MECHENG 4Z03
Anish Paramsothy, paramsa, 400300097

Design Project: Remote Controlled Vehicle

MECHENG 4Z03, Design Report, McMaster University

Date Submitted: April 6th, 2025

Due Date: April 10th, 2025

Contents

1 Introduction

Figure 1: Final Submission

The over-arching assignment of the MECHENG 4Z03 course was a design project in which students were tasked with designing a toy remote controlled vehicle that implemented a battery and motor component precreated for them to use. The purpose of the project is to gain experience in top-down design using most of the components learned in the course.

Various tools used in the project include:

- Surface Modelling
- Assembly Modelling
- Gear Creation, Gear trains, and Assembly Motion Mechanisms
- Material Analysis, Finite Element Analysis
- CAM and CAE

Overall, the goal is to take what we had learned and use our creativity to make something cool.

This report discusses various aspects of the course, such as my design choices for each part of the assembly, stress simulations, motion studies, CAM simulations, and optimizations made to the design using CAE.

2 Initial Design

2.1 Initial constraints

Using my MacID, **paramsa**, I was able to define a couple of constraints for my vehicle design.

- "P" ⟶ Motor location in the middle of the vehicle
- "A" ⟶ Gear ratio numerator 12
- "R" ⟶ Gear ratio denominator 40
- "A" ⟶ Body shape Car

With these design constraints in mind, we can move on to the actual planning of the model.

2.2 Planning And Design

I had initially planned to go for a more iconic model of car, since it would something I could enjoy creating. At the start of my planning, I decided on going with a similar design to Doc Hudson from the *Cars* series, which is modelled after the Hudson Hornet made in 1951 by the Hudson Motor Company. The design is also made to be a toy car, so the choice feels very applicable for this project. Along with this, the sleek design allows me to showcase the skills learned in this course, such as surface modeling of the shell. The larger raise in the middle of the vehicle also gives me room to play around with the motor, and the abundance of space that the shape provides benefits me in designing a working assembly.

I want to note that since I based my model off a pre-existing car, the shell of the vehicle used a side profile image of Doc Hudson, just to get the proportions. All dimensioning was decided by me and was relative to the motor and battery.

In order to create this model, surface modelling techniques were used, which were centered around the height/distance from the front of 3 points: the hood to the front window, the front window to the end of the back window, and then the end of the back window to the end of the vehicle. These were then lofted together as surfaces and then stitched together in order to get the best possible surface acheivable in the complex model.

See the figure 2 for the initial surface design proces.

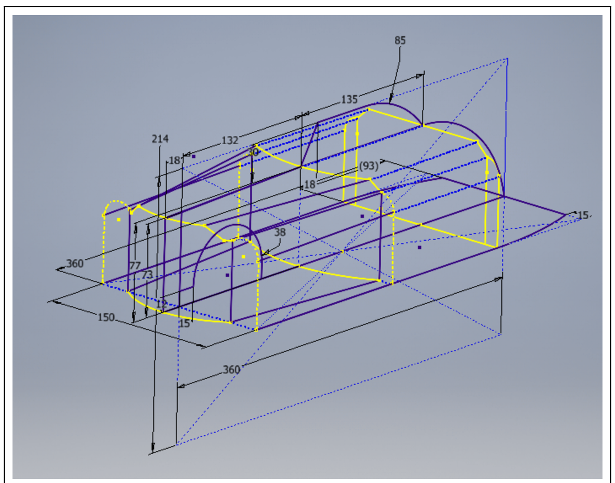

Figure 2: Initial Surface Modelling

After thickening the surfaces, the shell was created.

2.2.1 Design Choices

Some design choices made were to include some press/friction fit parts. Most connections in the entire assembly include bolted connections for security, but there was friction fit for connecting the base to the shell. This was for the simplicity in manufacturing, and there are small mm offsets between the hole and rod sizes for the connection in the case of error from the tolerancing. This consideration was useful for those that may be making the toy car (kids).

