Design Analysis & Optimization of Two Wheeler Handlebar

Prof. Susheel S. Pote[1], Pravin A. Savale[2], Azar A. Pathan[3], Bhushan B. Mahale[4], Gaurav B. Shevale[5]

Sandip Institute of Engineering and Management, Nashik, India

---

Abstract: *As India is second largest manufacturer of two wheelers, so most of the population prefer these for daily transportation. Also there are economic reasons, road conditions, and transportation distance for the people preference to motorcycle. Driving for long duration exposes motorcycle rider to extreme vibration arises from engine, road surface etc. resulted in severe effects on human health like musculoskeletal disorders, spinal injury, Harm Vibration Syndrome (HAVS), vibration induced white finger, finger numbness etc. In this paper, optimization, structural and modal analysis of the handlebar is done. Handle bar has been modelled using CREO V3, meshing and post processing is done in ANSYS (Workbench V18.2). Boundary forces will be calculated. In this Modal vibration analysis is done to find out response of structure to vibrations. Modal Analysis is well established technique which provides inherent dynamic properties of structure like natural frequency, mode shapes. Also Static analysis is done based on which further optimization will be carried out. Then experimental testing is going to be carried out for correlation of FEM results. There is good correlation found between software and experimental test results. Aim of this work is to develop method for vibration reduction by using software package (FEM) so, as to reduce time, cost required for experimental testing.*

Keywords *– Finite Element Method, Handle-bar, Modal Analysis, Structural Analysis, and UTM.*

1. INTRODUCTION:

In recent periods requirements from customer are increasing according to safety, cost, reliability, comfort of product in automotive sector. As our project Structural analysis, Model analysis and design Optimization of two wheeler handle bar having some challenges and infields customer complaints to enhance functionality, reliability, customer comfort and safety the optimum Design of component to be find out and validate from higher authority. The two-wheeler and the four-wheeler industry are normally faced with challenges related to function and safety.

Whole handle bar assembly is subjected to different types of excitations as experienced from road excitations, bumps, engine vibrations etc. Well established technique like Modal Analysis which provides inherent dynamic properties of structure like natural frequency, mode shapes, etc. is used to find out response of structure to vibrations.

Why modal analysis: Modal frequency response analysis is although a different approaches to decisively find the frequency response of a structure. Modal analysis is an economical dynamics. The dynamic behaviour of a structure in a provided frequency range can be modelled as a set of individual modes of frequency. The important modal parameters are: natural frequency or resonance frequency, and mode shape.

That's the reason why modal analysis on handle bar makes a very important investigation for getting out the mode shapes and frequency.

The main objective of this paper is to check different materials i.e. Aluminium, glass fibre, Mild steel, Stainless steels on handle bar and to search out its natural frequency.

Fig: 1 - Handlebar Geometry

2. LITERATURE REVIEW:

Borse et al [2017], Studied Design and Vibration Analysis of Motorcycle Handlebar by FEA Method and correlating it with Test Results. In this paper approach towards handlebar vibration has been carried by using analytical tools like Finite Element Method (FEM) where Modal vibrational analysis is done to find out response of structure to vibrations. Modal Analysis is well established technique which provides inherent dynamic properties of structure like natural frequency, mode shapes. Then experimental testing carried out for correlation of FEM results. There is good correlation found between software and experimental test results. The six different mode shape results are obtained showing different deflections of handlebar assembly.

Then experimental vibrational analysis has been carried for the results correlation.

Khemkar & Kadam et al [2015], Studied Structural Analysis and Design Optimization for Handle Bar Assembly of Motor-Cycle. A study is being carried by identifying the source of this failure of the handle bar assembly addressing the same with modified or improved design features for reducing the incidence of failure. Finite Element Analysis is used for the structural analysis using Radios or suitable solver. For this work, experimentation is performed for validating the performance parameter identified as `Buckling' of Handlebar. The load Vs displacement is recorded using load cells with data logger to display results. As discussed in paper the method of FEA and Experimentation is very useful for development of new products. With the help of FEA we are able to design a product with high reliability and quality which is useful to increase the performance for its optimum cost. It also reduces the chances of failure and cycle time required for the development of new product.

