Structural Analysis of an Ergonomic Motorcycle Test Rig using Finite Element Analysis (FEA)

Muhammad Firdaus Bin Azhar
2010426086
Supervisor: Professor Ir. Dr. Hj. Abdul Rahman Bin Omar
Co-Supervisors: Helmi Bin Rashid, Muhammad Izzat Nor Bin Ma'arof
Second Examiner: Associate Professor Ismail Nasiruddin Bin Ahmad

ABSTRACT

As part of an effort to reduce global issue concerning motorcycle accidents, an ergonomic motorcycle test rig named Postura Motergo was established. In validating the test rig's design, structural analyses were performed for blue print approval prior to fabrication. Computer Aided Design (CAD) model of the test rig was analyzed using Generative Structural Analysis (GSA) package in the state-of-the-art CATIA V5R20 software. Parameters considered were weight of a motorcyclist and material used as test rig structure as input to the Finite Element model. From there, the design was detailed out by the test rig's design team before coming out with a final detail design which was also analyzed using FEA simulations. Tetrahedron mesh was used to generate the meshes on the finite element model. Boundary conditions were located at every area that was assumed to have zero degree of freedom (DOF) in real condition. The loading applied was 150kg taking a maximum average weight of a motorcyclist. Results showed that the test rig's Von Mises Stress value was 1.79 x 10^8 MPa when 150kg of weight was applied. The yield strength of mild steel is 2.00 x 10^8 MPa and the factor of safety obtained was 1.12. With the positive results obtained from the FEA simulations, the Postura Motergo got its approval to be fabricated and safe to be used for further motorcycle ergonomic researchers especially the Motorcycle Engineering Test Lab (METAL) researchers. Upon the completion of development of the Postura Motergo, its first debut to the public during the Invention, Innovation and Design Exposition 2014 (IIDEX 2014) managed to secure a Bronze medal under the Invention category. Apart from that, two academic research papers relating to the Postura Motergo's test rig were also accepted to be presented in two International Ergonomics Conferences.

Keywords: *motorcycle, ergonomic, finite, element, analysis.*

Introduction

Motorcycles are considered as a "unique workstation" by Robertson and Minter [1] who strongly emphasized that motorcycle presents an "interesting problem to ergonomist in that it is a constrained workstation in which there is a limited available adjustment to suit the different need of riders". However, the significant increase in the number of fatal accidents involving motorcyclists in recent years is alarming. According to the Statistical Report Road Accidents Malaysia, Malaysia is one of the leading countries involving motorcycle road accidents and fatalities [2]. Among the ASEAN countries, Malaysia contributed to the highest road fatalities per 100, 000 population and road fatalities involving motorcyclists covered more than half of the distribution of the fatalities by the mode of transport [3]. This road fatalities may caused by riding discomfort experienced in their body parts during the riding process. The riding postures are proven to be the cause that contributes to the discomfort of body parts [4].

A test rig was intended to evaluate the cause of these problems from the perspective of ergonomics. The aim was to establish a test rig where the assessment could be perform in a safe and controlled environment, though, capable in replicating real world motorcycling scenarios. For instances assessments are performed in a lab without having the subjects to be exposed to environmental hazards. In addition, the motorcycle test rig was designed to cater for various riding postures based on the Riding Posture Classification (RIPOC) System [5]. The newly established motorcycle test rig was named the Postura Motergo.

In supporting the design of the Postura Motergo, a thorough structural analysis using Finite Element Analysis (FEA) method in CATIA V5 R20 CAD software has been executed in this project. Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures [6]. Types of mesh, mesh elements, material properties, boundary conditions, load and free body diagram (FBD) of the designed motorcycle test rig has been carefully outlined to run the FEA analysis. Manual calculations of the FBD to determine resultant forces acting on the motorcycle test rig and also limiting degree of freedom of boundary conditions are the main factors leading to the success of this project in running the FEA analysis [7-9]. Generative Structural Analysis

(GSA) package in CATIA V5 R20 Computer Aided Design (CAD) software has been extensively used where the whole structure were pre-assembled before running the FEA analysis. Final results comprising of maximum Von Mises Stress, maximum deflection and factor of safety [10] are the main outcomes in determining how safe the design is before it is being sent for fabrication.

Problem Statement

Firstly, in order to replicate real scenario of motorcycle riding, commonly a high-end, thus, expensive motorcycle simulator is needed. Common simulator use real motorcycle, hence expenses to buy new motorcycle is needed. Besides that, the features available in the simulator such as vibration and control synchronization with the audio and video system caused the simulator to be expensive.

Secondly, there are various riding postures documented via the RIPOC System [5]. The issue here, for the assessment of various riding postures, there is the need for various types of motorcycles and their designs.

