IJASRM
International Journal of Advanced Scientific Research and Management, Volume 3 Issue 11, Nov 2018
www.ijasrm.com
ISSN 2455-6378

Natural Frequency and Modal Analysis of a Bolted Joint Test Rig for Bending and Axial Loading

Vishal Achwal[1], Rahul Lodha[2] , S Dravid [3]

[1]Research Scholar, Mechanical Engineering Department, Mewar University, Chittorgarh, Rajasthan, India

[2]Associate Professor, Mechanical Engineering Department, Faculty of Engineering and Technology, Mewar University, Chittorgarh, Rajasthan, India

[3]Associate professor , Mechanical Engineering Department, SDBCT, Indore , MP, India

Abstract

Research study includes modal analysis studies for a bolted joint test rig with bending and axial loading. Machinery vibration problems, e.g. bolt loosening, may get more severe if the operational frequency of the systems matches with any of the natural frequencies of the system. Such problems can be controlled by carrying out modal and harmonic analysis of the system. The test rig considered in this includes eight components in it. Various components of the system have been modelled in the CAD environment. These components are then assembled to get the overall CAD model of the experimental bolted joint test rig. Modal analysis of flexible components of the system has been carried out. Modal analysis helps in identifying resonance regions. Such primary analysis of system plays important role in studying loosening of bolted joints as well as in harmonic of the dynamic systems.

Key Words: *Bolted Joints, Natural Frequency, Modal Analysis.*

1. Introduction

A mechanical fastener is a device which is used to join two or more objects together mechanically. For the simulation and experiment work on the analysis in a single lap bolted joint under bending loads, FEM commercial packages are used to develop a three-dimensional finite element model of a bolted joint. In this simulation, a detailed discussion is done on different methods in modeling the contact between the joint, which affects the efficiency of the models. Testing was done by using a four-point bending load type in both the simulation and experiment works. The comparison of the results from both simulation and experiment were done and this comparison shows good agreement. Various factors which potentially influenced the variation of the results were noted. Finally critical areas were identified and confirmed with the help of stress distribution results from simulation. [1] Mechanical components are commonly fastened together using bolts. In many applications, they are subjected to impact loads during their service life. Their response and failure behavior between these conditions needs to be known for their safe use. The objective of this study was to develop computationally efficient finite element models for bolted joints under impact loading. First, a three-dimensional detailed finite element model for a bolted joint was developed by using solid elements. With this full modeling, the plan was to suggest the physics of the impact event accurately as possible without any unease about computational cost. In the design of mechanical structures containing numerous fastening elements, use of detailed models is not viable, because the computational cost of the analysis dramatically increases with the increased number of complex interact parts. Instead, simplified models that only account for dominating effects should be utilized so that the analysis time can be reduced without compromising the level of accuracy. Accordingly number of simplified finite element bolt models were developed and then compared with the full model with regard to the solution accuracy and computational cost to select the most representative and cost-effective simplified model. [2-8] The experiments are often time consuming and expensive. To overcome these problems, the finite element analysis (FEA) is seldom use in sheet material joints in recent years. The FEA has the great advantage in terms of mechanical properties in a sheet material joint of any kind of geometrical shape under various load conditions can be determine. For understanding knowledge of the current development in FEA of the sheet material joints latest literature relating to FEA of sheet material joints is analyzed in this chapter, in terms of procedure, strength, vibration features and assembly dimensional prediction of sheet material joints. Some major numerical issues are discussed, including material modeling, meshing process and failure criteria. It is concluded that the FEA of sheet material joints will ease future applications of sheet material bonding by allowing some system parameters to be selected to give a suitable process window for possibility of successful joint to the manufacture. This will allow various tests to be simulated that would currently take too long time to perform or quite a bit expensive in practice, such as modifications to joint geometry or material properties. [9-12] Particular research ao analytical model of bent beam is presented. On performing analytical solution to calculate and solve integral of differential equations. In this research there is a nonlinear function which is to be again calculate. This function is basically extracted from the combination of developed equations. And the values which are unknown are found by running a programmed code written in C language. In order to verify and validate the results issued from analytical model Finite Elements simulations are turned on ANSYS software. 3D FEA simulations are used to analyze the evolution of contact interface and also to study the stability limits of bolted joint under loading. [13-17]

