

Design and Analysis of Tooth Impact Test Rig for Spur Gear

Wafiuddin Bin Md Ghazali¹, Ismail Ali Bin Abdul Aziz¹, Daing Mohamad Nafiz Bin Daing Idris^{1,3}, Nurazima Binti Ismail¹ and Azizul Helmi Bin Sofian²

¹Faculty of Mechanical Engineering, Universiti Malaysia Pahang, 26600, Pekan, Pahang, Malaysia

²Faculty of Chemical and Natural Resources Engineering, Universiti Malaysia Pahang, 26300, Gambang, Kuantan, Pahang, Malaysia

E-mail: ³daingnafiz@ump.edu.my

Abstract. This paper 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 analysed 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 analysed. MSC Nastran Patran software was used to analyse the model, which was designed by using SolidWorks 2014 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.

1. Introduction

The common impact test that can be found are Charpy and Izod. These tests dates back to late 1800s and early 1900s. In 1901, Charpy proposed a standardized testing method known as Charpy impact test [1] 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 [2]. 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 [3]. This will give out an energy amount, which are empirically related to the ductility or brittleness of the material. Usually the material are 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 [4] 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 [5] involves creating a virtual impact test rig to study the effects of the test on a set of elements with intralaminar 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 analyse the effects on the virtual test rig.

Shin and Tuazon studied the impact factor behaviours of structural steels using an instrumented drop-bar testing apparatus [6]. From the experiment they obtained a load-displacement curve. A drop-bar was used instead of conventional Charpy impact test to reduce the oscillations appearing in the load time graph. The specimens that were used are S45C (AISI 1045) and SM490B (A633). A data acquisition system (DAQ) system and LabVIEW S/W based processing scheme is used to display the results. Their



experiment showed that brittle behavior is present in the lower testing temperature, at a higher temperature, the blunting formation in the notch root of the specimens becomes noticeable. They proved that with the results obtained, the new instrumented drop-bar impact testing technique adopting a long bar striker was effective and provide reliable results in evaluating the impact fracture behaviour and fracture mechanism of metallic materials.

A simulation of wheel impact test was done Chang and Yang [7]. Their study was to determine the impact failure of the wheel. Their finite element results were then compared with actual tests. The conclusion that they draw was a nonlinear dynamic finite element with a reasonable mesh size and time step can reliably estimate the dynamic response. They also stated that the total plastic work or limit can be set as a fracture criterion for prediction of wheel fracture during the impact test.

Shuaieib et al [8] noted that energy will be dissipated in several ways in a drop impact test rig. The authors states that energy will be dissipated by deformation of the specimen, rebound of the striker and elastic deformation of the test rig. The energy of rebound depends on the coefficient of restitution between two contact surfaces, which is shown in equation (1).

$$E_r = \frac{1}{2} m_1 V_1^2 \quad (1)$$

Frictional force in the guidance will cause some energy lost (E_g), as well as the vibration of the rig and also the energy absorbed by the specimen in form of heat. The energy losses can be grouped as transmitted energy (E_t), and can be expressed as follows.

$$E = E_c + E_r + E_g + E_t \quad (2)$$

However, this energy (E), can be equal to the potential energy of the striker assembly at its falling height. The total potential energy of the striker is defined as blow efficiency (η_b) which can be calculated using equation (4). E_c can be calculated if η_b and E are known using equation (3).

$$E_c = \eta_b E \quad (3)$$

$$\eta_b = \frac{1 - e^2}{1 + W_1 + W_2} \quad (4)$$

$$e = \frac{V_2 - V_1}{V_0} \quad (5)$$

From the various literatures, an impact test on a spur gear tooth was nowhere to be found, prompting the authors to study this topic. Moreover, most literatures do not consider in analysing the rig, which is important if the rig is in house produced. This paper seeks to address that analysis should be done to in house produced impact test rig as well as the test specimen. The designed impact test rig follows the design of Gardner impact test [9].

2. Methodology

Several steps must be done in order to complete the design of the test rig and analysis for both test rig and the spur gear. The rig is built based on the size of gear that will be tested. The flow chart of this research are shown in Figure 1. The gear specifications are shown in Table 1.