Finally, prior to the fabrication of the new Postura Motergo, the approval of the new test rig design blue print is required. Optimum design of test rig in terms of material, structure, weight, and cost are needed to ensure design validity and safety. Optimal design also ensures that overdesign would be avoided, thus, keeping the fabrication cost under a low cost constraint (the total fabrication and overhead cost was approximately RM8000).

Objectives

The objectives of this project were:

1. To manually evaluate loading conditions on standard motorcycle white frame chassis.
2. To perform a Finite Element Analysis (FEA) simulation using established loading conditions on a newly designed ergonomic motorcycle test rig using CATIA V5R20.
3. To fabricate a motorcycle test rig as proof-of-concept using the established design parameters.

Scope of Project

Firstly, the scope of this project is to evaluate the loading conditions on a standard motorcycle white frame chassis using manual calculation of Free Body Diagram (FBD). The design and modelling of the motorcycle test rig used in this project was done by another Final Year Project (FYP) student because this project is a shared project between 3 FYP students including the author. The FBD calculations were done to determine the resultant forces acting on the motorcycle test rig and limit the degree of freedom of boundary conditions. In this research, only the resultant forces acting on the seat are considered since the forces acting on the handle bar and the foot peg are assumed to have been transferred to the seat.

Secondly, the author performed a Finite Element Analysis (FEA) simulation using established loading conditions on a newly designed ergonomic motorcycle test rig using CATIA V5R20 for this project. For the mesh size, not all parts use small mesh size as smaller mesh size will consume more computational power [11, 12]. Therefore, only critical parts such as the seat and the chassis were analyzes using smaller mesh size for the Finite Element Analysis.

Finally, the fabrication of the motorcycle test rig as proof-of-concept using the established design was done. In the fabrication process, the material used was mild steel due to its suitable mechanical properties. The material not only has high toughness property, but also has high weldability [13].

Significance of Project

The FEA was to ensure that the fabricated test rig is safe to use and the design of the test rig to have optimum structure strength. Therefore, when the experiment of the test rig is carried out, the safety of the test rig user will be guaranteed because the structure of the test rig will not fail.

Finally, the cost of fabricating of the test rig was reduced since the optimization of the test rig design has been finalized; hence the overdesign issue can be avoided. The optimization of the test rig design also has reduced the overall weight of the test rig. Since the idea of motorcycle test rig is still new to the world, this project also can benefit the research community of FEA whereby this project can be a reference for future researches that intent to do FEA on motorcycle test rig. Not only limited to motorcycle test rig, this project also can be a reference to researchers who want to do FEA on existing motorcycle chassis.

Methodology

During this project, a proper methodically scheduling is important to ensure all procedures are performed in the right order. Figure 1 shows the flow chart that represented the process flow compromised of Final Year Project 1 (FYP 1) and also of Final Year Project 2 (FYP 2).