A secondary choice made was the use of a worm gear in order to convert rotational motion between axes. Since the choice of gear type was not constrained or limited, I thought that trying something I have not done before would be great for improving my skills. The worm gear is able to efficiently use space in comparison to bevel gears, so I thought that it was the perfect fit for the design. Note that since the worm prevents the actual worm wheel from being the driver, it means that the gear can essentially lock the movement of the car to only be controlled by the motor, which is a neat bonus to the functionality of my car.

2.3 Debugging and Optimizations

This design had been optimized several times before creating the final shell. A noteable fix made was reducing the number of surfaces that were required to be thickened separately. The initial issue was that there were several errors when trying to thicken the main surfaces together, which is a result of an abundance of incomputable or meshing surfaces that the system could not understand. The failed design can be seen in figure 3.

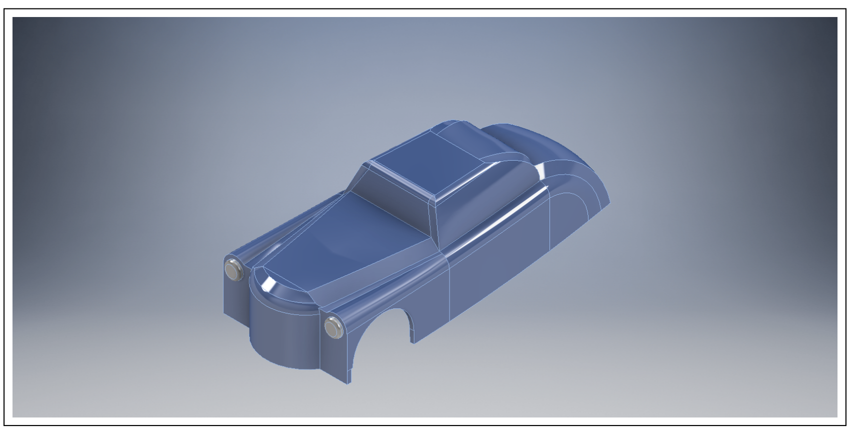

Figure 3: Failed Initial Surface Modelling

In the final design, the main surfaces of the shell were accomplished in one main thicken, and then a final one for the front section.

The final design included recreating the windows as separate bodies on the shell, which could then have their materials and finish changed to something else. The final design is presented in figure 1.

As a note, a complete overhaul of the original design was needed, which was originally accomplished by extruding the side profile and “chiselling” away at the solid body with sets of cut extrudes. This was extremely inefficent and did not produce the intended output. This design also had several issues when using the shell command, which was a result of the various intricate surfaces created unintentionally from all the extrude cuts. See Figure 4 for the failed design.

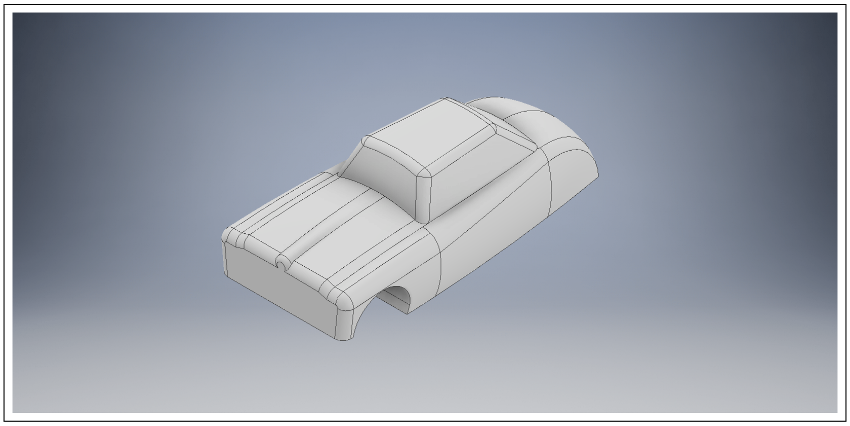

Figure 4: Failed design project shell

3 Simulations

3.1 Stress Analysis

A finite element analysis (FEA) was completed on the front axle of my car, as it was one of the components that would experience the greatest amount of stress. This is because it transfers the weight of the base and gear to the wheels. Because of this, the force was placed on the middle of the axle angled downwards, while the square portions of the rod were fixed for testing purposes.

