Simulation and Improvement of Vehicle Frame Using FEM

Wang Li-rui

College of Mechanical and Vehicle Engineering, Hunan University

Changsha, China

wanglirui91@126.com

Yang Xiao-long

College of Mechanical and Vehicle Engineering, Hunan University

Changsha, China

xyangsuc@163.com

Abstract—In order to develop and improve the design of a new type of vehicle frame, this paper illustrates the modeling and simulation of a new type of vehicle frame. Modeling of the frame was done via UG NX6.0, and simulation of the frame was done through ANSYS 12.0. Based on the simulation results, component was designed to strengthen the frame. Based on the results and improvements, the design was accomplished.

Keywords- vehicle frame; FEM; ANSYS 12.0; strengthening component

I. INTRODUCTION

Vehicle frame is to fix engine, vehicle body, steering components and transmission components, and is the key part of the vehicle. Frame with ideal structure will improve the performance of the vehicle significantly. Our previous study calculated the wheel base and wheel span of the vehicle, to be 1300 mm and 450 mm, respectively. Therefore, the frame should be designed within such limitations. To be compatible with the steering and transmission part, the width and length are 350 mm and 2000 mm, respectively.

Additionally, to reduce the weight of the frame, 6061 aluminum alloy, which contains Mg and Si, was used to manufacture the frame. Moreover, carbon fiber was applied in manufacture of the chair and other accessories to additionally reduce the weight.

Based on the work above, in order to assemble an eco-power racing vehicle and to join Honda Eco-mileage Competition and other relevant competitions, this paper used finite element method to simulate and improve the performance of the frame to accomplish the design which can meet the requirements of the vehicle.

II. Establishment of the Model

Based on our previous study [1], the requirement of the new frame, and according to our investigation of the market, we chose the 6061 aluminum alloy with the length, width and thickness of 38 mm, 25 mm, and 2 mm, respectively. The frame was then modeled via UG NX6 as fig 1.
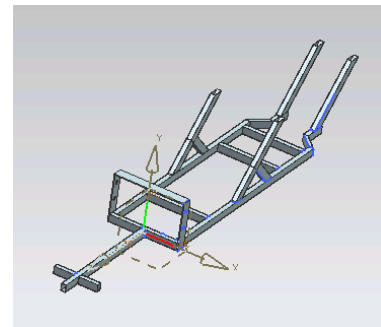

Fig 1 Frame model

In this model, the engine is put in the rear of the frame and the vehicle was driven by the rear wheel (RR). Moreover, to make the frame suitable for the tire with diameter of 305 mm and make the center of gravity lower, the bearing pedestal was designed in the rear of the frame on the beam of the steering part.

III. Definition and Application of the Load and Boundary Conditions

The load mainly includes the weights of the engine and the driver, which are 20 kg and 45 kg, respectively. Acceleration when braking is 2 m^2/s. Acceleration when steering was 6 m^2/s. The safety index was defined as 1.5 according to previous study. The load was applied to the frame where the engine was to be mounted. The sum of the wheel weight and weight of other light components were balanced against the weight to the engine (fig 2).
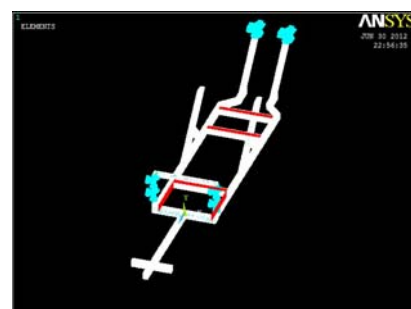

Fig 2 Static load and boundary conditions

Boundary conditions were restraints on the 4 holes where the front axle and bearing pedestal were to be mounted.

