The Analysis and optimization of the structure of the supporting frame for a light electric vehicle.

K00273073
Liam Murphy
TUS

Project Aim

The aim of this project is the analysis and optimization of the structure of the supporting frame of a light electric Vehicle (LEV)
Other aims of this project include the improvement of safety of the driver of the LEV while making the frame lighter and more of an efficient design

Objectives

1. Improve the frame of the LEV with safety as the primary design constraint .
2. Create a 3C CAD model of the new design frame.
3. Carry out FMEA, FEA and topology analysis on the CAD model.
4. Based on simulation results introduce some corrections and possible improvements to the CAD design.
5. Prepare technical documentation of the project in the form of assembly drawings of the vehicle and its subsystems.

Background

The supporting structure is an essential part of any vehicle especially a LEV that will be used for high-speed racing so the need for the design of this component to be perfect is quite high. The idea of a Supporting structure has been around a long time originating from 1964 with John Aley having the first design. The main function of the supporting frame for the LEV is to give structure to the LEV but has other important functions besides that such as storing and protecting the internal components. One of the most important additional functions of the supporting frame is to protect the driver from any harm, that is the main of this project to create a design that will keep the original functionality of the existing frame but to add more passive safety for the driver. The design for this supporting structure will be based on the results from a topology study and so there are no concept designs.

Design Methodology

As discussed in the previous section there are no concept designs for this project as the final model is based only off the topology results. So , for the topology optimization a design space model that has the same overall dimensions as the final chassis, which have been obtained from the Formula SAE competition, must be made which is shown in the picture below.

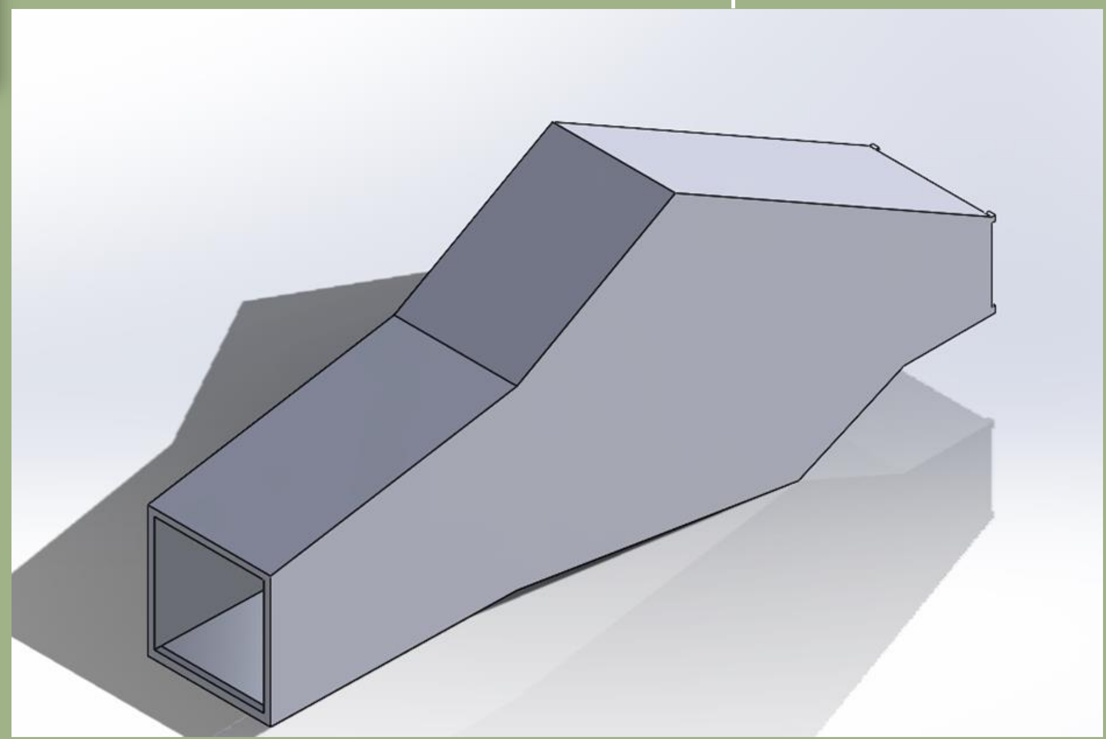

Figure 1: Design space model for Topology Optimization

After this all the relevant settings to complete the topology optimization method were inputted and the study was ran a total of four different times twice for steel and then twice for aluminium with both an 80% and then 75% mass reduction for both materials. As this would give the best results for a final design. The topology optimization solution that was chosen to be the final model was the 80% mass Reduction of aluminium as it represented a supporting frame the best and had the best geometry..

