM connecting rods can be replaced by fracture splittable steel forged connecting rods with an expected weight reduction of about higher than existing connecting rod, with similar or better fatigue behavior.
6. The stress multiaxiality is high, especially at the critical region of the crank end transition. Therefore, multiaxial fatigue analysis is needed to determine fatigue strength. Due to proportional loading, equivalent stress approach based on von Mises criterion can be used to compute the equivalent stress amplitude. Outputs include fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity.

## REFERENCES

- [1] Pravardhan S. Shenoy, "Dynamic load analysis and optimization of connecting rod", 2004, Master's thesis, University of Toledo.
- [2] Sarihan, V. and Song, J., 1990, "Optimization of the Wrist Pin End of an Automobile Engine Connecting Rod with an Interference Fit," *Journal of Mechanical Design*, Transactions of the ASME, Vol. 112, pp. 406-412.
- [3] ANSYS WORKBENCH 9.0 Reference Manual.
- [4] Pro/E Wildfire 3.0 Reference Manual.
- [5] "DESIGN DATA", Faculty of Mech. Engg., P.S.G. College of Technology, Coimbatore,
- [6] R. S. Khurmi and J. K. Gupta, "A Textbook of Machine Design", 2005, S chand and Comp. Ltd., Delhi.
- [7] Hero Honda Splendor Company Manual.
- [8] Hancq, D.A., Walters, A.J., Beuth, J.L., "Development of an Object- Oriented Fatigue Tool", *Engineering with Computers*, Vol 16, 2000, pp. 131-144.
- [9] Pai, C. L., 1996, "The shape optimization of a connecting rod with fatigue life constraint," *Int. J. of Materials and Product Technology*, Vol. 11, No. 5-6, pp. 357-370.
- [10] Sonsino, C. M. and Esper, F. J., 1994, "Fatigue Design for PM Components," European Powder Metallurgy Association (EPMA).



