MODELING & ANALYSIS OF CRANKSHAFT

Mr. K.V.Naveen Kumar  
B.Tech  
Department of Mechanical Engineering  
Marri Laxman Reddy Institute of Technology & Management  
Hyderabad, T.S  
naveenmec@yahoo.co.in

Mr. T.Balaji Gupta  
M.Tech  
Department of Mechanical Engineering  
Marri Laxman Reddy Institute of Technology & Management  
Hyderabad, T.S

Mr. D.Muppala  
M.Tech  
Department of Mechanical Engineering  
Marri Laxman Reddy Institute of Technology & Management  
Hyderabad, T.S  
vd.muppala@yahoo.com

ABSTRACT

Among the many important parts of the power transmission system of an engine the Crank shaft is one. It is a structural member of the engine, which takes in the reciprocating motion from the piston and converts it into rotary motion.

The crank shaft takes the power from piston which is generated due to combustion process inside the combustion chamber of the cylinder. During the power transmission process the load acts at a particular crank angle to the max and hence the connecting rod is analyzed for the stress developed, due to load conditions and the changes mentioned.

In this project the stress analysis and modal analysis of a 6-cylinder crankshaft are discussed using finite element method. Three-dimension models of 480 diesel engine crankshaft and crank throw were created using CATIA software. The finite element analysis (FEM) software ANSYS is used to analyse the modal, Harmonic and stress status of the crankshaft. The maximum deformation, maximum stress point and dangerous areas are found by the stress analysis of crank shaft. The relationship between the frequency and Amplitude is explained by the modal & Harmonic analysis of crankshaft.

1.0 INTRODUCTION:

Crankshaft is one of the most important moving parts in internal combustion engine. It must be strong enough to take the downward force of the power stroked without excessive bending. So the reliability and life of internal combustion engine depend on the strength of the crankshaft largely.

Crankshaft is a large component with a complex geometry in the engine, which converts the reciprocating displacement of the piston to a rotary motion with a four link mechanism. Since the crankshaft experiences a large number of load cycles during its service life, fatigue performance and durability of this component has to be considered in the design process. Design developments have always been an important issue in the crankshaft production industry, in order to manufacture a less expensive component with crankshaft production industry, in order to manufacture a less expensive component with requirements. These improvements result in lighter and smaller engines with better fuel efficiency and higher power output.

Strength calculation of crankshaft becomes a key factor to ensure the life of engine. Beam and space frame model were used to calculate the stress of crankshaft usually in the past. But the number of node is limited in these models. With the development of computer, more and more design of crankshaft has been utilized finite element method (FME) to calculate the stress of crankshaft.

1.1 Function of Crankshafts in IC Engines

The crankshaft, connecting rod, and piston constitute a four bar slider-crank. The crankshaft, connecting rod, and piston constitute a four bar slider-crank a rotary motion. Since the rotation output is more practical and applicable for input to other devices, the concept design of an engine is that the output would be rotation. In addition, the linear displacement of an engine is not smooth, as the displacement is caused by the combustion of gas in the combustion chamber. Therefore, the displacement has sudden shocks and using this input for another device may cause damage to it. The concept of using crankshaft is to change these sudden displacements to a smooth rotary output, which is the input to many devices such as generators, pumps, and compressors. It should also be mentioned that the use of a flywheel helps in smoothing the shocks.

1.2 Finite Element Modelling

Since the crankshaft has a complex geometry for analysis, finite element models have been considered to give an accurate and reasonable solution whenever laboratory testing is not available. Uchida and Hara (1984) used a single throw FEM model in the extrapolation of the experimental equation in their study.

A theoretical study followed by experimental results was conducted by Guagliano et al. (1993) to calculate the stress concentration factor in a diesel engine crankshaft. They conducted experimental tests by mounting strain gages at high stress concentration areas (crank fillet). A three
3.0 INTRODUCTION TO F.E.M.