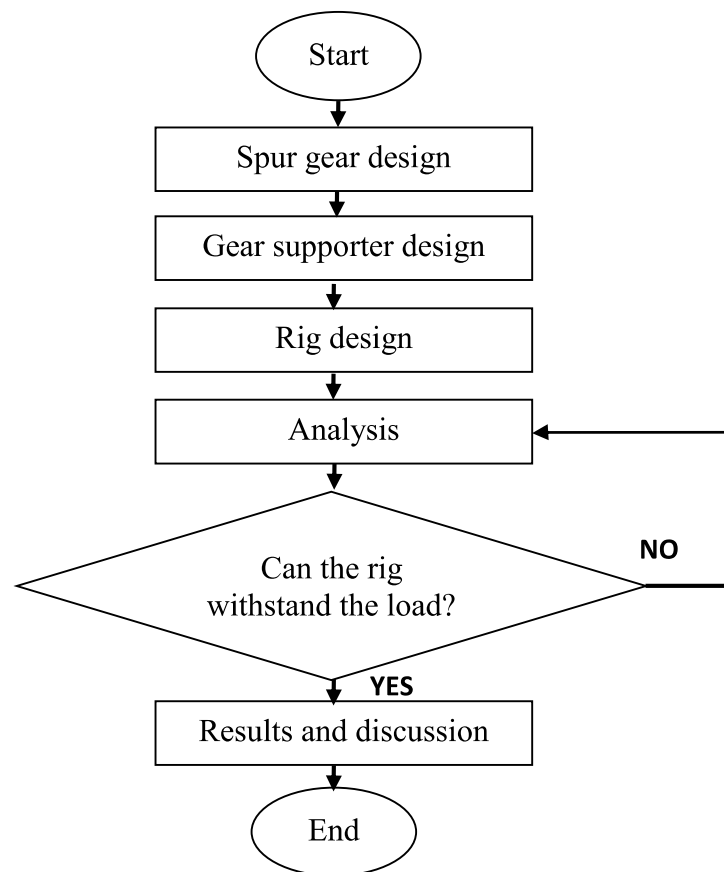


Figure 1. Process flow of designing and analysing

Table 1. Gear specifications

Parameters	
Module, m	4mm
Pressure angle, α_0	20°
Number of teeth, z	18
Face width	10mm
Materials	AISI:1040, 3215, 4140

2.1. Test rig design

All of the parts were drawn using SolidWorks 2014. The other parts are drawn gradually on the same working space. By doing it this method, it is more effective and the model will become a single solid. When the model is imported into Patran 2014, the program will detect it as a single solid, thus the analysis can be done without error.

Figure 2 -7 show the components of the tooth impact test rig for spur gear. Some of the components are not shown independently, but can be seen in the complete design.