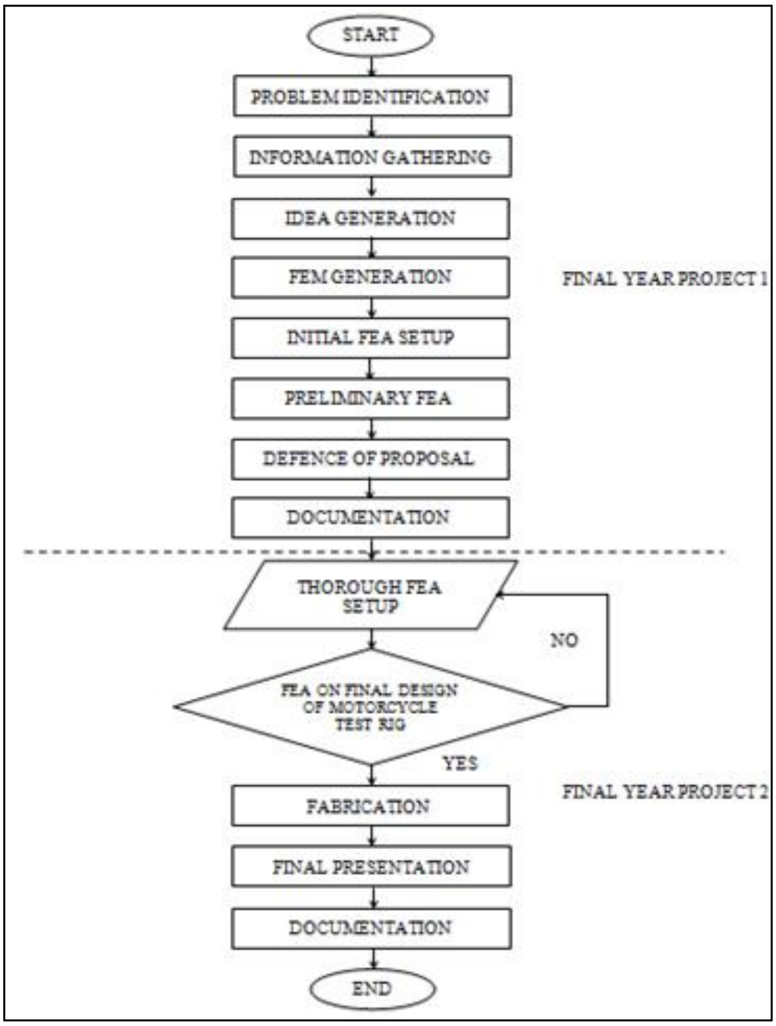


Figure 1: Flow chart of Final Year Project 1 and 2

Methodology of FYP 1

The author initiated the project with some initial reading that related with the title of the project that has been entrusted to the author based on the project synopsis and objectives. Then, discussion was held with the supervisor and co-supervisors in order to identify the problems regarding the project. The scopes and the significance of the project also were discussed. Then, the author proceed with information gathering. The purpose was to find the sources and supporting document to understand the concept of the test rig design and the significant of finite element analysis to the test rig. The author used several databases of webpages and e-book provider such as IEEE Xplore Digital Library, ScienceDirect, and Scopus for this purpose. Statistics on fatal accidents involving motorcyclists also were considered in order to strengthen the significant of this project. The supporting documents were mainly from published journals concerning the statistics on fatal accidents involving motorcyclists, designation of test rig, and the finite element analysis that were related directly and indirectly with this project.

For the idea generation, the author has identified the parameters to be considered in the Finite Element Analysis on the preliminary design of the motorcycle test rig as shown in Figure 2. The author calculated the Free Body Diagram based on the preliminary design of the motorcycle test rig model that has been designed by another Final Year Project student. Figure 3 shows the seat and the seat holder at the test rig. The position of loading and boundary condition was determined as shown in Figure 4. This was done in order to find the resultant force acting on the seat and the seat holder of the motorcycle test rig since both parts are the most critical part that receives the most body weight from the user of the test rig as shown in Figure 5. The author does not consider the force acting on the handle bar and foot peg since it is assumed that the force acting on both elements are being transferred to the seat and the seat holder.