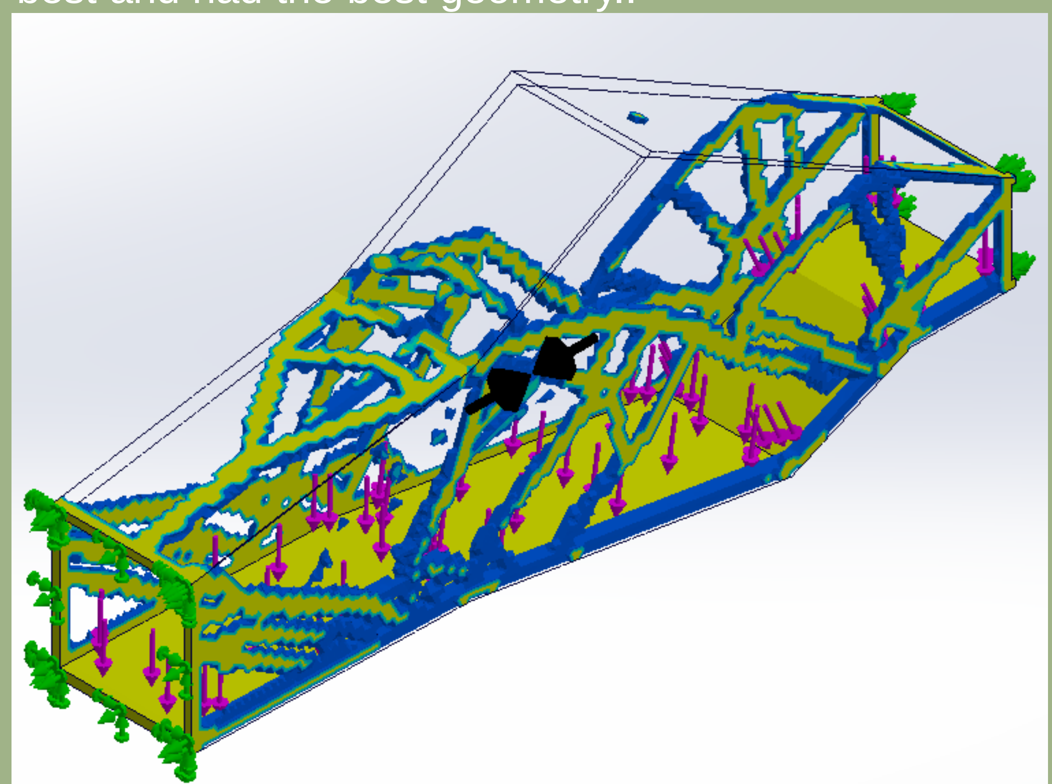

Figure 2: Final Topology Optimization Result

CAD model of final design

After the topology optimization results had been ran and the compared it was decided that the final design would be based off the 80% mass reduction and that the material that would be used would be aluminium. But as can be seen in figure 2 there is no halo design or any sort of protection for the head of the driver, so a common design was used to protect the driver.