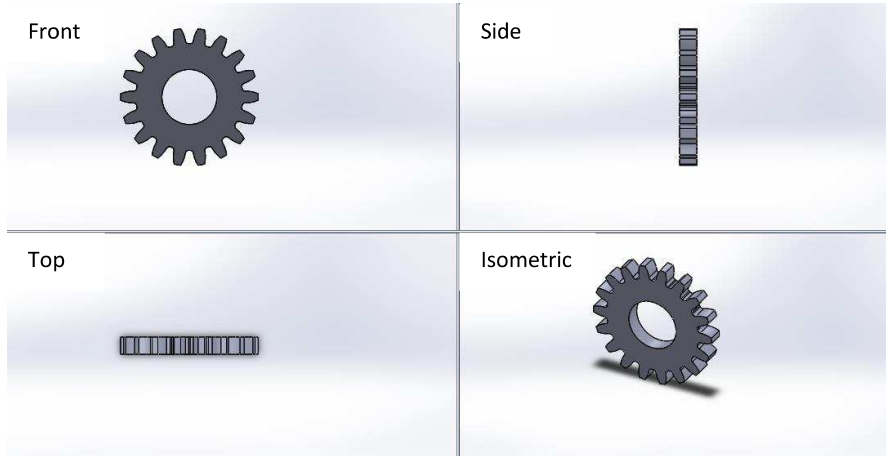


Figure 2. Spur gear model

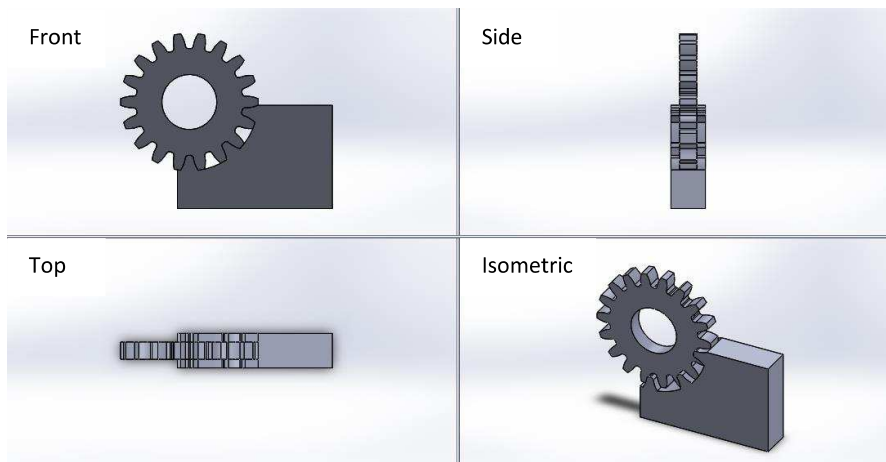


Figure 3. Spur gear with supporter

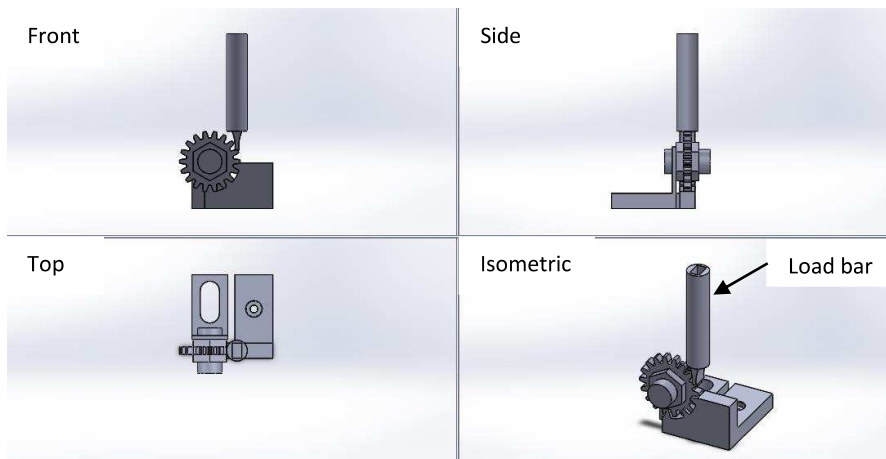


Figure 4. Spur gear, supporter, load bar and supporter locking mechanism

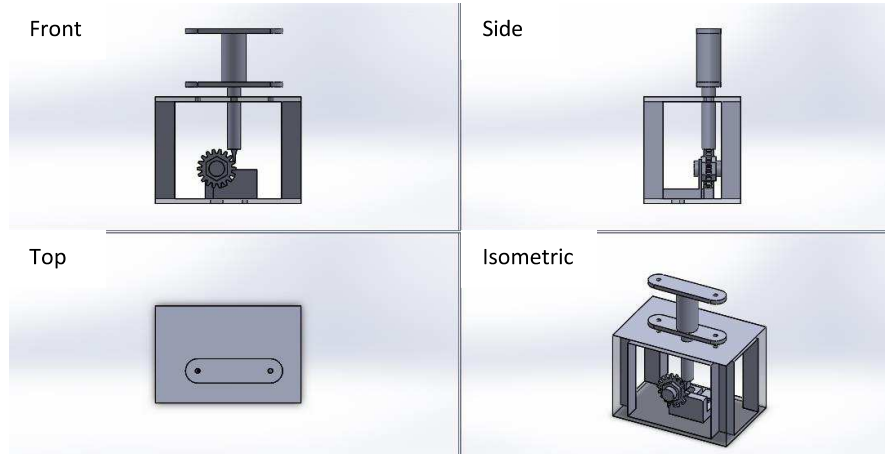


Figure 5. Tooth impact test rig with weight

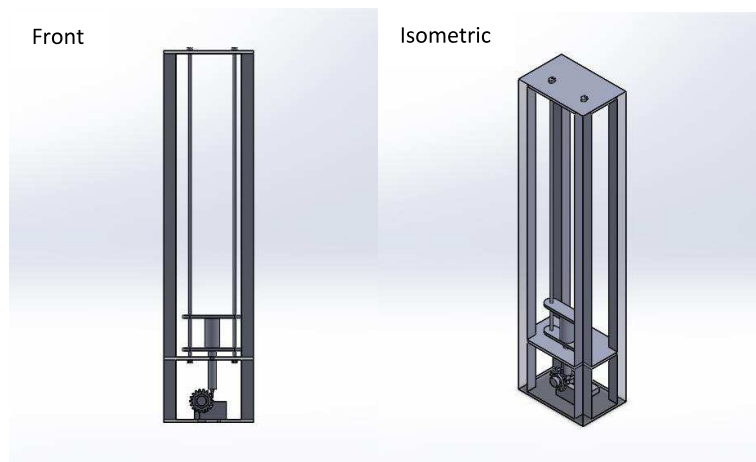


Figure 6. Complete design of tooth impact test rig

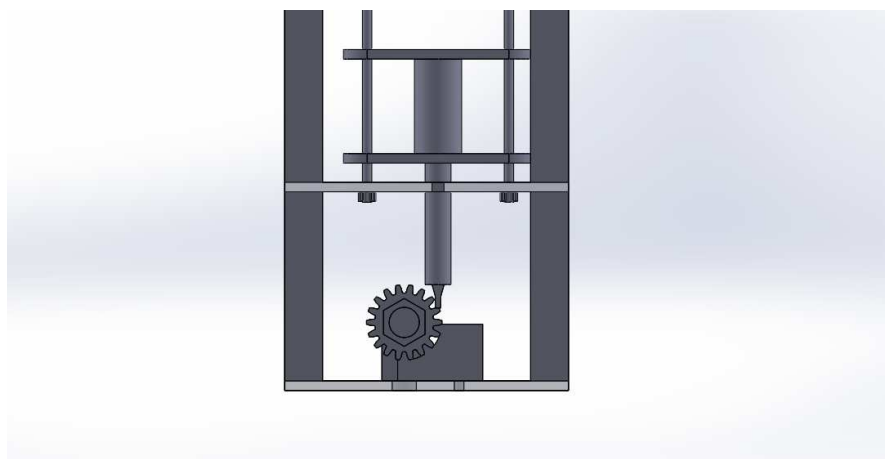


Figure 7. Complete design of tooth impact test rig (zoomed in view)

The overall height of this test rig is 1.25 meters. The weight can be raised up to a height of 1 meter. The gears that can be fitted to this test rig must have the parameters stated in Table 1 but can have a boss width up to 20mm. The weight is guided by wire rope to reduce friction during the free fall. The proposed material for the test rig is AISI 1040.