2. FEM Modeling

In these study to Finite Element Analysis (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life, as well as in engineering. We used these methods for design analysis, hand calculations, experimental and computer simulations. FEA is the most widely applied computer simulation method in engineering and closely integrated with CAD/CAM applications. The three dimensional model of an experimental setup is developed on CREO software which will ease the job of analysis of various parameters on HYPERWORKS software. For the analysis we are providing/working with contact between the plates where bolt is tighten, joints, support (Fixed), directions, loading conditions and meshing etc. Bolt joint testing rig has been designed and constructed for checking the performance of loosening of threaded fasteners. In this set up, an unbalanced mass is used to produce dynamic force in the system. This unbalanced mass is mounted on a pulley which is driven by a motor through a v-belt. An external motor is used to provide rotation to the belt which connects the motor to the pulley. The pulley further is mounted on a bell crank lever. Further, the bell crank lever is connected with a plate across its other end, as shown in figure 1 [18]

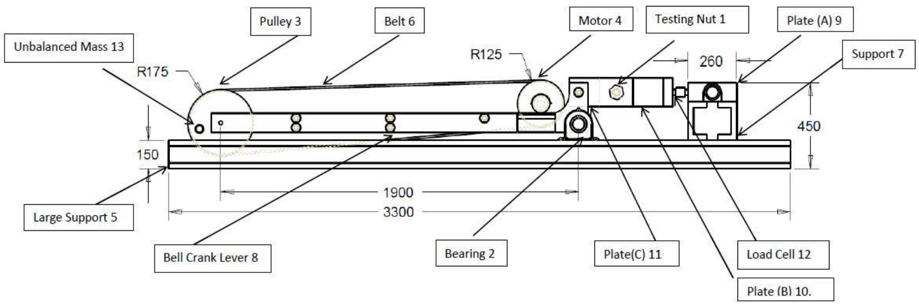

Figure 1 Elevation- Bolted joint testing rig.

Figure 2(a, b) show the drawings of plates with actual dimensions for the bending and axial conditions respectively.

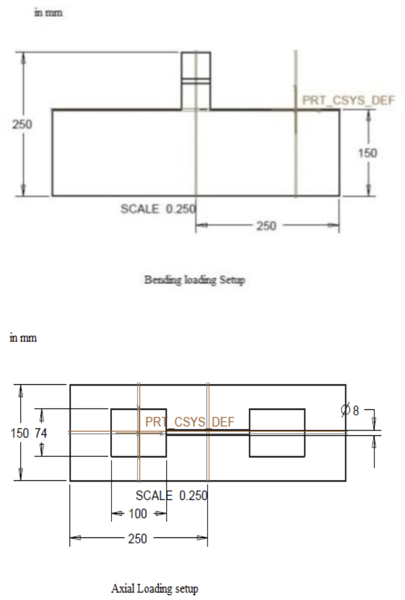

Figure 2(a, b) bolted joint plates dimensions for axial and bending attachment

3. Modal Analysis

The shape in which structure vibrates at its natural frequency us called its mode shapes. Hypemesh was used for the modal analysis of individual component and the rig. The model was built in Creo 2.0. After assembly & converting the model into .igs format this model is imported into Hyperworks. After importing the model, contact and constraints are defined according to the real conditions of the rig. A default mesh is automatically generated during initiation of the solution. The user can generate the mesh prior to solving to verify mesh control settings. A finer mesh produces more precise answers but also increases CPU time and memory requirements. Modal analysis of the test rig is done between 1 to 150 Hz.

3.1 Modal Analysis for bending loading attachment

Table 1 lists the first six natural frequencies of bending test rig. Figure 3(a) to 3(f) show the mode shapes for the respective modes, where lateral, longitudinal and torsional Modes were obtained for 6 natural frequencies.

