INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

SPECIAL ISSUE FOR
NATIONAL LEVEL CONFERENCE
"Technology Enabling Modernization of Rural India (TMRI- 2018)"

DESIGN, ANALYSIS AND INCREASING STRENGTH OF TVS JIVE BIKE HANDLE

MR. PANDURANG B. BHISE[1], ASST. PROF. Y. A. KHARCHE[2], DR. C. R. PATIL[3]

1. PG Student, Mechanical Engineering Department, VBKCOE MALKAPUR
2. Assistant Professor, Mechanical Engineering Department, VBKCOE MALKAPUR
3. Professor Mechanical Engineering Department, PRMIT BADNERA

Accepted Date: 19/03/2018; Published Date: 01/04/2018

Abstract: In day today life we frequently come across some minor incidents occurring in case of two wheeler. 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 bending. This dissertation aims at studding the deformation taking place due to bending by analyzing the stresses and reducing this stresses by making modification in dimensions or by changing material properties of handlebar. The stresses include were calculated out by making use of numerical methods. By analysis of the total deformation occurring in handle was estimated and comparison of obtained results was checked experimentally. Increase in thickness and changing the material properties of handle makes it safer.

Keywords: Structural analysis, Modal analysis, Optimization, ANSYS etc

Corresponding Author: D V KATURE

Access Online On:

www.ijpret.com

How to Cite This Article:

Pandurang B. Bhise, IJPRET, 2018; Volume 6 (8): 391-398

PAPER-QR CODE

INTRODUCTION

The two-wheeler and four-wheeler industry are normally faced with challenges related to safety. The compliance of vehicle in this regard is of almost importance while the same could be approved by the concerned regulatory authorities for being used on the public roads. Besides, all other parts and components that support and/or form an integral part of the assembly of the sub-system could be required to comply with the norms. The other areas attracting compliance are the warranty claims received from the customer during usage over the field or the report filed by the concerned field Engineer observing the field test for the vehicle. The breakage and/or damage to the component could be highlighted during the time the vehicle is put to actual use. The scope of this dissertation work falls in this area where the design of the component or the sub-assembly needs to be reviewed for the sake of failure during use.[1-2]
Handlebars are created from hollow metal tube, typically from aluminium alloys, mild steel, chrome plated steel and Stainless steel however additionally of carbon fibre and Ti, shaped to the required contour. Holes may be drilled for the internal routing of control cables for brake, clutch, and throttle. Risers hold the handlebars above their mounting position on the higher triple tree or at the high end of fork, and may be integrated into the bar itself or separate things. Bar-end weights are typically added to either end of the bar to damp vibration by moving the bars' resonant frequency away from that derived by the engine. Electrical heating elements could be added below the handle bar grips to provide comfort to the user in cold atmospheric conditions. [2]

Growing competition in automotive market makes it more and more necessary to reduce the development time and cost of the product development process. One of the most costly phases in the vehicle development process is the field durability test. High Expenses for this phase can be attributed to the number of prototypes used and time/efforts needed for its execution. Also, multiple iterations during designing, building and prototype testing are no longer affordable against the time and cost constraints for developing a competitive product. Today, analytical tools in the form of computer simulation have been developed to such a level that they reliably predict performance. [4]

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. [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. 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 In this paper ANSYS v16 package is used for Modal vibrational analysis of the motorcycle Handlebar assembly. 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. [1]

Gund1 & Sayyad et al [2016], Studied Modal Analysis on composite Pulsar Bike Handle Bar using Finite Element Analysis. In this paper, optimization and modal analysis of the handle bar of Pulsar 150 is done. Handle bar has been modelled using CATIAV5, meshing will be done in HYPERMESH12.0, and ANSYS will be used for post processing. Boundary forces will be calculated. Static analysis will be done based on which further optimization will be carried out. Re-analysis (modal analysis) will bedone on the handle bar by considering glass fibre composite material. From results of finite element analysis it is observed that all materials have stress values less than their permissible yield limit. So the design is safe. But if we compare the density of MS and glass fibre, glass fibre has much lesser density than any other. [2]