To obtain the actual reading from the impact test, strain gages will be attached to the root of the teeth. A DAQ system will be setup to collect the data and it will be compared with the data obtained from the finite element simulation.

2.2. Finite element analysis

For the finite element analysis (FEA), MSC Patran Nastran 2014 is used. The drawing design is imported to Patran as a Parasolid file. The force applied on top of the weight will be 50 N. The results obtained will show the deformation and stresses undergone by the spur gear teeth and impact test rig. The material properties of the gear are shown in Table 2.

Table 2. Material properties

	Material	AISI 1040	AISI 3215	AISI 4140
	Young modulus, E	206 GPa	207 GPa	210 GPa
	Poisson ratio	0.29	0.29	0.29
	Hardness, Brinell	84	84	84
Component Element Properties, %	Carbon, C	≤ 0.42-0.50	≤ 0.10-0.20	≤ 0.38-0.43
	Iron, Fe	≤ 98.51	≤ 97.11	≤ 97.86
	Manganese, Mn	≤ 0.60-0.90	≤ 0.30-0.60	≤ 0.75-1.00
	Phosphorous, P	≤ 0.04	≤ 0.04	≤ 0.035
	Sulphur, S	≤ 0.05	≤ 0.05	≤ 0.04
	Chromium, Cr	-	≤ 0.90-1.25	≤ 0.80-1.10
	Molybdenum, Mo	-	-	≤ 0.15-0.25
	Nickel, Ni	-	≤ 1.50-2.00	-

In the finite element analysis, several boundary conditions will be set up on the model. The boundary condition that were set up are the base of the test rig and the location where the force is applied. In the analysis, the upper part and the weight are not considered. This is due to their insignificance effect on the test rig. The upper part just holds the cable for guiding the weight. The weight was not considered as Patran 2014 gives us an option to add force. In addition, the resulting effects on the weight after the drop does not affect our test rig.

The automatic meshing by Patran 2014 was chosen as meshing method. When the global edge value is calculated automatically, the meshing size will be smaller at the circular edges compared to rectangular edges. Giving a better mesh result as shown in Figure 9. Figure 10 and 11 shows the constraint and the force applied on the test rig.

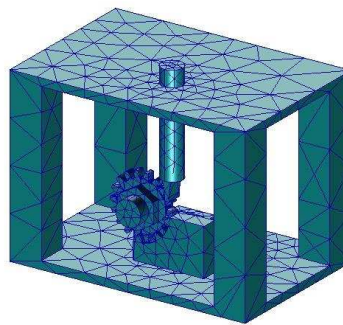


Figure 8. Meshed test rig

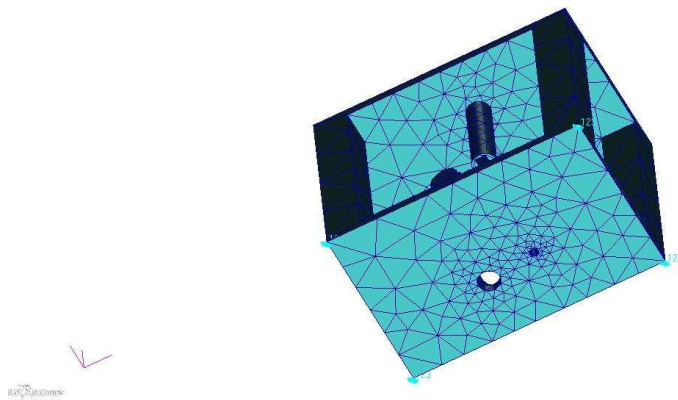


Figure 9. Fixed constraint on the bottom surface of the test rig.

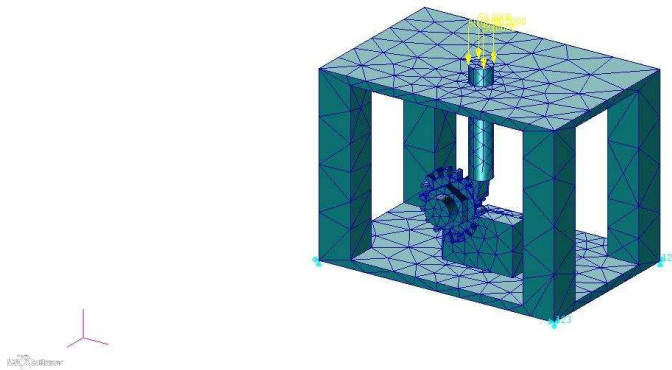


Figure 10. Forces and fixed constraint acting on the test rig.

3. Results and Discussion

The results from FEA using different type of material gives similar values. This is due to the Young modulus that have small difference the between materials. Figure 11 shows the constraint forces acting on the test rig.