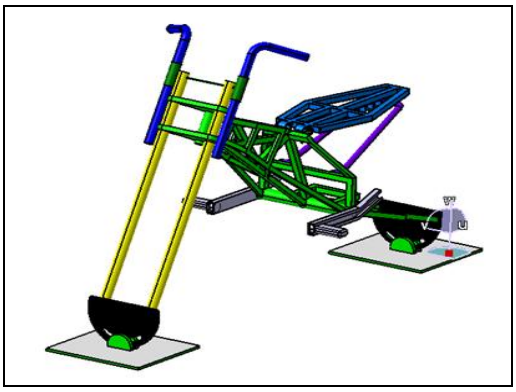


Figure 2: CAD preliminary design of the motorcycle test rig

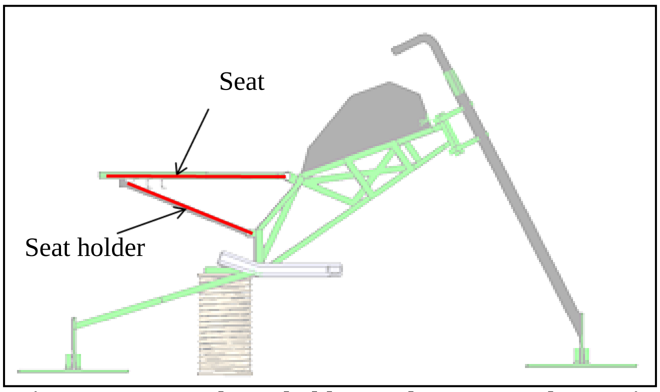


Figure 3: Seat and seat holder on the motorcycle test rig

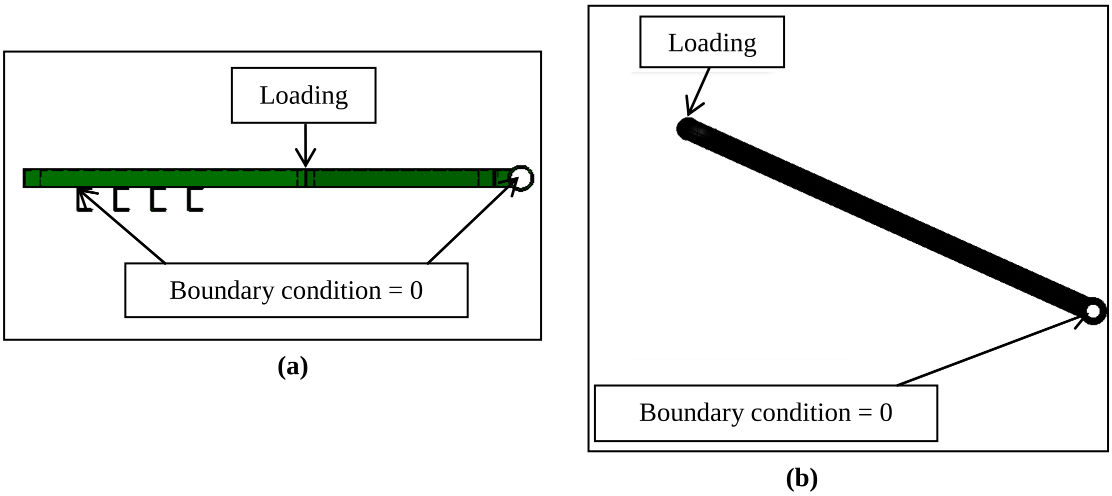
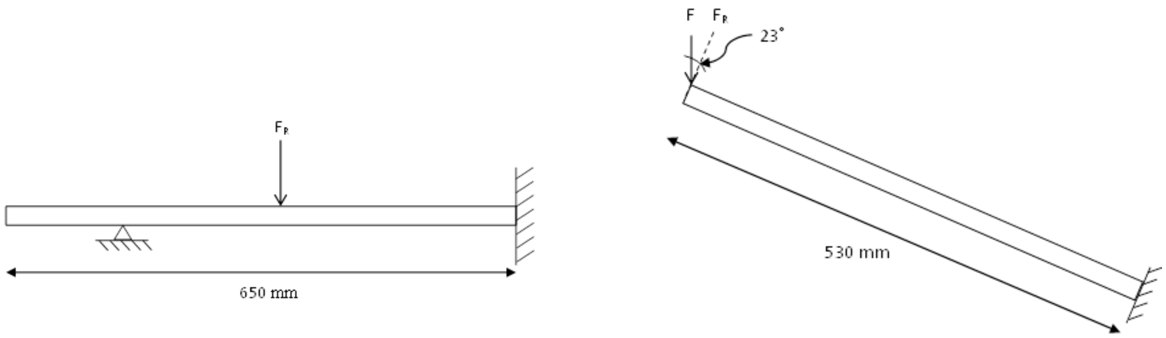


Figure 4: (a) Loading and boundary condition (a) at the seat and (b) at the seat holder



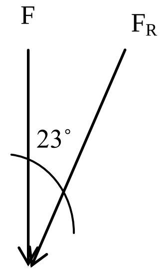
(a) **(b)**

Figure 5: Free Body Diagram (FBD) of the (a) seat and (b) seat holder, of the motorcycle test rig preliminary design

For the seat, since the position of the seat is horizontal, therefore there is no value of angle to be considered. Then, the value of theta (θ) is 0. The result of cos (0) is 1. Anything that is multiplied by the number 1 will get number back. The calculation of the resultant force of the seat is shown in Equation (1).

$$F_R = 1500\ (1) \tag{1}$$
$$F_R = \underline{\underline{1500\ N}}$$

Therefore, the resultant force is equal to the actual force exerted to the seat, which is F_R = 1500 N.
Different with the seat holder, it has slightly slanted position compared to the seat. Therefore, the resultant force is not the same as the actual exerted force. The calculation of the resultant force of the seat holder is shown in Equation (2).



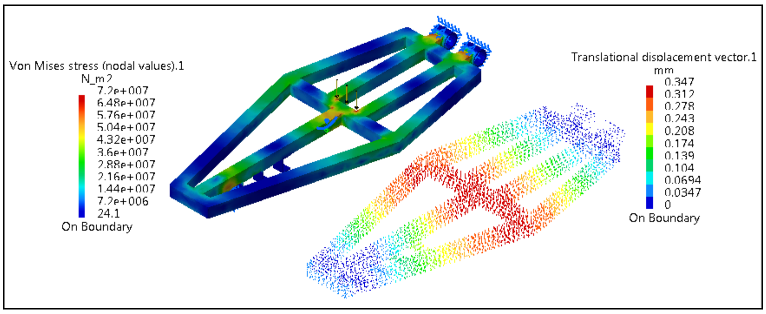
$$F_R = 1500 \cos (23) \tag{2}$$
$$F_R = 1500\ (0.92)$$
$$F_R = \underline{\underline{1380\ N}}$$

Therefore, the resultant force that acted on the seat holder is F_R = 1380 N.

