

Design and experimental analysis of tooth impact test rig for spur gear

Akshay Shelke¹, Prof. D. D. Shinde², Prof. D. R. Satpute³

¹M.Tech student, Design engineering, Sandip university, Nashik, Maharashtra, India

²Associate Professor, mechanical Engineering, Sandip university, Nashik, Maharashtra, India ³Assistant Professor, mechanical Engineering, Sandip university, Nashik, Maharashtra, India

Abstract

This project is about the design and analysis of a prototype of tooth impact test rig for spur gear. The test rig was fabricated and analysis was conducted to study its' limitation and capabilities. The design of the rig is analyzed to ensure that there will be no problem occurring during the test and reliable data can be obtained. From the result of the analysis, the maximum amount of load that can be applied, the factor of safety of the machine, the stresses on the test rig parts were determined. This is important in the design consideration of the test rig. The materials used for the fabrication of the test rig were also discussed and analyzed. Static analysis of spur gear will be perform using ANSYS 19 workbench. Modeling of test rig and spur gear will be design using CATIA V5R20 software. Based from the results, there were limitations found from the initial design and the test rig design needs to be improved in order for the test rig to operate properly. Experimental study using strain gage to determine the strain at the gear tooth can be conducted with the availability of this test rig.

1. INTRODUCTION

- The common impact tests that can be found are Charpy and Izod. These tests date back to late 1800s and early 1900s. In 1901, Charpy proposed a standardized testing method known as Charpy impact test which is still used widely till today.
- Another test which is widely known is Izod impact strength test. Both are an ASTM standard method of determining the impact resistance of materials. The energy absorbed by bending or fracturing of the test piece is calculated from the mass of the pendulum, initial angle of swing, and the recorded angle of rise after the impact. This will give out an energy amount, which is empirically related to the ductility or brittleness of the material. Usually, the material is steel which have undergone heat treatment.
- During the years, several standard test methods such as ASTM, ISO and SAE begin to emerge to become a guideline in testing materials.
- Ziegler and Eberhard did a finite element analysis and experiment to investigate the impact on gear wheels subjected to force. In the FEA, ABAQUS is used and SIMPACK is used for the rigid body model. They simulated contact on 6 potential contact points. The result obtained using are almost similar, thus showing that the elastic multibody analysis is highly accurate.
- Experiments were made to validate the data obtained from the simulation. Two-impact body are used, spherical and cuboid. Both present different results due to the contact condition. They concluded that the experimental results validate their finite element model and elastic multibody model.
- Studies done by Gonzalez et al involves creating a virtual impact test rig to study the effects of the test on a set of elements with intraluminal failure model. The impact energy is varied and the results were compared with experimental values. Two laminates were considered in their study. However, they did not analyses the effects on the virtual test rig.

2. Objective

- Build a functioning test rig for spur gear.
- Modeling of test rig and spur gear will be design using CATIA V5R20 software
- Redevelop the test rig setup based on Pneumatic cylinder and control valve.
- Design and analysis of test rig setup using ANSYS 19 workbench.

3. Conclusion From literature review

In this report we successfully develop spur gear tooth impact test rig using CATIA model. We analyze the spur gear using ANSYS software. And find out Total deformation and equivalent stress using static analysis.

4. Methodology

Step 1- Initially research paper is studied to find out research gap for project then necessary parameters are studied in detail. After going through these papers, we learnt about Tooth impact test rig for spur gear.