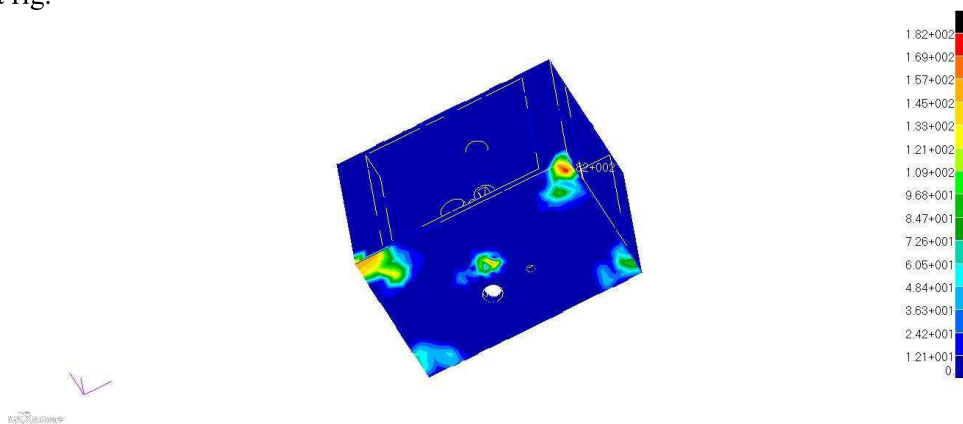


Figure 11. Constraint forces acting on AISI 1040

Figure 11 only shows the result from AISI 1040 since the maximum constraint forces acting on AISI 1040 is the same for AISI 3215 and AISI 4140 which is 182 MPa. Most of the force concentrates at the edges, and since the rig will be placed on the floor, the floor can absorb the force.

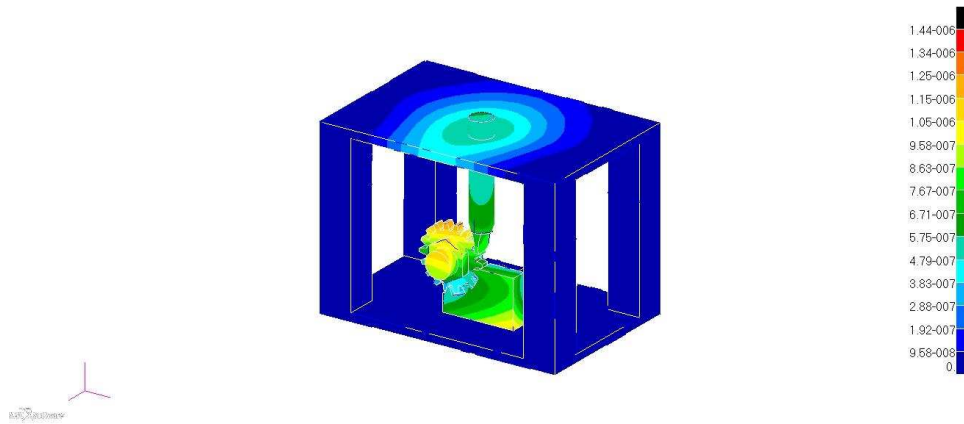


Figure 12. Displacement acting on test rig using AISI 3215

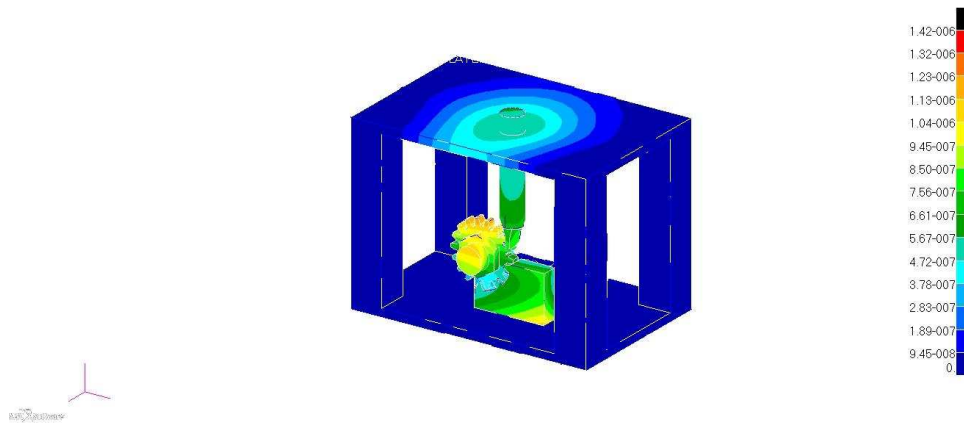


Figure 13. Displacement acting on test rig using AISI 4140

In Figure 12, the highest displacement value is 1.44×10^{-6} mm. this value is equal for AISI 1040 and AISI 3215. In Figure 13, the highest value of the displacement is 1.42×10^{-6} mm. All the materials undergo displacement which is very small, thus not influencing the structure of the test rig.