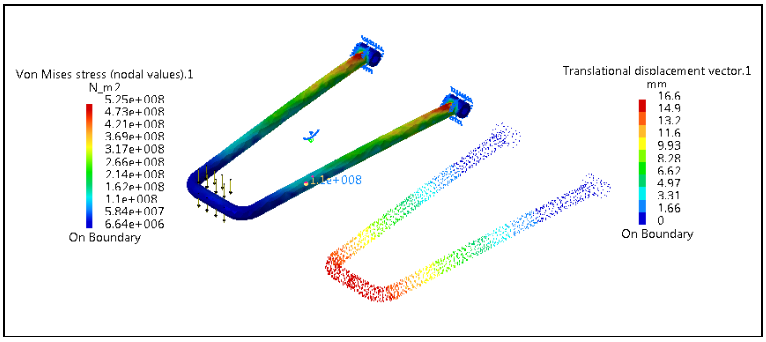
Based on the raw sketches of the motorcycle test rig generated by another FYP student, the author generated the Finite Element Model (FEM) of the motorcycle test rig. Various parts of the test rig were drawn by using CATIA V5R20, and then assembled to form a complete preliminary design of the motorcycle test rig. Based on the CAD Finite Element Model, the author run the Finite Element Analysis of the motorcycle test rig. However, before any Finite Element Analysis was carried out, initial setup needs to be determined. The boundary condition, loading, material and mesh need to be confirmed. After that, then only the preliminary Finite Element Analysis can be carried out.

For the material to be used to fabricate the test rig, the project team decided to use mild steel because of the suitable mechanical properties. In performing Finite Element Analysis, the smaller size of mesh means more accurate result will be obtained. However, smaller mesh will consume more computational power to solve the complex problem. The strategy of optimizing the mesh size can definitely reduce the analysis time without having to compromise on the quality of analysis results. For this project, the author used the mesh size of 10 mm.

After the boundary condition, loading, material and mesh have been finalized, the preliminary Finite Element Analysis was performed by using CATIA V5 R20 software. From that, the results of Von Mises Stress and the displacement of the examined parts were obtained. From the Finite Element Model that has been generated by another FYP student, the author has applied the material to each of the part. For this project, the author has selected the steel as the material. The results of Von Mises Stress and displacement of both seat and seat holder are shown in Figure 4.



(a)



(b)

Figure 4: Results of Von Mises Stress and displacement of the (a) seat and (b) seat holder

Methodology of FYP 2

Some adjustment need to be carried out to the motorcycle test rig design based on the results of the preliminary Finite Element Analysis that have been carried out at the Final Year Project I. The part that is considered as unsafe has been reinforced in order to make the design sturdier. Besides that, the material and the dimension of the motorcycle test rig also were reviewed and a more detailed material and dimension were finalized. After that, a thorough Finite Element Analysis setup of the complete motorcycle test rig have been determined such as the boundary condition, loading, material, and mesh. Figure 5 shows the final detail design of the motorcycle test rig.

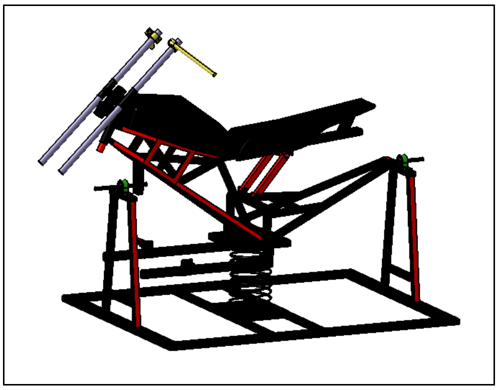


Figure 5: Final detail design of the motorcycle test rig

Then, the author has proceeded with the Finite Element Analysis of the modified design of the motorcycle test rig. The data of the Von Mises Stress, displacement, and deformation were obtained and the value of factor of safety has been calculated. Figure 6 shows the Finite Element Analysis results of the motorcycle test rig.