IV.Structural Analysis

A.Definition of Element type and Material

Import the model from UG NX6 and use solid 45 element to mesh the whole model. Solid 45 element is designed for 3D solid structure and was defined by 8 nodes. Every node has three degrees of freedom along x, y and z dimensions. This element has the character of plasticity, creepage, expansion, stress strengthening, large deformation and large displacement, and is appropriate for the isotropic materials. For these reasons above, solid 45 element can be used for the analysis of the vehicle frame. 6061 aluminum alloy is an isotropic material. Its Young Modulus is 68.9GPa, Poisson's ratio is 0.33, yield strength is 55.2MPa and fatigue strength is 62.1MPa.

B.Meshing of the vehicle frame

After the element type and material defined, mesh the vehicle frame. Use smart size 5 to mesh the whole vehicle frame and then refine the meshing at key positions. There are 280172 element s meshed in this process.

C.Applying of Load and Solution

The calculation of the static load is as following procedure: first calculate the space of the truss under the engine which is 15000mm^2, then the pressure on the tress is 13333Pa, the space of the truss bearing the chair is 20000 mm^2, then the pressure on the truss is 25000Pa. After applying the load(fig 2), solutions were obtained and plotted(fig 3).
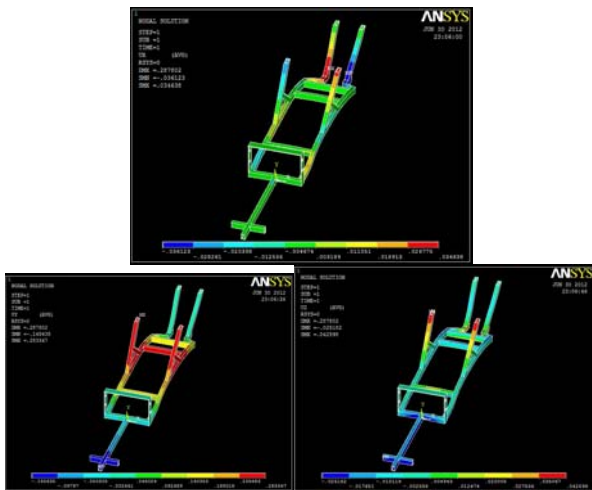

Fig 3 static analysis solutions

As we can tell from the simulation, the displacement from the X, Y, and Z dimensions of the frame are 0.03 mm, 0.28 mm, 0.04 mm, respectively. The stress of the frame along the X, Y, Z axes are 22.1 MPa, 16.1 MPa and 41.0 MPa, respectively. All of the displacements and stresses are within the limitations [1].

In the dynamic braking simulations, the acceleration was defined as 2 m^2/s. Therefore, the pressure applied on the truss bearing chair and engine are 19200 Pa and 7111.12 Pa, respectively. After the load was applied, including the weight and acceleration induced pressure on the frame, solutions were obtained and plotted (fig 4).
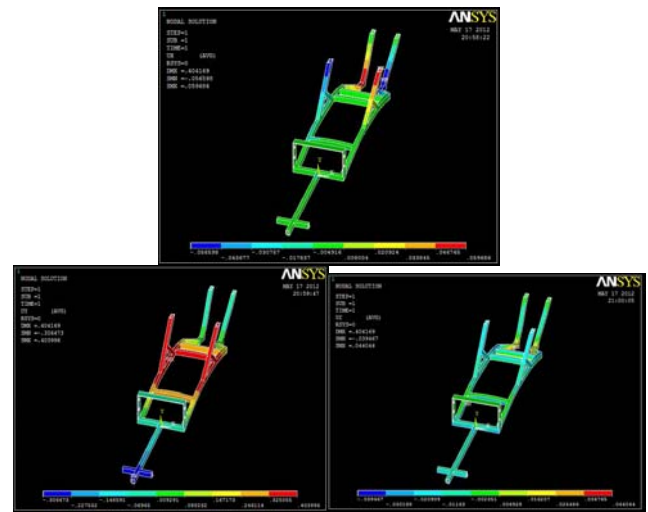

Fig 4 braking condition solutions

As we can tell from the simulation, the displacement from the X, Y, and Z dimensions of the frame are 0.059 mm, 0.40 mm, 0.044 mm, respectively. The stress of the frame along the X, Y, Z axes are 30.1 MPa, 22.1 MPa and 59.1 MPa, respectively. The maximum stress exceeded the material limitations which means improvements shall be made[2].