Table 1 Natural Frequency of the rig with bending force Attachment

Mode No.	1	2	3	4	5	6
Natural Frequency (Hz)	12.9	15.49	59.7	61.21	100.21	152.27

3.2 Modal analysis of Axial loading

Table 2 lists the first six natural frequencies of axial test rig. Figure 4(a) to 4(f) show the mode shapes for the respective modes, where lateral, longitudinal and torsional Modes were obtained for 6 natural frequencies.

Table 2 Natural Frequency of the rig with axial force Attachment

Mode No.	1	2	3	4	5	6
Natural Frequency (Hz)	13.37	15.76	52.79	55.02	103.17	135.24

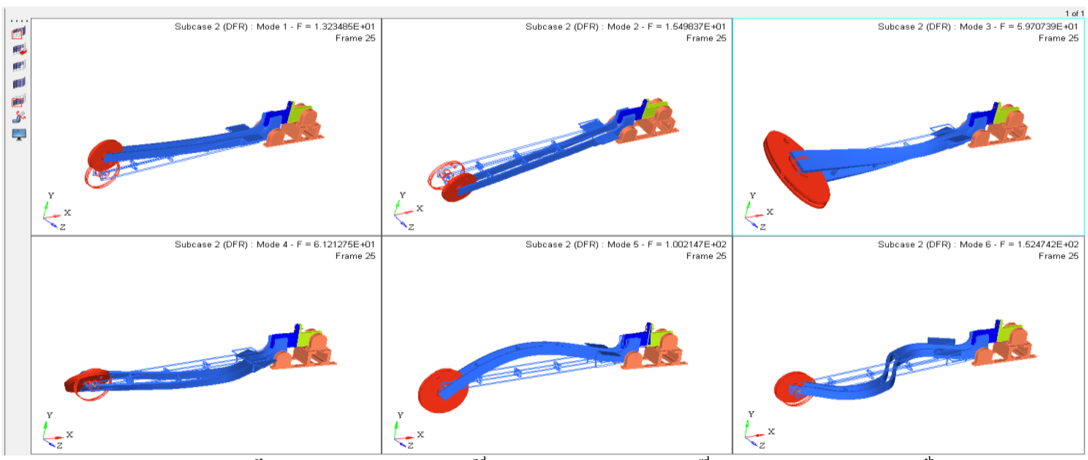

Figure 3 Mode Shape of Test Rig (a) 1st mode at 12.90 Hz; (b) 2nd mode at 15.49 Hz; (c) 3rd mode at 59.70 Hz; (d) 4th mode at 61.21 Hz: (e)5th mode at 100.21 Hz: (f) 6th mode at 152.27 Hz

IJASRM

International Journal of Advanced Scientific Research and Management, Volume 3 Issue 11, Nov 2018