The finite element method combines in an elegant way the best features of the two approximate methods of analysis (viz.,) Functional approximation and Finite differences method .In particular finite element method can be explained through physical concept and hence it is most appealing to the engineer. And the method is amenable to systematic computer program and offers scope for application to a wide range of analysis problems .The basic concept is that a body or a structural may be divided into small elements of finite dimensions called finite elements. This process of dividing a continuum into finite elements is known as discretization. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called nodes or nodal points. Similar concept is used in finite difference method.

3.1 CONCEPT OF A FINITE ELEMENT

The finite element method is based upon the general principle known as going from part to whole .In engineering; problems may come which cannot be solved in closed form, which is as a whole. Therefore, we consider the physical medium as an assemblage of many small parts. Analysis of the basic part forms the first step towards a solution.

This notion which in mathematical rather than physical, does not consider the body or the structure to be sub-divided into separate parts that are re-assembled in the analysis procedure. Instead of that the continuum is zoned into regions by imaginary lines or planes inscribed on the body. Using this concept, variation-procedure is applied in the analysis of the continuum by assuming a patchwork of solution or displacement models each of which applies to a single region.The first decision the engineer must take is to select the shape or
configuration of the basic element to be used in the analysis. This choice depends upon the geometry of the body or structure and also upon the number of independent space co-ordinates necessary to describe the problem. The finite element usually has a simple 1-D, 2-D or 3-D configuration.

3.2 CRITERION OF FINITE ELEMENT SELECTION:
Element to be used is dependent on physical problem itself. Following are important considerations in the selection of type of element.
1. Number of degrees of freedom required.
2. Expected accuracy.
3. Easy in designing the necessary equation.
4. Degree to which the physical structures can be modeled without approximation.

3.3 TYPES OF ELEMENTS:
1. Bar, Truss and beam elements.
2. Plane stress and plane strain elements.
3. Axisymmetric elements.
4. Plate bending elements.
5. Three dimensional elements.

Finite element analysis became an indispensable tool in engineering analysis and design.

3.4 FORMULATION OF CONTINUUM ELEMENTS:
For a continuum finite element, it is most case effective to calculated directly the element matrices corresponding to the global degrees of freedom. However, we shall first present the formulation of the matrices that correspond to the element local degrees of freedom because additional consideration may be necessary. when the element matrices that correspond to the global degrees of freedom are calculated directly. In the following we consider the derivation of the element matrices of straight truss elements; two-dimensional plane stress, plane strain, and axisymmetric elements; and 3-D elements that all have a variable number of nodes.

3.5 FORMULATION OF STRUCTURAL ELEMENTS:
The concept of geometry and displacement interpolations that have been employed in the formulation of two-and 3-D continuum elements can also be employed in the evaluation of beam, plate and shell structural element matrices. However, where as in the formulation of the continuum elements the displacement u, v, w (whichever are applicable) are interpolated in terms of nodal point displacements of the same kind, in the formulation of structural elements, the displacements, u, v, and w are interpolated in terms of mid-surface displacement and rotations. This procedure corresponds in essence to a continuum iso parametric element formulation with displacement constraints. In addition there is of course the major assumption that the stress normal to the mid-surface is zero. The structural elements are for these reasons appropriately called degenerate iso parametric elements, but frequently still referred them simply as iso parametric elements.

3.6 SOLUTION STEPS OF DYNAMIC PROBLEM:
In dynamic analysis problems, the displacements, velocities, strains, stresses and loads are all time dependant. The procedure involved in deriving the FE equations of a dynamic problem can be stated by following steps.
1) Idealize the body into finite elements
Assume the displacement model of element ‘e’

3.7 Optimization of Crankshafts with Geometry, Material, Manufacturing, and Cost Considerations
Crankshaft is among large volume production components in the internal combustion engine industry. Weight and cost reduction of this component will result in high cost savings. Weight reduction of a crankshaft will also increase the fuel efficiency of the engine.