In the dynamic steering simulations, the acceleration was calculated as 6 m^2/s. Therefore, the pressure applied on the truss bearing chair and engine are 48000 Pa and 32000 Pa, respectively. After the load was applied, including the weight and acceleration induced pressure on the frame, solutions were obtained and plotted (fig 5).

Fig 5 steering condition solutions

As we can tell from the simulation, the displacement from the X, Y, and Z dimensions of the frame are 0.803 mm, 0.338 mm, 0.133 mm, respectively. The stress of the frame along the X, Y, Z axes are 37.1 MPa, 36.7 MPa and 60.3MPa,

respectively. The maximum stress exceeded the material limitations which means improvements shall be made[2].

V. Improvements

Sine the maximum stress exceeded the material limitations, strengthening components should be designed to strengthen the weak positions(Fig 6). Analysis of it indicates this component will make the maximum displacement and stress within material limitations.
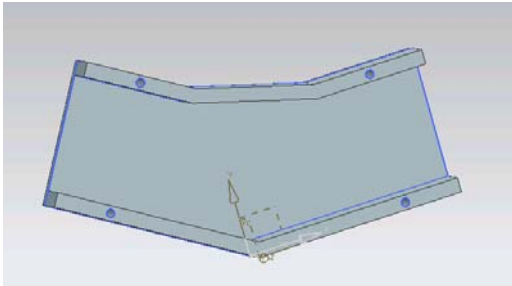

Fig 6 strengthening component

VI. Discussion

We also used Hyperworks to validate the model using another shell element type algorithm[3]. However, the validation shows a slight difference in the displacement in the three dimensions. Although the variation will not influence the structural stability and safety, further validation shall be done to verify the variations.

Moreover, since this vehicle, which is based on the frame, will join the competition to compete for the lowest oil consumption, the frame seems to be a little bit heavy. This means further improvement shall be made if more appropriate materials are available [4][5] [6].

The lightweight design and improvement should not be relied only on materials, rather, optimization of components and structure shall be considered when designing[7].

VII. Conclusions

Simulation using ANSYS 12.0 and Hyperworks has successfully validated the model in static conditions and dynamic conditions with the safety index in mind. Based on the solutions, new component was designed to strengthen the frame. After improvements above, all statistics, including displacement, stress and load distribution are within the material limitations. The model is appropriate for the vehicle to join the competition.

REFERENCES

[1] Zhou Jianmei, Wang Guijiao, "Lightweight Design of an Energy Conservation Vehicle Frame Based on FEA", J. Tianjin Auto, vol. 9, pp.72~74, 2008

[2] Wang Guijiao, Zhou Jianmei, "The Type Selecting and Lightweight Designing of Energy-saving Vehicles' Frame" ,J. Auto Technologies, vol. 9, pp. 41~44, 2008

[3] Dong Xueqin, Xin Yong, Yang Fan, "On Finite Element Modeling of a Vehicle's Frame Using Hyperworks", J. Mechanical Science and Technology for Aerospace Engineering, vol. 7, pp.906~908, 2008

[4] Gao Yunkai, " Analysis of a Vehicle Body", Beijing Institute of Technology Press, 2006

[5] Xia Xiaokun, Yan Fuwu, Du Changqing, "A Study on Reducing Oil Consumption of Energy-efficient Motorcyle", J. Motorcycle Technologies, vol. 10, pp.70~72, 2008

[6] Yin Huijun, Wei Zhilin, Shen Guanglie, "Finite Element Analysis on Frame of Trucks", J. Journal of Machine Design, vol. 11, pp.27~28, 2005

[7] Li Xiya, Li Chenggang, Hu Yujin,"General Conditions of the Finite Element Analysis Technic", J. Special Purpose Vehicle, vol. 1, pp.11~15, 2001