Analysis of these components of our design ensures that the key connections can withstand stress and continue functionality, while also determining if there are any changes that need to be implemented. For this study, a 100N downwards force was applied, which essentially is a 10kg mass subject to gravity; an extreme case for the weight distribution. The material used was PBS plastic.

After taking the various tests at different element sizes, I developed the following plot and graph to showcase the results:

MATERIAL: PBS Plastic

Test Number	Average Element Size (fraction of bounding box length)	Minimum Element Size (fraction of average size)	Grading Factor	Maximum Turn Angle	Create Curved Mesh Elements	Mesh Size [Elements]	Force N	Von Mises [Min] MPa	Von Mises [Max] MPa	Deflection mm	
1	1	1	1.5	60°	Yes		100				NOT NEEDED
2	0.5	0.5	1.5	60°	Yes		100				NOT NEEDED
3	0.2	0.2	1.5	60°	Yes		100				NOT NEEDED
4	0.1	0.1	1.5	60°	Yes	1866	100	0	11.77	0.06047	
5	0.05	0.1	1.5	60°	Yes	1420	100	0	10.86	0.0604	
6	0.025	0.1	2	60°	Yes	2617	100	0	11.93	0.061	
7	0.015	0.1	2	60°	Yes	9461	100	0	13.05	0.06095	
8	0.01	0.1	2	60°	Yes	25064	100	0	14.91	0.06092	
9	0.008	0.1	2	60°	Yes	50623	100	0	19.12	0.06101	
10	0.004	0.1	2	60°	Yes	287733	100	0	24.67	0.06121	

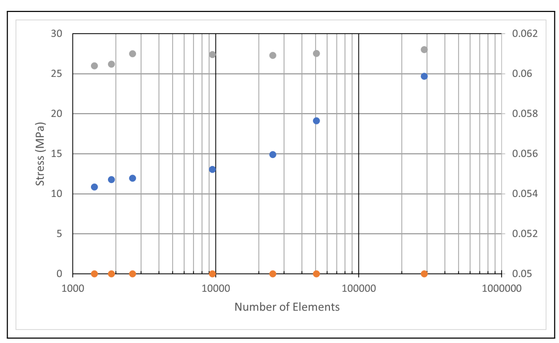

Figure 5: FEA Plot and Table

As seen in 5, the max stress does not converge, while the deflection does converge. This signifies that there is a weakness in the shape or form on the axle, and when looking at the pictures of the analysis in Inventor (figure 6), there is a noticable weakness in the square and circular portion of the bar near the ends. A potential fix to this issue is increasing the surface area of both shapes, while also allowing for a larger gap between the circular rod and square extrusion. Another potential fix would be to fillet the connections between these two parts of the bar, so that the stress is spread troughout the shape.

The test screenshots can be seen below. *note that the views were adjusted to* *<u>Adjusted 0.5x</u>* *to show the visible deformation that could occur, even though it is not actually happening to this degree.*:

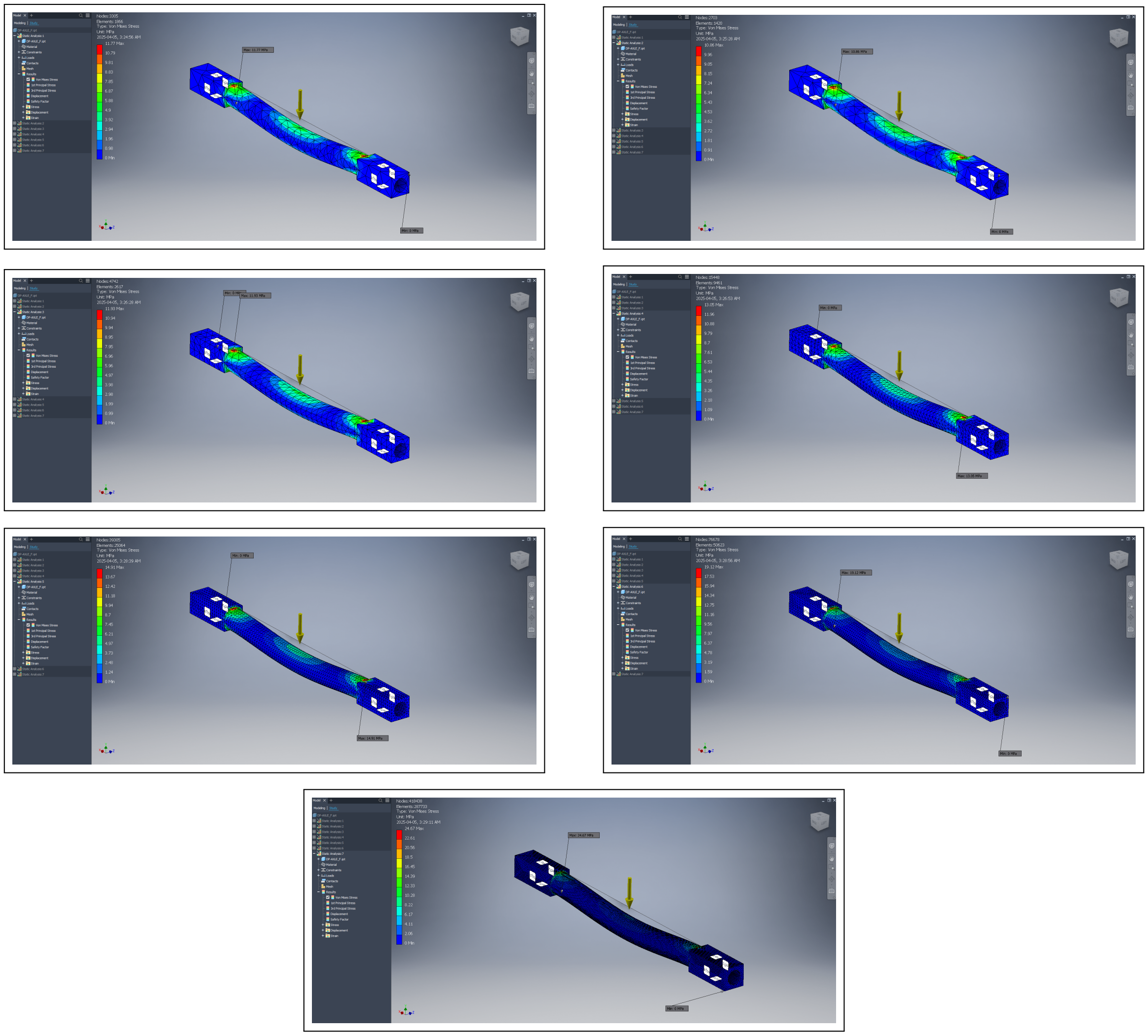

Figure 6: FEA tests

Overall, this shows that there can be several optimizations for this car already, which is helpful to know for future projects.

3.2 Motion Study

A motion study was completed, which is essentially driving the vehicle through the main drive shaft to make sure the components connected to it rotate correctly. A video was taken for this, which can be seen in the `DP-MOTION_STUDY.wmv` file in the folder. The drive shaft is driven through the motors rotation, which is then fed straight to the worm, rotating the gear and the axle connecting to the wheels. The car is FWD, but the back wheels were kept rotating in accordance to the front wheels, and not based on a connecting axle..

3.3 CAM Simulation

For the CAM simulation, additive manufacturing of the base in PLA plastic (direct drive) was used, as this was most likely going to be how the part would be created. Realistically, for toy car parts, going ahead with plastic and a simpler approach to manufacturing such as 3D printing would provide the most benefits.

During the creation process in Fusion 360, a 3D printer needed to be selected. Note that for the base, since it is bigger than the Anycubic I3 Mega base plate, a larger printer was required. The **Anycubic Chiron** was chosen as a replacement since the surface area of the base plate was nearly double.