Fig: Dynamic Load Analysis of the Crankshaft
The crankshaft experiences a complex loading due to the motion of the connecting rod, which transforms two sources of loading to the crankshaft. The main objective of this study was the optimization of the forged steel crankshaft which requires accurate magnitude of the loading on this component that consists of bending and torsion. The significance of torsion during a cycle and its maximum compared to the total magnitude of loading should be investigated to see if it is essential to consider torsion during loading or not. In addition, there was a need for obtaining the stress variation during a loading cycle and this requires FEA over the entire engine cycle.

3.9 FEA with Dynamic Loads

There are two different approaches for applying the loads on the crankshaft to obtain the stress time history. One method is to run the FE model many times during the engine cycle or at selected times over 720° by applying the magnitude of the load with its direction in a way that the loading could define the stress-time history of the component. Another approach to obtain stresses at different locations at different times during a cycle is by superposition of the basic loading conditions. This involves applying a unit load in the basic conditions and then scaling the stresses from each unit load according to the dynamic loading. Then similar stress components are added together. In this study both approaches were used for the engine speed of 3600 rpm to verify that results from both approaches are the same. After verification of results, the superposition approach was used by developing a code in Excel spread sheet to perform the necessary calculation and obtain the results for the stresses at different crankshaft angles. magnitude and direction of these two loads.

3.10 Stress Analysis and FEA:

The accuracy of the model and explains the simplifications that were made to obtain an efficient FE model. Mesh generation and its convergence are discussed. Using proper boundary conditions and type of loading are important since they strongly affect the results of the finite element analysis. Identifying appropriate boundary conditions and loading situation are also discussed. Finite element models of two components were analyzed; the cast iron crankshaft and the forged steel crankshaft. Since these two crankshafts are from similar engines, the same boundary conditions and loading were used for both. This facilitates proper comparison of this component made from two different manufacturing processes. The results of finite element analysis from these two crankshafts are discussed in this chapter. Above mentioned FE models were used for dynamic analysis considering the boundary conditions according to the mounting of the crankshafts in the engine.

3.11 Finite Element Modelling

Finite element modelling of any solid component consists of geometry generation, applying material properties, meshing the component, defining the boundary constraints, and applying the proper load type. These steps will lead to the stresses and displacements in the component. In this study, similar analysis procedures were performed for both forged steel and cast iron crankshafts.

4.0 RESULTS AND DISCUSSION

CRANK SHAFT MATERIALS

<table>
<thead>
<tr>
<th>MATERIALS</th>
<th>YOUNG'S MODULUS (MPA)</th>
<th>POISSON RATIO</th>
<th>DENSITY (Kg/m3)</th>
<th>YIELD STRESS (MPA)</th>
<th>ULTIMATE STRESS (MPA)</th>
</tr>
</thead>
<tbody>
<tr>
<td>42CrMn</td>
<td>210E3</td>
<td>0.3</td>
<td>7.9E3/7.9E9</td>
<td>930</td>
<td>1080</td>
</tr>
<tr>
<td>SAE1050(CS)</td>
<td>190E3</td>
<td>3</td>
<td>8.03E3/8.0E3</td>
<td>365.4</td>
<td>636</td>
</tr>
<tr>
<td>SAE4140(AS)</td>
<td>200E3</td>
<td>3</td>
<td>7.7E3/7.7E9</td>
<td>417</td>
<td>655</td>
</tr>
</tbody>
</table>

As per FEM base only one part of crank is required for calculation.
The main basic model with its complete parts meshed is as shown below:

MAIN MESHED PART:
The meshed part which is to be analysed as shown below:

MATERIAL: 42CRMN-C1
MODAL ANALYSIS:
1ST MODE SHAPE: The maximum stress obtained for the natural frequency of 98.539Hz is 13.273

2ST MODE SHAPE: The maximum stress obtained for the natural frequency of 108.137Hz is 11.531 MPA