www.ijasrm.com

ISSN 2455-6378

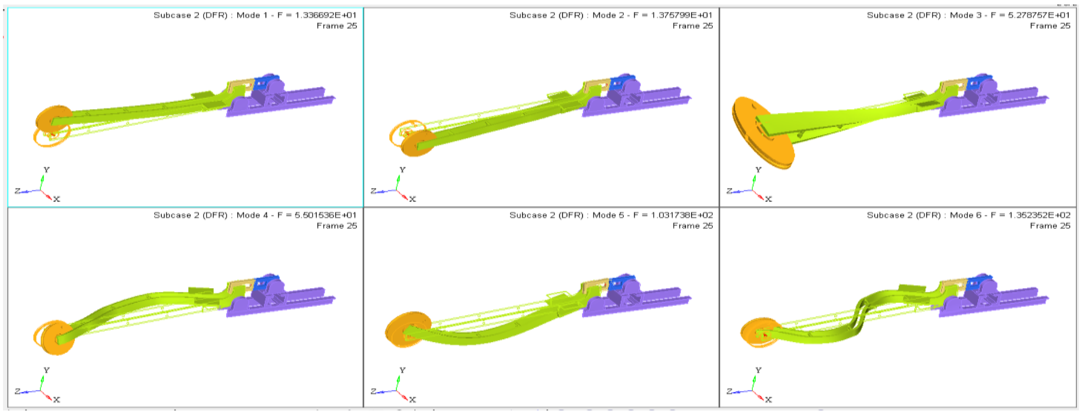

Figure 4 Mode Shape of Test Rig (a) 1st mode at 13.37 Hz; (b) 2nd mode at 15.76 Hz; (c) 3rd mode at 52.79 Hz; (d) 4th mode at 55.02 Hz: (e)5th mode at 103.17 Hz: (f) 6th mode at 135.24 Hz

4. Conclusion

A convergencce study for the test rig for all the 6 modes of test rig is also performed and shown in figure 5,6, which conclude that at range of 180000-182000 elements natural frequency dose not change by much which validate and confirm the result of analysis.The individual component natural frequencies also match with the test rig results which corelate the anaysis.

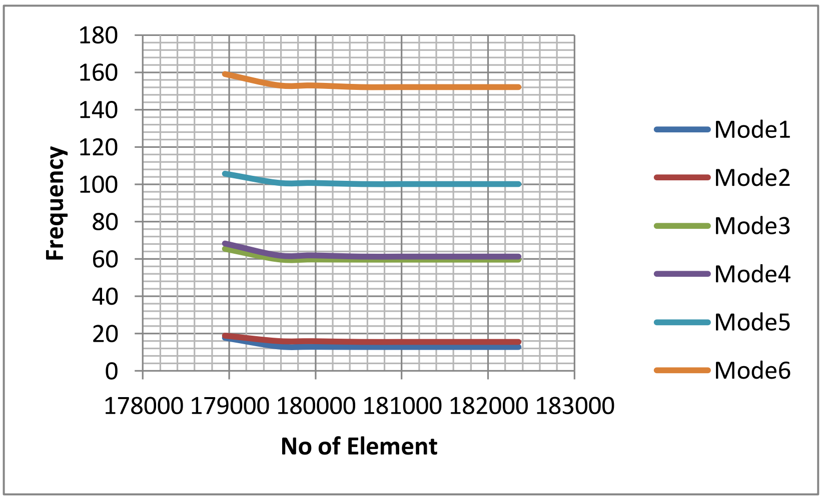

Figure 5 Convergence Graph for All the Modes against Natural frequency and No of element for bending test rig

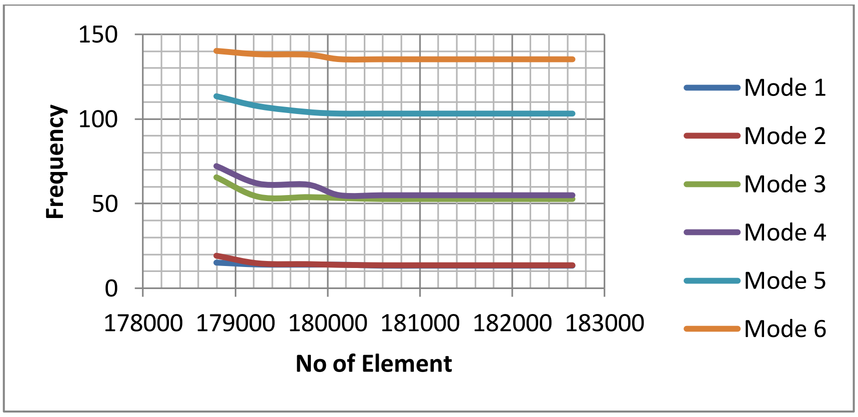

Figure 6 Convergence Graph for All the Modes against Natural frequency and No of element for axial test rig

The modal analysis of bolt test rig has the convergence in all the modes and it is also compared with the individual components which also give the same pattern. This shows that the results are accurate and convergent also. Further analysis of other attachment in the same test rig and different bolts can be performed. For this purpose, designer is liable to provide a table so that manufacturer & industrialist can study it and choose the range for their product. Developing mathematical model for such types of structure is very time consuming and requires deep knowledge which is only available in expert system. Element stresses results are analyzed and verified in further harmonic analysis.