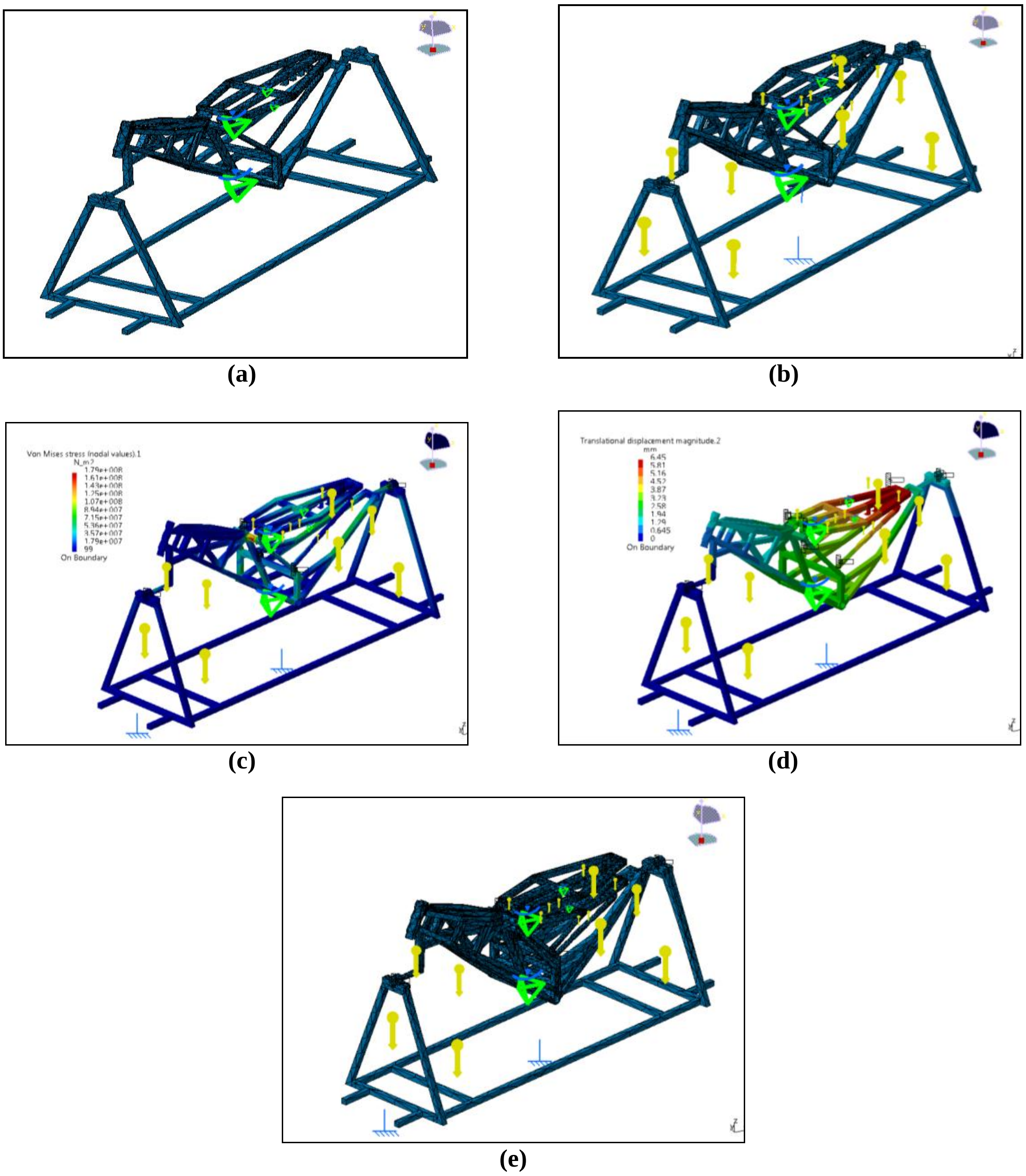
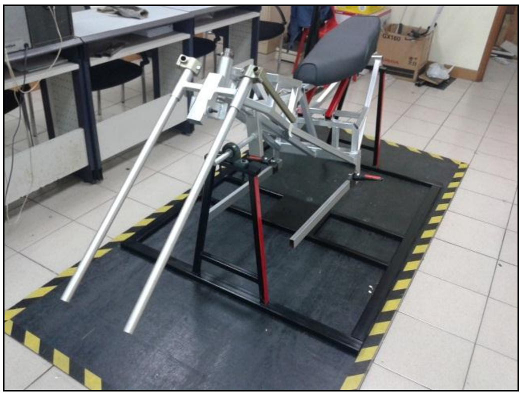


Figure 6: (a) Meshed finite element model (b) Assigned boundary condition and loading at 1500 N (c) Von Mises Stress (d) Displacement and (e) Deformation of the motorcycle test rig

After the calculation of factor of safety and the design was confirmed to be safe, the fabrication process of the motorcycle test rig started. Since the motorcycle test rig was fabricated at the Machine Shop at the faculty, the author and other FYP students started the fabrication as early as possible which was during the semester break, due to the absence of the students for the holiday to avoid congestion at the workshop and difficulties to use the fabrication equipments and tools. Metal inert gas (MIG) welding was used to bond the structures together giving the motorcycle test rig a rigid and robust chassis quality. During the fabrication works, the test rig design also went through some design revisions due to fabrication constraints. The revised designs were analyzed again using Finite Element Analysis to observe any obvious changes to its maximum Von Mises stress and deflection on the structure. It was found that the revisions on the design did not affect much on the strength of the structure but only a minimal increase in total weight that was not substantial to the design criteria. Figure 7 shows the fabrication work involved to construct the motorcycle test rig.