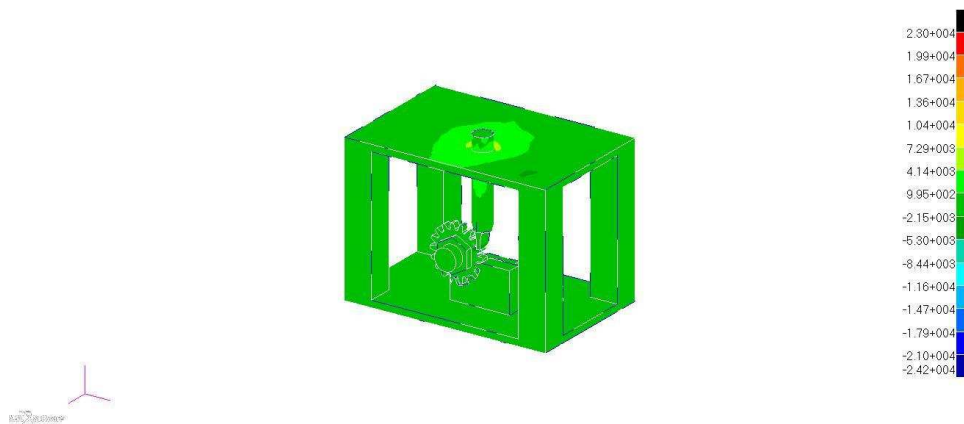


Figure 14. Isometric view of mean pressure acting on the test rig with material AISI 1040

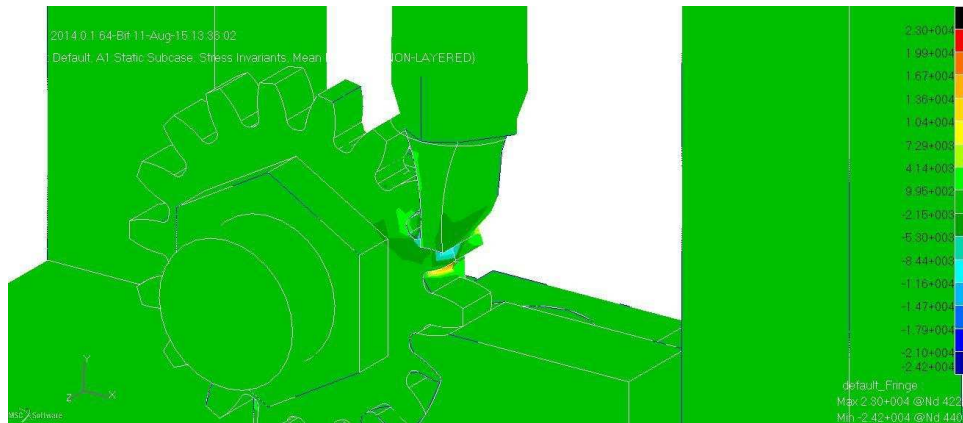


Figure 15. Zoomed in isometric view of mean pressure acting on test rig with material AISI 1040

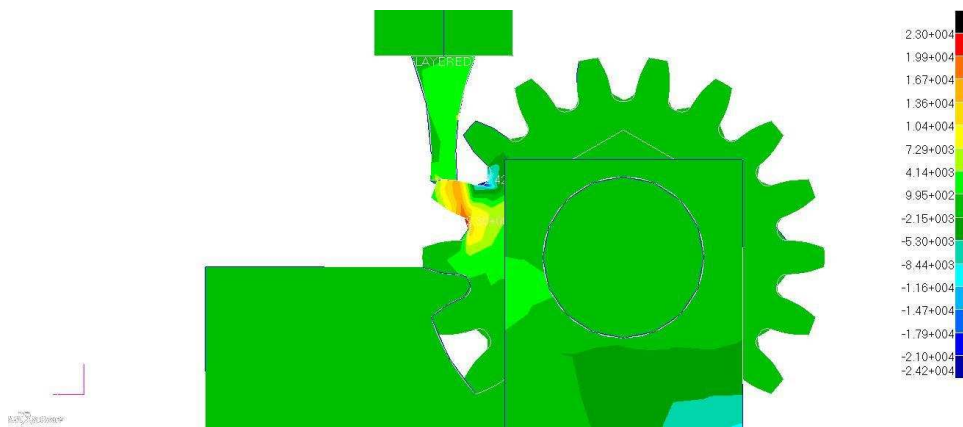


Figure 16. Rear view of mean pressure acting on test rig with material AISI 1040

The mean pressure acting on the whole rig can be approximated to be 0 on all material from Figure 14. This shows that the pressure undergone by the test rig is low but it is high at the contact between the teeth and load bar. The pressure tends to be higher at the rear position of the gear compared to the front position based from Figure 15 and 16 for all material. This occurrence can be due to the position of the gear or supporter. This can be further studied during the actual test after the rig is built and strain gages are attached to the front and rear face of the gear.