Khande et al [2014], studied Structural Analysis of Two Wheeler Handlebar. This includes problems like damage to various two wheeler parts because of vehicle slipping, collapsing, and minor dash or due to impact of heavy weight. In such cases the handle of vehicle deforms or undergoes buckling/bending. The attempt has been made in studding the deformation taking place due to buckling by analysing the stresses and reducing this stresses by making modification in dimensions or by changing material properties of handlebar. Simulation of the process helps to check the design of dies and plug as well helps to visualize the deformation of handlebar. The experimental investigation was conducted to turn mild steel as per ASTM A36 using AISI 302 stainless steel handlebar and by employing UTM (Universal Testing Machine) and Finite Element Analysis. The effect of dimensions of specimen on the buckling/bending stresses of the handlebar was studied under dry condition.

Parihar & Huzare et al [2015], Studied Performance Analysis of a Handle Bar Using Finite Element Methods. FEA methodology can be used to decrease design cycle time, quantity of prototypes and more importantly testing time and its related charges. The Aim of this paper is to improve a design and a prototype for a handle bar assembly of two wheeler which come across strength requirement. The handle bar vibrations can be contained within the prescribed limits using computational methodology for problem solving.

3. GENERAL PROCEDURES FOLLOWED:

From above literature reviews general procedures for FEM analysis are almost similar. It includes following steps.

3.1. Vibrational Analysis (modal):

ANSYS v 18.2 FEM package is used to carry out Modal analysis of the motorcycle handlebar. The entire FEM analysis procedure was completed in following steps.

A. Model preparation

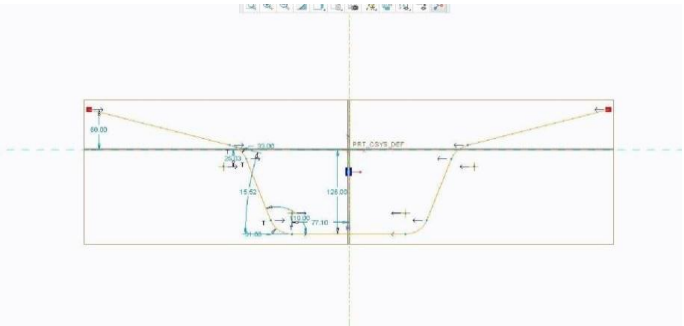

Fig: 3.1.1-Creo Model

The ultimate purpose of finite element analysis is to create mathematically the behaviour of an actual engineering structure with all nodes and elements, material properties, real constants and boundary conditions. Handlebar geometry is modelled by using Creo v3 and imported into **ANSYS v18.2** and further analysis is carried out.

B. FEM Modal Analysis:

After the geometric model formulation, modal analysis was carried out for the given boundary conditions. Among the different solver available on ANSYS v18.2 for Modal analysis, of which program control type solver was used to extract the natural frequencies and mode shapes of the motorcycle frame. Vibration analysis was carried out for finding natural response of structure by some impact or displacement. For this purpose handlebar meshing is carried out. By applying limits of nodes and elements meshing quality is kept whiting the acceptable range. Meshing is shown in fig 3.2.1

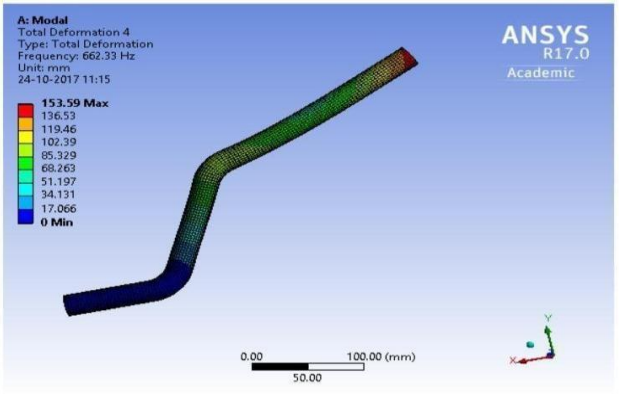

Fig: 3.1-Modal Analysis (For Ref. only)

3.2 Structural Analysis:

After applying a boundary conditions and suitable meshing for various materials (MS, SS, Al) total deformation and equivalent stress strain is obtained. Values can be varied according to the thickness selected.