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. After FEA analysis experimentation is done on prototype of product to confirm the results and compare the results of FEA and experimentation and results found closely equals its clearly shows FEA readings are correct and component is safe for revised material AISI 404 (SS 404) and 2mm thickness. [6]

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. [3]

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. CAE software such as HyperMesh as a pre-processor, MSC Nastran or Radioss as a solver and HyperViewas a post-processor are considered for this dissertation work. Using this methodology, nature, amplitude, and frequency of the vibrations can be predicted during the design phase 3D modelling of the tube drawing process helped in visualization and conceptualization. The modelling saves the research time and minimizes the risk of design failure. Simulation of the process helps to check the design of dies and plug as well helps to visualize the deformation of handlebar. [4]

Harale Shivraj. N & Gyanendra Roy et al [2012], Studied RADIOSS has been used to model and simulate the behaviour of the Handle-Bar Assembly for Vibrations. The results are matching closely with the experimental test and required design modifications have been done from analysis and iterations. The work presented in this paper is in the early phases of ongoing work and it is important to note these promising results will strongly demand more detailed analysis for future projects.[5]

Harshada G. Deshmukh & S. G. Dambhare et al [2014], As this paper is on generalized approach on formulation of a multi-degree vibratory model for two wheeler rider system, result we get in terms of frequency response of human body and two wheeler for ideal operating condition. Also from analysis we will come to know to that which part of human body is affected more due to vibration.[7]

Rajratna M. Kharat & Dr. K. K. Dhande et al [2015], The results of this study show that applying different evaluation methods may lead to different conclusions about the WBV exposure. This study found that RMS produced much lower results for WBV exposure in motorcycle riding than VDV and Se (Static compression dose). This observation indicates that a motorcycle rider encounters many shocks. Vibration peak distributions encountered by motorcycle rider’s result from a combination of riding speeds and road profiles. If possible it is necessary also to avoid vibrational frequency below 90 hz.[8]

PROBLEM STATEMENT

Automobile bike steering assembly contains different components like handle bar, housing, frame etc. In that mostly handle bar fail in the course of use. These recalls are expensive for manufactures and lead to cause of accident to rider, to avoid failure of the component which could be reason to serious injury. That's why study was started to support in improvement of design and product qualification of handle bar assembly. For a two-wheeler/ motorcycle, the handle bar offers a means for steering the vehicle while in motion. The problem for this work is to minimize the adverse effects of vibrations magnified with the occurrence of resonance in the manoeuvring elements of the system i.e. the handle bar and the base. [6]

Vibration analysis is usually done to make sure that potentially catastrophic structural natural frequencies or resonance modes don't seem to be excited by the frequencies present in the applied load. Sometimes this is not possible and designers then got to estimate the maximum response at resonance caused by the loading. [2]

OBJECTIVES

- The scope of this work is to study the existing design and analysis.
- The main objective of the study is to optimize and find natural frequency of handle bar. This optimized model will have better performance.

Identify design alternative/s for improvement.

- To carryout Modal vibration analysis and Static Structural analysis for Handlebar geometry.
- To correlate FEM results with the experimental test results.
- Mathematical Model for the existing System.
- Validation through physical experimentation on over the existing.

Strength Analysis and Optimization

- To measure deformation in various type of handlebar.
- To study defects in handlebars in manufacturing process.
- Calculation of various designs for handlebar.
- Optimization of handlebars by using ANSYS.
- Modelling of handlebar for circular shaped tubes using CATIA software.
- To simulate the impact bending process of handlebar using ANSYS and analyze the results.
- Optimization of the handlebar profile.
- Reduction of the cost of expensive trials required for new product or process development.