Reference

[1] A Ali*, T W Yao, N Abdul ,M Yunin "Simulation and experimental work of single lap bolted joint tested in bending", Suranaree J. Sci. Technol. 14(4):331-345

[2] DJ Ewins "Modal Testing : Theory and Practice" Research Studies Press, Technology Engineering (1984),269 pages

[3] M Chouksey, JK Dutt, SV Modak "Model updating of rotors supported on ball bearings and its application in response prediction and balancing" , Measurement (2013),Volume 46, Issue 10, Pg No 4261-4273

[4] M Margarida, M Pedro, F Paulo, L M Hamid, "Compliant contact force models in multibody dynamics: Evolution of the Hertz contacts theory", Mechanism and Machine Theory (2012), Volume 53, Pg No 99–121.

[5] CD Hwan, PJ Hun, YH Hee, "Modal analysis of constrained multibody systems undergoing rotational motion", Journal of Sound and Vibration (2005), Volume 280, Issue 1-2, Pg No 63–76.

[6] A Suleman, "On the use of System Modes to Model Multibody flexible Structures", Acta Astronautica (2002), Volume 50, Issue 11, Pg No 653–664.

[7] LD Chan, HC Soo, "CAE (computer aided engineering) driven durability model verification for the automotive structure development", Finite Elements in Analysis and Design (2009), Volume 45, Issue 3, Pg No 324 – 332.

[8] T Hongliang, L Bin, L Hongqi, M Kuanmin, P Fangyu, H Xiaolei, "A new method of virtual material hypothesis-based dynamic modelling on fixed joint interface in machine tools", International Journal of Machine Tools & Manufacture (2011), Volume 51, Issue 3, Pg No 239–249.

[9] A handbook on "Hyperworks engineering solutions 11.0 Tutorial", Altair Engineering (2011), 173 Pages

[10] A Saxena, M Chouksey , A Parey, "Effect of mesh stiffness of healthy and cracked gear tooth on modal and frequency response characteristics of geared rotor system", Mechanism and Machine Theory(2017), Volume 107, Pages 261-273

[11] S Dravid, K Triphati, M Chouksey, "Experimental study of loosening behaviour of plain shank bolted joint under dynamic loading", International Journal of Structural Integrity(2015), Volume 6, Issue 1, Pg No 26-39,

[12] JS Tsai, YF Chou " Modelling of the dynamic characteristics of two bolt joints", Journal of Chinese institute of engineers (1998) Volume 11, Issue 3, Pg No 235-245

[13] X Liao, J Zhang, XXu "Analytical Model of Bolted Joint Structure and Its Nonlinear Dynamic Characteristics in Transient Excitation", Shock and Vibration journal(2016), volume 2016, Pg No 1-7

[14] KJ Bathe "Finite Element Procedure", Prentice Hall, Pearson Education 2nd Edition 2016.

[15] JR Baker, KE Rouch, "Use of Finite element structural models in analyzing machine tool chatter", Finite Elements in Analysis and Design (2002), Volume 38, Issue 11,Pg No1029–1046.

[16] M Chouksey, JK Dutt, SV Modak, "Modal analysis of rotor-shaft system under the influence of rotor-shaft material damping and fluid film forces", Mechanism and Machine Theory (2012),Volume 48,Pg No 81-93

[17] BF Robertson, DF Radcliffe, "Impact of CAD tools on creative problem solving in engineering design", Computer-Aided Design (2009), Volume 41, Issue 3, Pg No 136-146.

[18] S Dravid, K Tripathi, M Chouksey, "Role of Washers in Controlling loosening of Full Threaded Bolted Joints", Procedia Technology (2014),Volume 14, Pg No 543 – 552