(a) **(b)**

(c) **(d)**



(e) **(f)**

Figure 7: (a) The cutting process (b) The welding process (c) The grinding process (d) The fabrication of the test rig almost finished (e) The colour spraying process and (f) The finished product of the fabrication process

Result and Discussion

Thorough Finite Element Analysis

In prior of doing the Finite Element Analysis, the author calculated the Free Body Diagram for both critical parts that are the seat and seat holder. The resultant force acting on the seat from the FBD calculation is the same as the actual force that is 1500N, while the resultant force acting on the seat holder is 1380N because of the angle.

Based on the preliminary Finite Element Analysis, the modification of the design of the motorcycle test rig was done by another FYP student. After that, a thorough Finite Element Analysis was carried out based on the modified and finalized design of the motorcycle test rig. Figure 8 shows the Von Mises Stress of the finalized design of the motorcycle test rig.

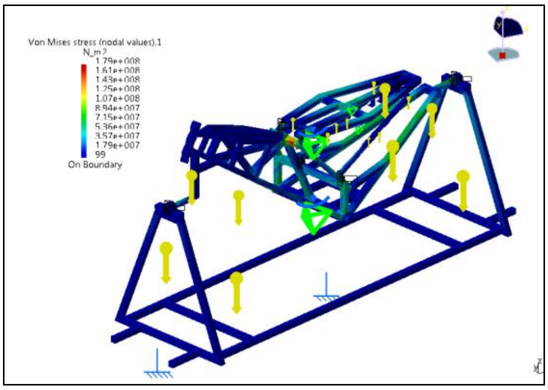


Figure 8: Von Mises Stress of the motorcycle test rig

The result of the Von Mises Stress shows that the maximum value of Von Mises Stress that the structure of the motorcycle test rig can withstand when a load of 150 kg is applied is 179 MPa.

Figure 9 shows the displacement of the finalized design of the motorcycle test rig.

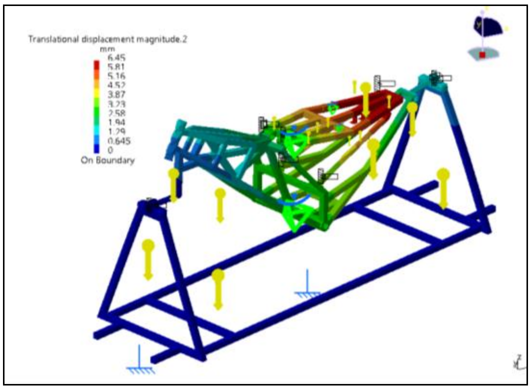


Figure 9: Displacement of the motorcycle test rig

The result of the displacement shows that the maximum value of displacement that the structure of the motorcycle can withstand before it fractures when a load of 150 kg is applied is 5.41 mm.

Factor of Safety of Thorough Finite Element Analysis

From the data of Von Mises Stress of the motorcycle test rig, the author then calculated the value of factor of safety based on the value of yield strength of material that was applied to the structure as shown in Equation (3).

Yield Strength of Mild Steel = 200 MPa
Maximum Von Mises Stress = 179 MPa

$$\text{Factor of Safety} = \frac{200}{179} = 1.12 \tag{3}$$

Achievement and Application

Establishment of Postura Motergo results in the establishment of the new Motorcycle Engineering Test Lab (METAL). The laboratory houses all the equipment related to motorcycle test rig. Using Postura Motergo, the author and other FYP students have carried out the analysis of muscle fatigue activity by using Electromyography (EMG) in the laboratory.

In the laboratory, the EMG test can be carried out in safe and controlled environment without having to worry about the environmental hazards but at the same time, real riding scenario can be replicated. Different riding postures were tested and the data of muscle activity for every posture was obtained. Figure 10 shows the EMG activity that has been carried out.

Figure 10: EMG test on the Postura Motergo

On 3 April 2014, the Postura Motergo has been filed for patent under Research, Innovation, and Business Unit (RIBU) to protect its Intellectual Property Rights (IPR). Besides that, the author also has participated in the Invention, Innovation, and Design Exposition (IIDEX) 2014 for the Postura Motergo and won bronze awards. Figure 11 shows the activity during the IIDEX 2014.