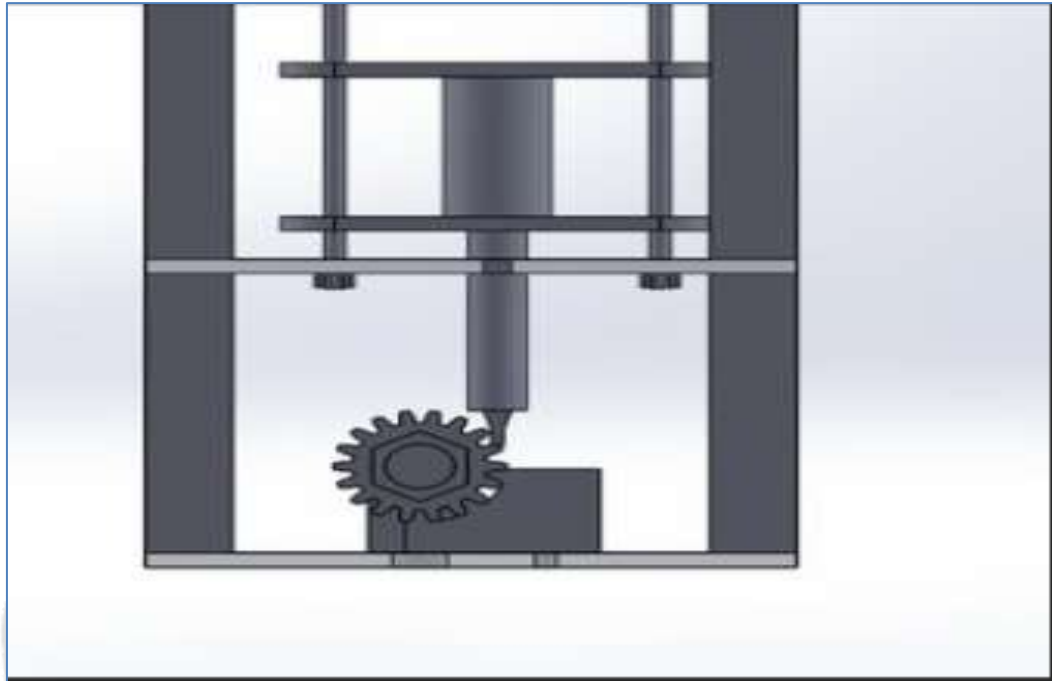
Step2 - Research gap is studied to understand new objectives for project.

Step 3-After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.

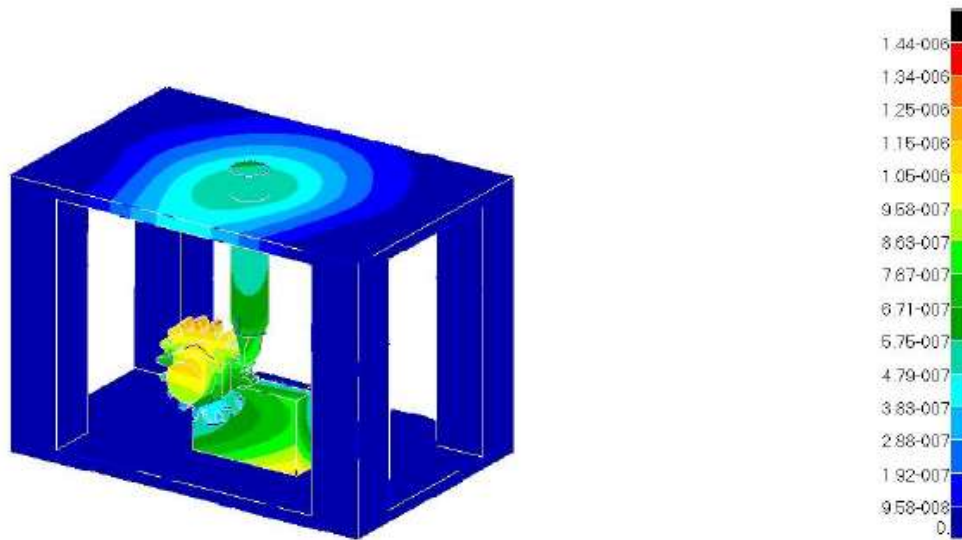
Step 4 - Redevelop the test rig setup based on Pneumatic cylinder and control valve. Design and analysis of test rig setup using ANSYS 19 workbench.

Step 5 - Comparative analysis between the experimental and analysis result.

5. CAD Model



6. Ansys of Cad Model



7. References

1. Experimental measurement and numerical validation of single tooth stiffness for involute spur gears by Fatih Karpal, Celalettin Yuce, Oguz Dogan.
2. Development and application of a test rig for tribological investigations under impact loads by Florian Böhmermann*, Oltmann Riemer.
3. Design and Analysis of Tooth Impact Test Rig for Spur Gear by Wafiuddin Bin Md Ghazali1, Ismail Ali Bin Abdul Aziz1, Daing Mohamad Nafiz Bin Daing Idris1, Nurazima Binti Ismail and Azizul Helmi Bin Sofian
4. Frictional Effects on Gear Tooth Contact Analysis by Zheng Li and KenMao.
5. Vibration analysis for tooth crack detection in a spur gear system with clearance nonlinearity by Yi Yang, Wenkai Xia, Jianming Han, Yunfei Song, Jiangfeng Wang, Yiping Dai.
6. A New Design of the Universal Test Rig to Measure the Wear Characterizations of Polymer Acetal Gears (Spur, Helical, Bevel, and Worm) by Samy Yousef, T. A. Osman, M. Khatta, Ahmed A. Bahr, and Ahmed M.