METHODOLOGY

Vibration Analysis

The outputs of a vibrating system generally, depend upon the geometry, boundary conditions, material properties and external excitations. The main Objective of this paper is to analyse the handle bar of motor-cycle for performance enhancement to increase safety and comfort and reduce accidents. When handle bar exposed to load applied by rider hands, that varies accidentally in both magnitude and direction.

Strength Analysis

1. Designing of handlebar for circular sectional Tubes Using CATIA.
2. Simulate the Process by Using ANSYS as finite element analysis Software.
3. Dimensional Study for various stresses.
4. After this, computational analysis is done with the help FEA tool to measure the bending stresses of the handlebar.
5. After that validate the results obtained by conducting the experimental test and FEA.

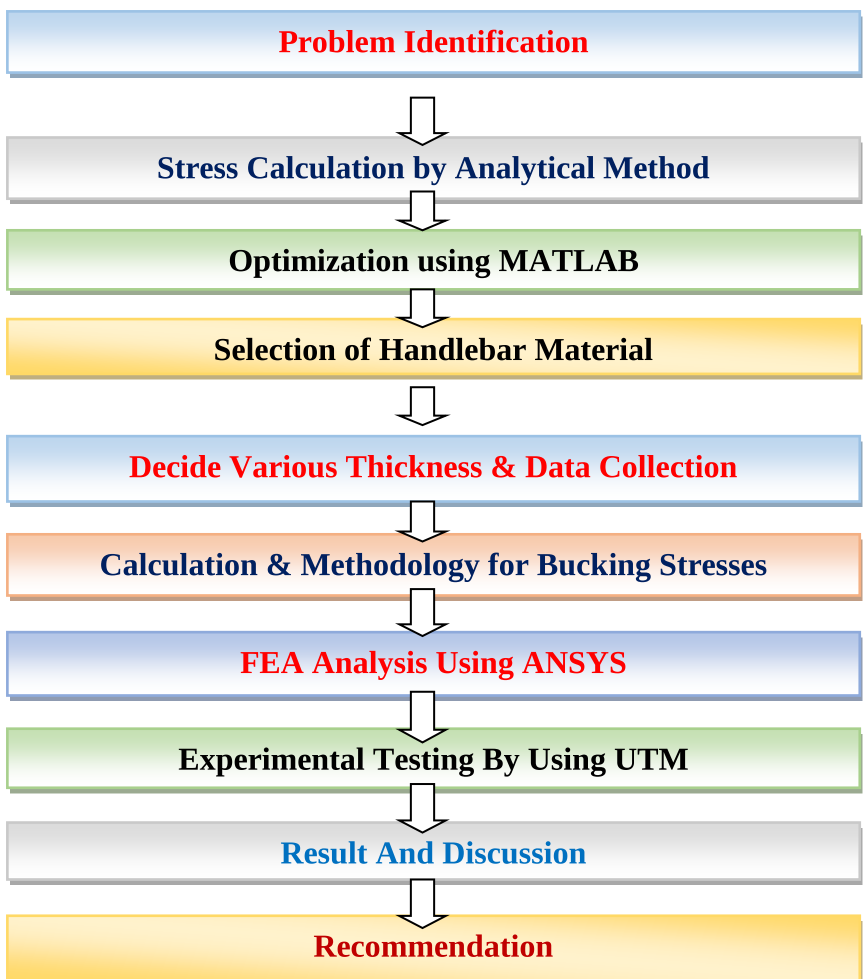

Fig 1:-Methodology/ Flow Chart

SELECTION OF MATERIAL

In practice however, the choice is limited by the following factors:

1. The ability of the component material to withstand with the required force pressure. Obviously, the component material must have a mechanical strength greater than that of the material to be worked.
2. Also, from the point of view of economy, the component (handle etc) must have a reasonable working life, that is, it must be able to withstand the considerably high working stresses for a reasonable period of time.
3. The economic requirement is that minimum possible deformation of the material should be obtained in a single working operation. This will depend upon the cold ductility and cold flow ability of the material.
4. Systematic selection for applications requiring multiple criteria is more complex. For hollow rod which should be stiff and light requires a material with high Young's modulus, high density and high yield stress. The characteristics such as *hardness*, high toughness, high compressive strength, and wear resistance are studied.
5. In particular, the analysis is focused on the evaluation of the limiting bending force necessary to increase or reduce the curvature of the beam in the plastic zone. The bending force depends on the compressive axial load, the geometrical dimensions of the beam, material coefficients, such as Young's modulus and yield stress, and the hardening model. The large number of variables involved, is reduced by introducing two dimensionless load parameters.

Material Properties	Yield strength (MPa)	Ultimate strength (MPa)	Density (g/cm³)	Elongation (%)	Elastic Modulus (Gpa)	Thermal Conductivity (W/mK)	Hardness (Brinell 3000kg)	Poisson's Ratio
Stainless Steel AISI 302(hard)	280	515	7.89	12-40	180	11.2-36.7	137-595	0.25-0.3
ASTM A36 steel(Mild Steel)	220	400-500	7.85	4-31	200	19.9-48.3	210-620	0.27-0.3

Table 1: Different Handlebar Materials

CHEMICAL PROPERTIES (Stainless Steel AISI302)											
Alloy	C	Mn	P	S	Si	Cr	Ni	Mo	Cu	N	Other
302	0.15	2	0.05	0.03	1	17.00-19.00	8.00-10.00	0.75	0.8	0.1	-

Table 2: Different Handlebar chemical properties

MECHANICAL PROPERTIES								
Material Properties	Yield strength (MPa)	Ultimate strength (MPa)	Density (g/cm³)	Elongation (%)	Elastic Modulus (Gpa)	Thermal Conductivity (W/mK)	BHN	Poisson's Ratio
Stainless Steel AISI 302(hard)	280	515	7.890	12-40	180	11.2-36.7	137-595	0.27-0.3

Table 3: Different Handlebar Mechanical properties

EXPECTED OUTCOMES

Optimization of the handle bar for its better performance is to be done in this project and we find out the design alternative for the improvement of handle bar. Modal vibration analysis and Static Structural analysis for Handlebar geometry is to be carry out.

REFERENCES

1. Pranavdeep A. Borse, Dr. Purushottam S. Desale et al, 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. Atul P. Gund, Dr. F. B. Sayyad et al[2016], Modal Analysis on composite Pulsar Bike Handle Bar using Finite Element Analysis, International Journal of Emerging Technology and Advanced Engineering, ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 6, Issue 8, August 2016.

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

4. Amol h. Parihar, PravinP.Huzare, Swapnil s. Kulkarni et al[2015], Performance Analysis Of A Handle Bar Using Finite Element Methods To Enhance The Strength,

5. International Journal of Advanced Engineering Research and Studies, EISSN2249–8974,April-June,2015/18-21.

6. HaraleShivraj. N, Gyanendra Roy et al [2012], "Vibration Analysis of 2 Wheeler HandleBar Assembly" Mahindra 2 Wheelers Ltd. Mahindra 2 Wheelers Ltd. 2012, PP. 1-7.

7. Mr.Chandrakant S. Khemkar, Prof. Ganesh A. Kadam et al [2015], 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

8. Harshada G. Deshmukh, S. G. Dambhare, M. R. Phate et al [2014], Review And Suggested Generalized Approach To Analyze The Impact Of Vibration On Two Wheeler Rider, International Journal of Science, Engineering and Technology Research (IJSETR), Volume 3, Issue 5, May 2014.

9. Rajratna M. Kharat, Dr. K. K. Dhande et al [2015], Impact of Vibration on Health of Automobile Rider- A Review, International Journal for Scientific Research & Development| Vol. 3, Issue 04, 2015 | ISSN (online): 2321-0613 .