For structural analysis purpose various quantities being measured these are equivalent strain, equivalent stress, total deformation etc. Structural analysis signifies stress-strain relationship while modal analysis signifies frequency at particular nodes in the element.

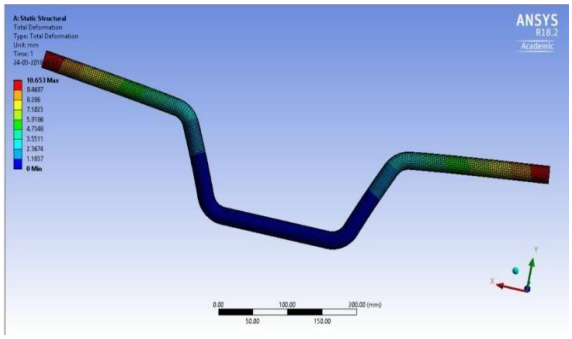

Fig: 3.2.-Structural Analysis (For Ref. only)

5. EXPERIMENTATION

Experimentation is conducted on the test rig at the client laboratory. The assembly would be mounted on the Universal Testing Machine and the frequency of the cyclic loading for bending and buckling would be set based on the historical data as well as the input received from the analysis data. Failure can be predicted before the component is produced through the use of software which relies on FEA principles. The prediction at the component design stage ensures that the chosen geometry is compatible with the conditions of use. Close collaboration between Component designers, Process Engineers and the Test Engineers assures the compliance with very short development times.

The parameters influencing the performance of the subject application are listed below:

- Type of material.
- Mechanical properties of the material.
- Thickness of the component at a given section.
- Type and magnitude of force exerted.

Finally a benchmark reading taken for specified loading condition and experimental results are approximately equals to analysis results obtained from FEA analysis. Hence it shows that results obtained from FEA are should be consider for safe working of component and it is used for development of component..

6. CONCLUSION:

In this project ANSYS v18.2 package is used for Modal vibrational analysis and Static structural analysis of the motorcycle Handlebar assembly. In the static structural analysis there were three materials out of six were taken into consideration. The equivalent stress and strain was calculated for each materials whereas in the modal analysis the six different mode shape results are obtained showing different deflections of handlebar assembly. While on the other hand in the experimental investigation strain probes were applied on specific points on the handlebar and accordingly the following conclusions have been made from the above study:

- As there is good correlation between the Software (FEM) results and experimental test results with average error found to be around 8-10 % only.
- As Software (FEM) method can predict good results and number of iterations can be taken within less time so, there is no need of actual or prototype formulation to conduct experimental test. FEM approach can be used to reduce design cycle time, number of prototypes and more importantly, testing time and its associated costs.

REFERANCES:

[1] Pranavdeep A. Borse, Dr. Purushottam S. Desale, Design and Vibrational Analysis of Motorcycle Handlebar by FEA Method and correlating it with Test Results, International Advanced Research Journal in Science, Engineering and Technology, ISO 3297:2007 Certified Vol. 4, Issue 7, July 2017.

[2] H. A. Khande and P. A. Narwade, (2014), Structural Analysis of Two Wheeler Handlebar, International Of Technology Enhancements And Emerging Engineering Research, Vol 2, Issue 7 , Issn 2347-4289.

[3] Amol h. Parihar, Pravin P.Huzare, Swapnil s. Kulkarni, Performance Analysis Of A Handle Bar Using Finite Element Methods To Enhance The Strength, International Journal of Advanced Engineering Research and Studies, EISSN2249–8974,April-June,2015/18-21.

[4] Mr.Chandrakant S. Khemkar1 Prof. Ganesh A. Kadam2, Structural Analysis and Design Optimization for Handle Bar Assembly of Motor-

Cycle, International Journal for Scientific Research & Development| Vol. 3, Issue 09, 2015 | ISSN (online): 2321-0613.

[5] Mr. S. V. Jaswandkar and Mr. P.N. Deshmukh, Design Enhancement Of The Handle Bar Assembly Of An Automobile Using FEA, International Journal Of Mechanical And Production Engineering, Issn: 2320-2092, Volume- 2, Issue10, Oct.-2014