Vibration Control in Human Powered Vehicle

Muhammad Fadhli bin Ab Wahab¹, Dr. Hosseini Fouladi¹*, Dr. Satish Namasivayam¹*

¹School of Engineering, Taylor’s University, Malaysia
*Corresponding email: thereds15@yahoo.com

Abstract— This project was about the study of the effect of vibration on the Human Powered Vehicle (HPV). The amount of the HPV’s vibration must be comply with the standards given by the SIRIM. The type of analysis used in this study was numerical analysis, which was done using ANSYS software to conduct the simulation on the chassis of the HPV. At the end of the project, two factors were decided. One of them was the most suitable type of suspension to be used while the other one was the most suitable position of the suspension on the HPV.

Keywords— numerical analysis, ANSYS, vibration

1. Introduction

Human powered vehicle (HPV) is defined as a vehicle that operates using human muscle power (Fehlau, 2003). The purposes of this type of vehicle are environmentalism, lower cost, exercise and leisure. One common example of HPV is bicycle (Little, 2006).

The project about the human powered vehicle is developed during the 3rd year in mechanical engineering degree programme. However, it was found out that the produced HPV still has many aspects that can be improved, such as it vibrates when driven through a rough surface of the road. In order to improve its overall quality, the impact of vibration on the HPV must be studied.

The objectives of the project were:

- To conduct the numerical analysis on the chassis of the HPV using ANSYS software.
- To apply the knowledge gained from the results of the numerical analysis and comply it with the standards given by SIRIM.

2. Methodology

At the start of the project, the first step was to select the suitable parameters and boundary conditions to be applied. The area of study in this project was to focus only on the chassis of the HPV, neglecting other parts such as the tires and the pedals. The purpose of this was to make the analysis become simpler and easy to understand. The 3D model of the chassis was shown in Figure 3. The type of material used for the chassis was structural steel and its total weight was 15 kg. The type of meshing used was 3D meshing and the size of the meshing was set to be medium. This type of meshing was used since it was suitable for a solid model and also considered to be sufficient enough to provide the detailed result. The flow chart of the project was shown in Figure 2.

After that, the modal analysis of the HPV’s chassis was conducted. The objective was to check whether the chassis vibrates near its natural frequency when it was driven on a rough road surface. The damper was not involved during the first analysis. The results obtained shown that there were few natural frequencies that the chassis encountered when it was driven. So a damper must be applied to the system in order to prevent the HPV from vibrating heavily. This can be done by shifting its natural frequency outside from its working frequency range. This analysis also were being repeated several times until the most suitable amount of damper was decided to be used.
The data collected from both the structural and modal analysis will be compared and analysed. The information gained from the data collected, such as its behavior was explained in detail. When the numerical analysis was carried out, it was essential for the analysis to be within the scope so that the project will not stray from its objective and should be able to achieve it at the end. The problems encountered during the experiment must also be stated and after that give some suggestions to improve the experiment in the future.

After completing all the studies, the suitable methods to control the vibration of the HPV are to be developed. There are many techniques of how to suppress the vibration. In this research, the most suitable technique chosen was to apply the bike suspension to the chassis. The bike suspension chosen also must satisfy both the structural and modal analysis so that the amount of vibration can be efficiently controlled.

3. Results and Discussion

3.1 Result of Static Structural Analysis

The result obtained shown that the maximum deflection of the HPV’s chassis (5 mm) occurred on the middle of the chassis, which was near to the position of the driver’s seat. From this it was known that the driver experienced almost the maximum amount of vibration the HPV experienced. Besides that, the amount of maximum vibration also did not comply with the standards given by SIRIM. So the safety and comfort of the driver was affected from this amount of vibration.

In order to reduce the vibration, damper was applied at the bottom of the driver’s seat. The most suitable damper chosen had a stiffness value of 200 GPa. After the damper was applied, the maximum deflection reduced to 2.5 mm, which was half of the original value. This amount of vibration the chassis experienced also complies with the standards so the driver’s comfort and safety were maintained in this situation. The minimum and maximum deflection that happened on the HPV was shown on Figure 4 below.

3.2 Result of Modal Analysis

From this analysis, it was shown that the chassis experienced 5 natural frequencies when it was operating in the range of working frequency. The working frequency was assumed to be within 1 to 200 Hz. So it experienced a high amount of vibration since it vibrated near to its natural frequencies.

The natural frequencies was shifted outside the range of working frequency by applying a damper to the system. The type of damper which was decided earlier in the structural analysis was applied first. However, there were still 2 natural frequencies that happened within the range of working frequencies. So the damper’s stiffness was changed to 250 GPa. There was no natural frequency that happened when this damper was used, so this damper was the most suitable to be used in the system. This damper also were applied back to the structural analysis in order to check whether it affects the results. The maximum and minimum deflection of the chassis was shown in Figure 5 below.

4. Conclusions and Recommendations

In conclusion, it is essential to control the vibration of the Human Powered Vehicle (HPV) within the standards given by ISO. This is to ensure that the health and comfort of the user is well-maintained at all times. In order to do that, numerical analysis must be carried out first before producing the product. This is to understand more about the vibration experienced by the HPV, such as the source of the vibration, HPV’s natural frequency and also the amount of vibration itself.

In the future, it is recommended to conduct both numerical analysis and also experimental analysis. The purpose of this is to validate whether the result obtained from the numerical analysis is the same with a real life. In order to do this, the complete 3D model of the HPV must be used rather than just the chassis of the HPV so that both the results will be familiar and can be compared.

Acknowledgment

Firstly, I would like to thank Dr. Hosseini Fouladi who has guided me during the process of this project. Besides, I also would like to thank my parents who gave me so much support in terms of both moral and financial support. Last but not least, I also would like to give my thanks to Taylor’s University for giving me an opportunity to produce my own project’s paper.

References