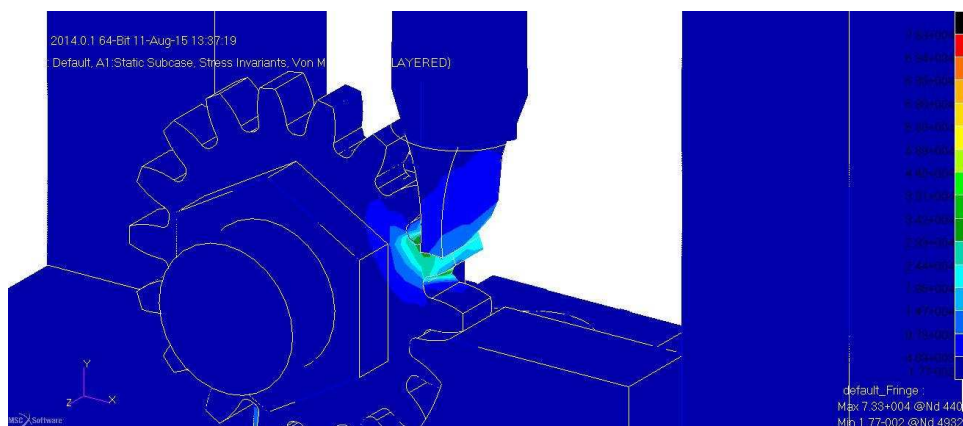


Figure 17: Isometric view of Von Mises stress acting on the test rig with material AISI 1040

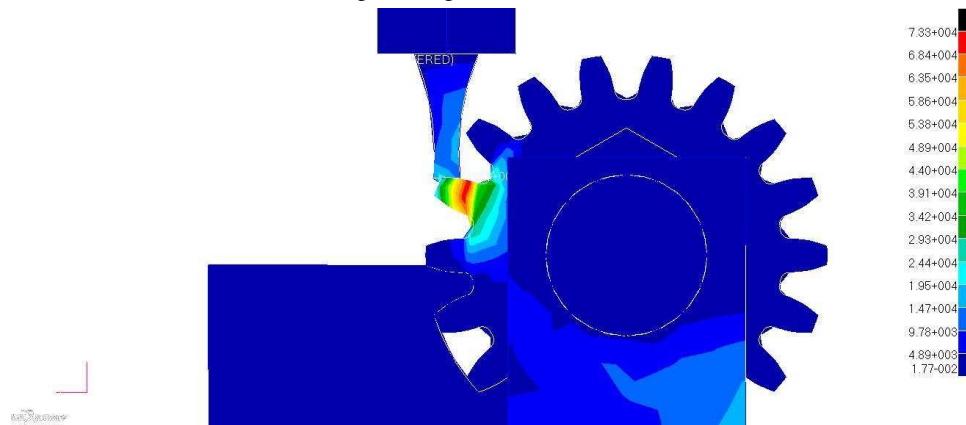


Figure 18. Rear view of Von Mises stress acting on the test rig with material AISI 1040

From Figure 17 and 18, the Von Mises stress acting on the rig can be deduced to be 0.0017 MPa. This value is too low to be considered as an influencing factor on the rig as it does not affect the rig structure which is shown in Figure 12 and 13. The location where the force effects the rig is only at the contact of gear tooth with the load bar.

A set of different value of force was used in order to find the difference that it may cause. The values are 100N, 500N and 1000N and the rig material is AISI 1040. This was done to ensure that a vary load can be applied to the rig without damaging it. Figure 19 to 21 shows the displacement underwent by the rig.