Supports were required for the base, but since it was mostly flat and was printed it in its regular orientation (top face facing up), it only required supports in the few holes left for conections to the axle and worm gear. Something to note is that the bottom chamfers did not require supports, which is because the change in angle moving upwards is not significant enough to require supports.

See figure 7, which demonstrates the Fusion 360 additive manufacturing preview. Please look for the `DP-BASE_CAM.f3d` file.

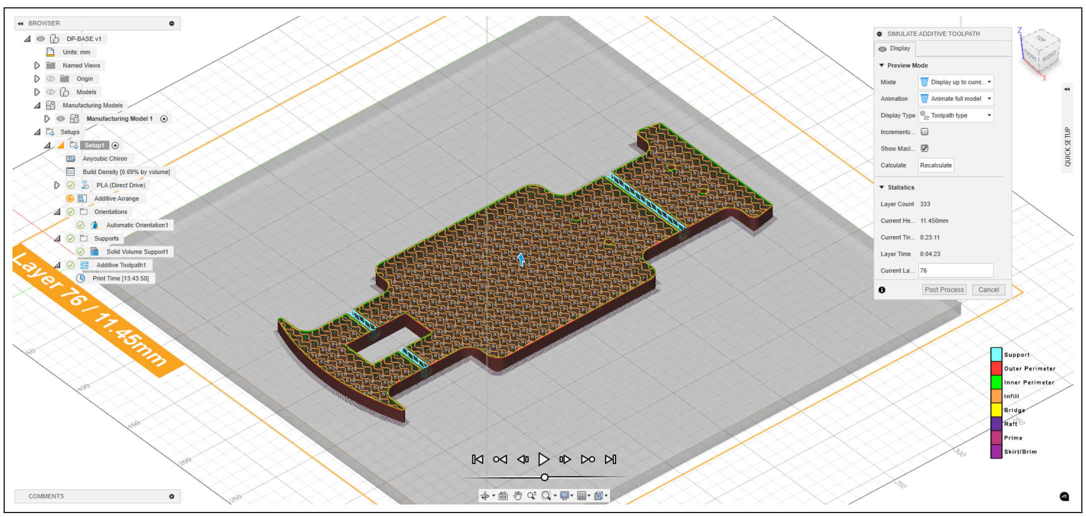

Figure 7: Additive manufacturing simulation

As we can see, supports are being generated in the hole. The total print time is 13:43:58. Taking the size of the base, and the several intricies of the part, the print time accuratelty describes the fabrication of the part.

3.4 CAE Simulation

The last simulation was a CAE optimization simulation, which would take one of the solid bodies created for the car and optimize it by evaluating what prexisting portions of the model are no longer required for the part to remain functional. In order to correctly simulate this, specific shapes on the design need to be preserved while the force is applied. Since the base of the car was chosen for optimization, these reserved features can range from the holes preserved for screws, or the height at which the bar attached to the worm will sit, or the open space allocated for the worm gear.

On top of this, forces need to be applied to the base so that the optimization can run, since it optimizes according to the areas of stress.

Figure 8 illustrates the initial set up for the optimization, which uses PBS plastic as the material.

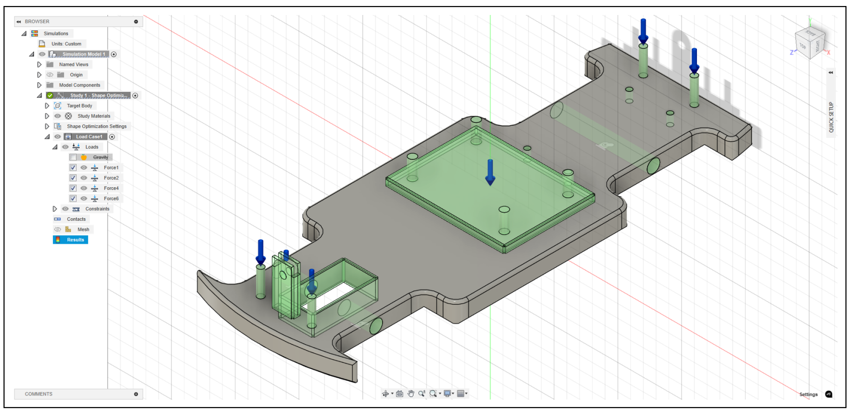

Figure 8: Preseved shapes and force application

This setup can now be optimized, which is shown in figure 9. Please look at the DP-BASE_CAE.f3d file.