(a) **(b)**

Figure 11: (a) The author with the Postura Motergo and (b) Demo on Postura Motergo to the judge

Involvement of the author in the field of ergonomic had resulted in the enrolment of the author in the Human Factors and Ergonomics Society Malaysia (HFEM) (membership number: 095) along with the rest of the team members. All of the achievements that tailed the project marks as the recognition received by the author and the rest of the team members.

Recommendation

This project has a great potential to be pursued in a more advanced development. In this project, the author has used the CATIA V5 R20 software as the data analysis tool used in order to perform the Finite Element Analysis since the students of Mechanical Engineering has only exposed to this particular software in order to analyze the strength of a structure. Improvement can be made if the software that only focused on analyzing the structural strength such as ANSYS was used.

Conclusion

As the conclusion, the objectives of the project were successfully achieved. The first objective which is to manually evaluate loading conditions on standard motorcycle white frame chassis has been successfully achieved by the calculation of the resultant force on the free body diagram on the raw design of the motorcycle test rig. Meanwhile, for the second objective which is to perform a Finite Element Analysis (FEA) simulation using established loading conditions on a newly designed ergonomic motorcycle test rig using CATIA V5R20 has been successfully achieved, since both preliminary Finite Element Analysis and thorough Finite Element Analysis have been carried out. From the second objective, the structure of the test rig is safe or not to be used is determined. Last but not least, the third objective which is to fabricate a motorcycle test rig as proof-of-concept using the established design parameters also has been successfully achieved by the existence of the finished product of the fabricated motorcycle test rig.

Ultimately, this project helps to ensure the safety of the test rig user by determining the optimum structural strength of the test rig. Not only that, this project also helps to reduce the cost of fabrication and the weight of the test rig.

Acknowledgement

The author would like to express his sincere thankfulness and appreciations to his supervisor and co-supervisors for their guidance and advice for the author to complete this Final Year Project. The author also would like to express his thankfulness to Motorcycle Ergonomics Test Lab (METAL) research team for their cooperation and support from the beginning until the completion of this project. Special thanks to expert physiotherapist from Faculty of Health Sciences, Universiti Teknologi MARA Malaysia, expert staff from Faculty of Computer Science and Mathematics, Universiti Teknologi MARA Malaysia, technicians at the Machine Shop, Faculty of Mechanical Engineering, Universiti Teknologi MARA Malaysia and also to the Human Factors and Ergonomics Society of Malaysia (HFEM) for their continuous support upon this study. Finally the author would like to give his appreciation to everyone who had got involved in the project directly or indirectly.

References

[1] S. A. Robertson and A. Minter, "A Study of Some Anthropometric Characteristics of Motorcycle Rider," *Applied Ergonomics*, vol. 27, pp. 223-229, 1996.

[2] K. Ampofo-Boateng, S. Muzlia, V. Parnabas, N. Abdullah, and R. Sueb, "Human Factors and Motorcycle Accidents in Malaysia," 36, Institute of Research, Development and Commercialization, Universiti Teknologi MARA, 1999.

[3] M. M. A. Manan and A. Várhelyi, "Motorcycle fatalities in Malaysia," *IATSS Research*, vol. 36, pp. 30-39, 2012.

[4] K. Karmegam, M. Y. Ismaild, S. M. Sapuana, N. Ismaila, M. T. S. Bahrib, S. Shuibd, *et al.*, "A Study on Motorcyclist's Riding Discomfort in Malaysia," vol. 4, pp. 39-46, 2009.

[5] M. I. N. Ma'arof and I. N. Ahmad, "Proposed standard method for motorcycle nomenclature system," in *Network of Ergonomics Societies Conference (SEANES), 2012 Southeast Asian*, 2012, pp. 1-6.

[6] C.-H. Chen, F.-M. Lin, H.-T. Hu, and F.-Y. Yeh, "Residual Stress and Bow Analysis for silicon Solar Cell Induced by Soldering," 2010.

[7] "NISA-II User's Manual," vol. 7.0, ed. Engineering Mechanics Research Corporation, Troy, Michigan, 1997.

[8] "Display-IV User's Manual," vol. 10.5, ed. Engineering, 2001.

[9] G. Vasudeva, "Finite element analysis: A Boon To Dental Research," vol. 6, 2008.

[10] R. C. Hibbeler, *Mechanics of Materials*, 8 ed. Singapore: Pearson, 2011.

[11] P. Widas and P. Briozzo, *A Brief Introduction into Finite Element Analysis as Part of MECH2400 5400*. Virginia Tech Material Science and Engineering, 1997.

[12] K. Ho-Le, "Finite element mesh generation methods: a review and classification," vol. 20, 1988.

[13] L. Kilchermann. (2013, 20 November 2013). *What Is Mild Steel?*