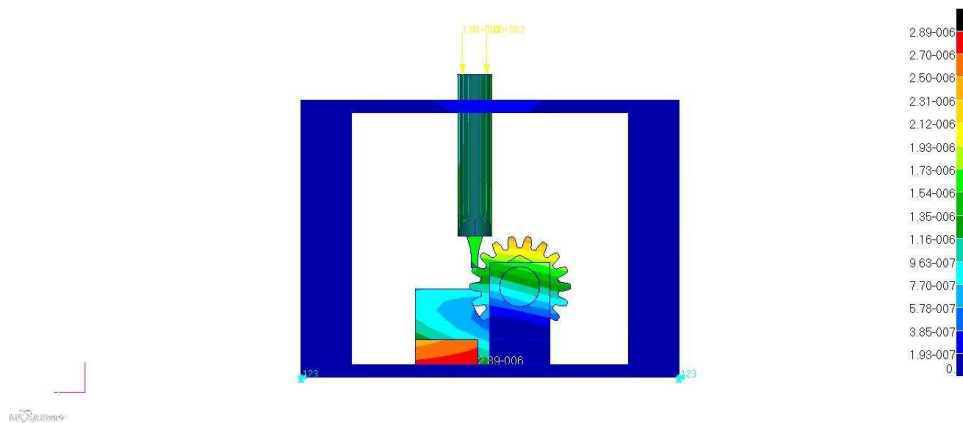


Figure 19. Displacement acting on test rig when force 100N is applied

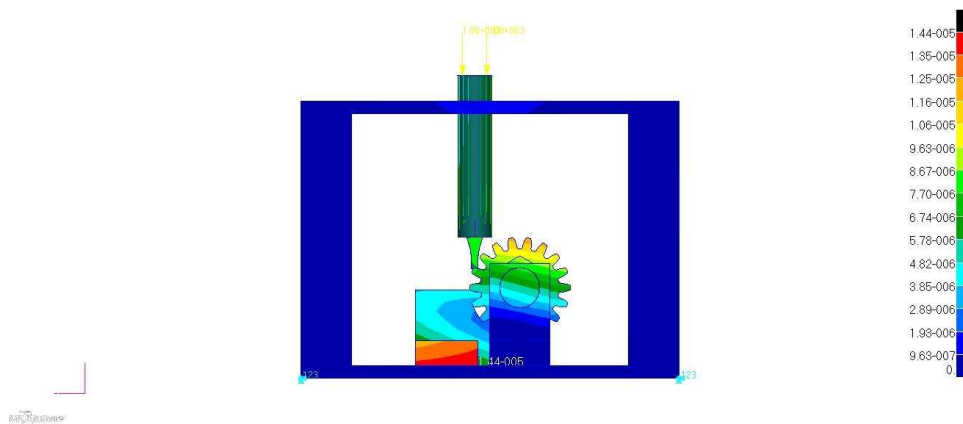


Figure 20. Displacement acting on test rig when force 500N is applied

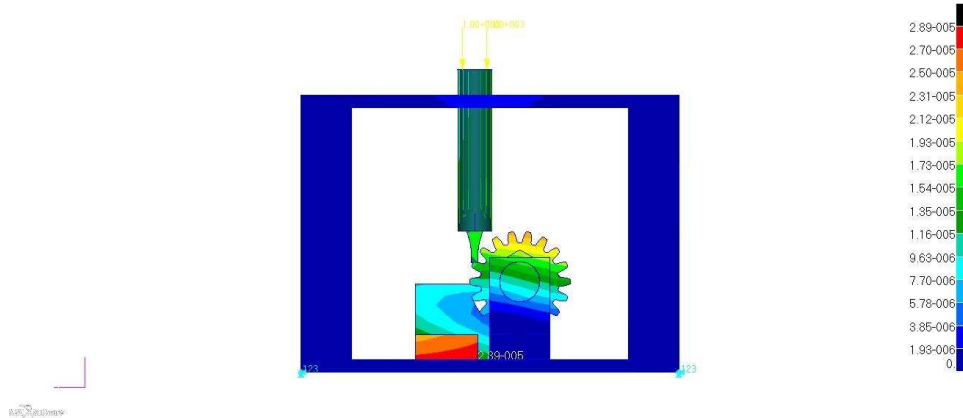


Figure 21. Displacement acting on test rig when force 1000N is applied

Table 3. Displacements under different forces

Force	Displacement
100 N	2.89×10^{-6} mm
500 N	1.44×10^{-5} mm
1000 N	2.89×10^{-5} mm

Table 3 shows the displacement underwent by the rig when different forces is applied. The highest displacement occurs when the force applied is 1000 N. All of the displacement is at the locking mechanism of the supporter. This is understandable as the supporter holds the gear and the load bar applies the force from the weight. However, the displacement is small and it does not affect the rig.

4. Conclusion

During this investigation, the deformation, mean pressure and Von Mises stress acting on the rig was found out. From these findings, it can be concluded that this rig design is suitable to test the impact strength of spur gear tooth.

The proposed material to build the rig is AISI 1040. This material is chosen after the analysis is done. From the obtained data after analysis, each material can withstand the force during the impact test. AISI 1040 is chosen because it is cheaper compared to AISI 3215 and AISI 4140.

With these findings, an impact test for spur gear tooth can be built and actual experiments can be done. This design impact height is not limited as the upper part of the rig can be disassembled and adjusted. In the near future an experimental study using strain gage to determine the strain at the gear tooth can be conducted with the availability of this test rig.

5. Acknowledgement

This research work has been done under the support of Universiti Malaysia Pahang under grant RDU1403132. The authors gratefully acknowledge the support given throughout this project.

References

- [1] C. W. Richards, *Engineering materials science*, Wadsworth Publishing Company Inc, 1968
- [2] W. K. Hatt and E Marburg, Bibliography on Impact Tests and Impact Testing Machines, *Proceedings ASTM*, Vol. 2, p. 283. 1902.
- [3] C. Turner, The Experimental Age of Strength of Materials, *The Centenary Of Imperial College, London 1907-2007*.
- [4] P. Ziegler and P. Eberhard, Simulative and experimental investigation of impacts on gear wheels, *Computer Methods in Applied Mechanics and Engineering*, 197, 4653-4662, 2008.
- [5] E.V. González, P. Maimí, P.P. Camanho, A. Turon, and J.A. Mayugo, Simulation of drop-weight impact and compression after impact tests on composite laminates, *Composite Structures*, 94, 3364-3378, 2012.

- [6] H. S. Shin and B. J. Tuazon, An instrumented drop-bar impact testing apparatus for investigating the impact fracture behaviours of structural steels, *International Journal of Impact Engineering*, 84, 124-133, 2015.
- [7] C. L. Chang and S. H. Yang, Simulation of wheel impact test using finite element method, *Engineering Failure Analysis*, 16, 1711-1719, 2009.
- [8] F.M. Shuaeib, S.V. Wong, K.S. Tan, A.M.S. Hamouda, R.S.R. Umar and M.M.H.M. Ahmad, Drop weight testing rig analysis and design, *Pertanika Journal of Science and Technology*, 12(2), 159-175, 2004.
- [9] ASTM INTERNATIONAL D 5420 - 04