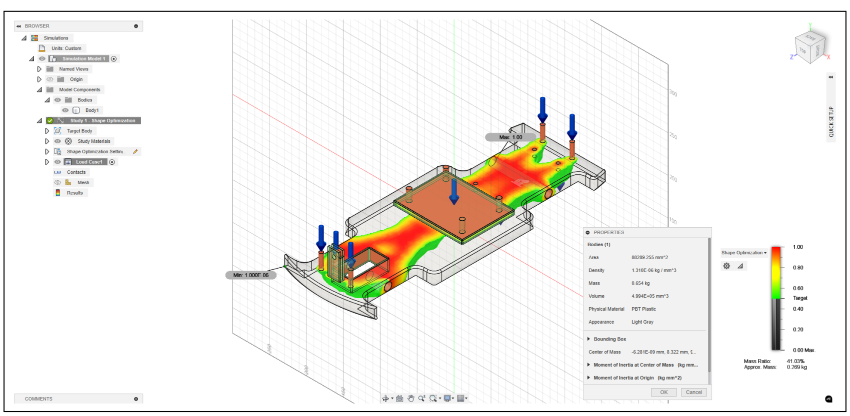

Figure 9: Solved optimization

Looking at the optimized body, most of the original surface was removed, as the mass was reduced from 0.654 kg to 0.289 kg. The original design philosphy was to create the base surface to fit the shape of the shell. This newly optmized design removes most of the excess material that was not required for the core functionality of the design. Something to note is when designing parts, it is important to evaluate the tradeoff between design aesthetic and practicality. Personally, the removal of the front and back portions of the design can mantain the aesthetic that was intended, while also reducing material. Trading the original idenitity of the base in order to optimize the material usage is more important when the material usage is much larger, but it is still something to heavily consider if the part was to be manufactured on a more grand level (frabrication of several thousands of this base).

The parts that were excess were removed and can be seen in figure 10.

Please look at the `DP-BASE_CAE_OPTIMIZED.f3d` file.

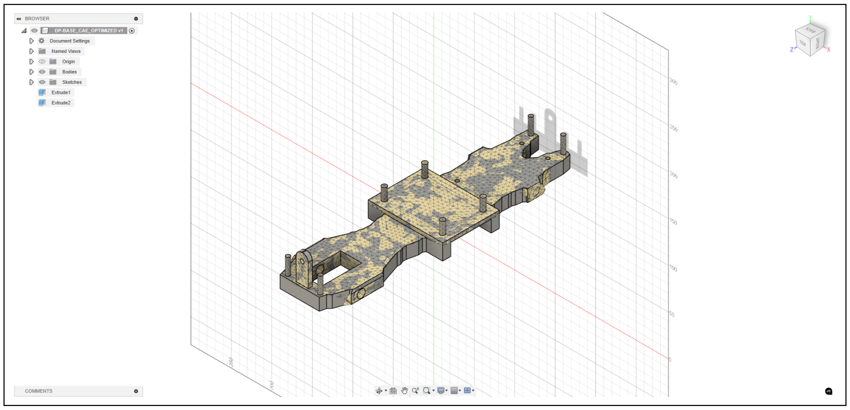

Figure 10: New Base

Overall, the optimization makes sense, and can be changed to this while maintaining functionality.

4 Conclusion

The design project was a great application of the several skills developed through the course, and forced me to think critically about the design in the early stages of planning so that I would not run into problems later on. It was crucial to evaluate how several of the connections between parts would work, and what this would mean for the state of my current design. There are several changes I could see making to the design in the future, but that would require more analysis at the moment. The simulations studies that were conducted helped with understanding core issues with my design, while also presenting me with new ways to optimize and think about the functionality of my design.