3ST MODE SHAPE: The maximum stress obtained for the natural frequency of 208.961 HZ is 17.81 MPA

STATIC ANALYSIS:
MESHED MODEL: The meshed model for a single crank is as shown figure:
DEFORMATION: The deformation obtained for the model due to the natural frequency applied on the model is as shown in the figure:

VON- MISES STRESSES: The von mises stresses developed in the material due to the deformation generated in the body is as shown in the figure:

STRESSES OBTAINED IN DIFFERENT DIRECTIONS:
X-Direction: The maximum stress obtained in the X-Direction is 360.654 MPa

Y-Direction: The maximum amount of stress obtained in Y-Direction is 151.305 MPa

Z-Direction: The maximum amount of stress obtained in the Z-Direction is 134.488 MPa

HARMONIC ANALYSIS: The maximum amount of stress developed due to the peak displacement is 528.322 MPa

Graph: The basic frequency vs amplitude graph is as shown below:
The von misses stress graph is as shown below:

MATERIAL: SAE1050(CS)

MODAL ANALYSIS:

BASIC COMPLETE MODEL: The main basic model with its complete parts meshed is as shown below:

MAIN MESHED PART:
The meshed part which is to be analysed as shown below:

RESULTS:

HARMONIC RESULTS TABLE:

<table>
<thead>
<tr>
<th>Type</th>
<th>42CRMN-C1</th>
<th>SAE1050(CS)</th>
<th>SAE4140(AS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformatio n</td>
<td>0.32</td>
<td>0.25</td>
<td>0.45</td>
</tr>
<tr>
<td>Stress</td>
<td>528.322</td>
<td>550.593</td>
<td>503.385</td>
</tr>
</tbody>
</table>

MODEL ANALYSIS (NATURAL FREQUENCY) TABLE:

<table>
<thead>
<tr>
<th>S.N</th>
<th>42CRMN-C1</th>
<th>SAE1050(CS)</th>
<th>SAE4140(AS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>98.539</td>
<td>92.967</td>
<td>97.405</td>
</tr>
<tr>
<td>2</td>
<td>108.14</td>
<td>102.02</td>
<td>106.89</td>
</tr>
<tr>
<td>3</td>
<td>208.96</td>
<td>197.15</td>
<td>206.56</td>
</tr>
<tr>
<td>4</td>
<td>254.29</td>
<td>239.91</td>
<td>251.36</td>
</tr>
<tr>
<td>5</td>
<td>282.05</td>
<td>266.10</td>
<td>278.80</td>
</tr>
<tr>
<td>6</td>
<td>311.79</td>
<td>294.16</td>
<td>308.20</td>
</tr>
<tr>
<td>7</td>
<td>394.05</td>
<td>371.77</td>
<td>389.51</td>
</tr>
<tr>
<td>8</td>
<td>456.15</td>
<td>430.36</td>
<td>450.90</td>
</tr>
<tr>
<td>9</td>
<td>481.26</td>
<td>454.05</td>
<td>475.72</td>
</tr>
<tr>
<td>10</td>
<td>515.17</td>
<td>486.04</td>
<td>509.24</td>
</tr>
</tbody>
</table>

CONCLUSION:

1. By varying material properties of crank shaft dynamic analysis is done.
2. As per above results, 42CRMN-C1 material Von Misses stress values are below yield stress. Remaining Material’s
3. Von Misses stresses are above yield stress of material 42CRMN-C1.
4. Dynamic loading is done on the crank shaft of 6-cylinder Engine and analysis is made on it as Modal & Harmonic Analysis
5. From Model analysis of above results deformation and Natural frequency for material 42CRMN-C1 is observed. The deformation obtained is low and the natural frequency is more as compared to other materials.

5. References

http://www.castsolutions.com/,
http://www.engineersedge.com/,
http://www.steel.org/,
http://www.wikipedia.org/,
http://www.makinamuhendisi.com/,