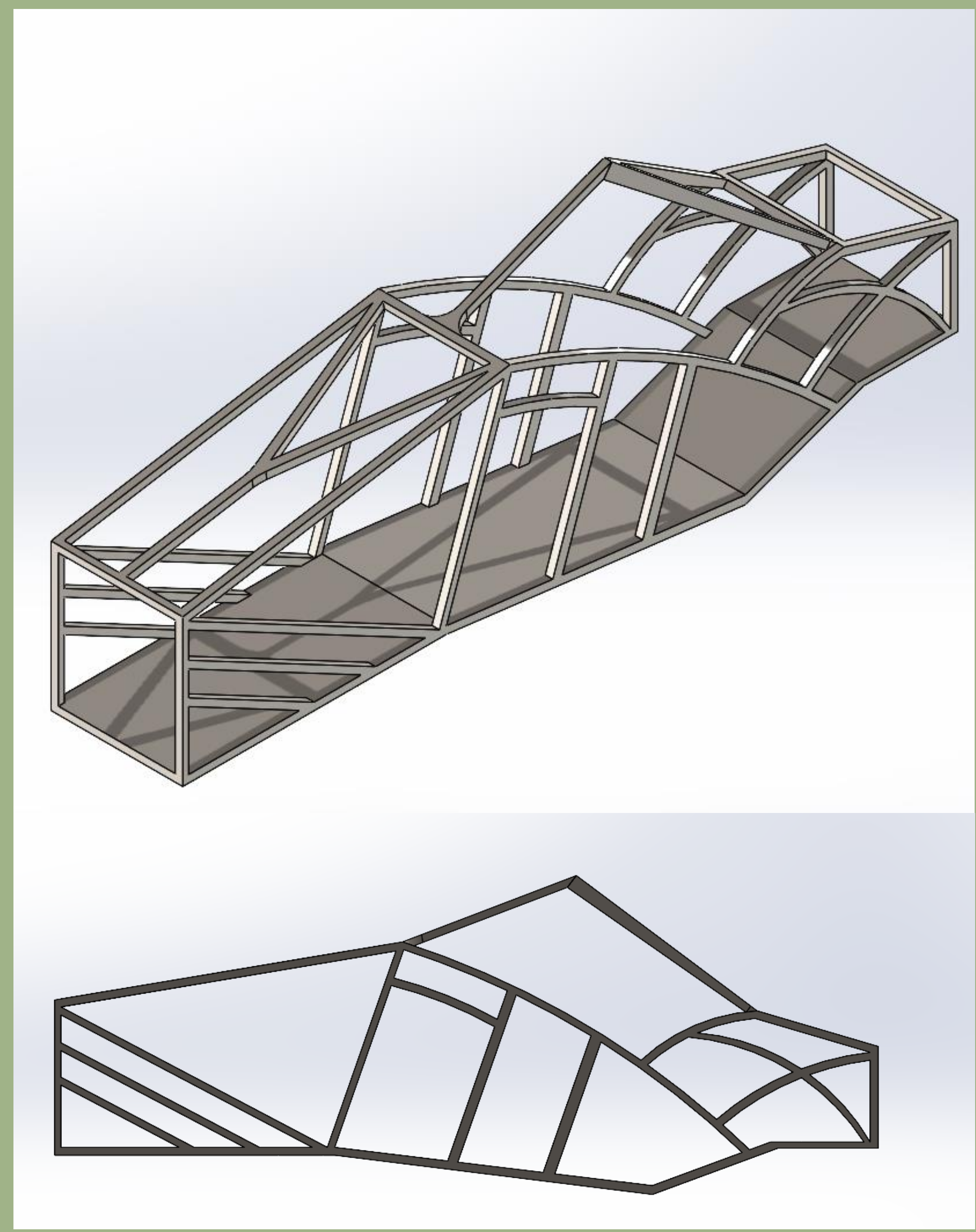

Figure 3: Final Design

The Frame as previously discussed will be made from Aluminium which is quite light having a density of approximately 2750Kg/m³. The overall length of the supporting structure is approximately 2050mm and the height is 880mm with the weight of the supporting structure being almost 90Kg it is almost half of the maximum weight allowed in the Formula SAE competition which is 200Kg (Not including driver weight). Also, the cross section of the beams is a rectangular shape instead of the previous circular cross section.

Virtual Prototyping (FEA)

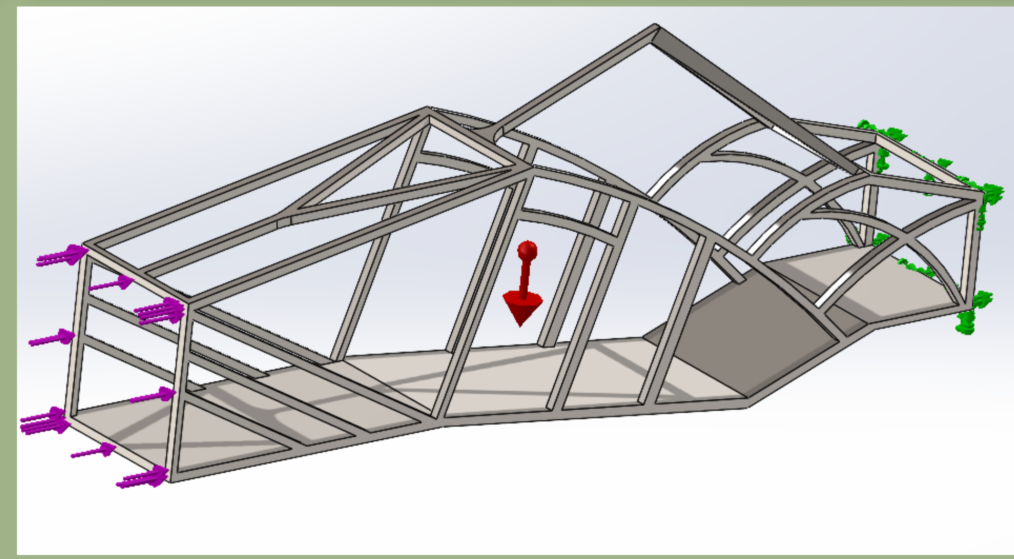

Figure 4: Boundary Conditions of FEA

Finally, the last part of this project ae the FEA simulations on the final design of the chassis to test the mechanical properties of the model. there were four tests carried out front, rear, side and halo crash impacts with impact of 1300N for the front and rear tests, 1800N for the side impact and 10000N for the halo impact. These values of force were calculated from weight of the chassis along with the other components in the supporting structure. Across all four tests the maximum displacement was 1.112mm which was on the halo test this might seem like a lot but comparing to the overall height it deformed by less that 0.15% of the height of the supporting structure. The minimum displacement was approximately 0.088mm which an excellent result.

Conclusion

Comparing all results of the virtual prototyping (FEA) it can be seen that the final design passed quite well and did not fail any tests and so this proves that the topology study did indeed provide the best and most efficient design possible for the supporting structure. The final design that was created did met the aims and objectives of the project and the design was completed without any problems or issues.

Acknowledgements

I would like to acknowledge and thank my supervisor Miroslaw Mrzyglod for the work that he helped me with over the course of the 13 weeks of this project.