UNIVERSITY OF BIRMINGHAM

Computational Fluid Dynamics

Mec[illegible]eering [illegible]ng

Mod[illegible] co-[illegible]ina[illegible] Dr Leu[illegible] Soo & Daniel Espino

Re[illegible] William [illegible]nyema (ID no.: 1826926)

Pers[illegible] Tutor: Dr Raya Al-Dadah

Abstract:

This report explores the use of ANSYS CFX to perform a computational Fluid Dynamic (CFD) analysis on a vintage car, which was designed using SolidWorks. The chosen car is a Model T Ford. From the results obtained by the analysis, improvements to the design were created and another CFD analysis was performed. Both results were then compared against real-life values to validate the values from the simulations.

Introduction

The Model T is a vintage car, which has poor aerodynamics due to its box-shape and large frame. Therefore, the manufacture has commissioned a new model which shows the improvements made on the air pressure distribution, minimisation of eddies and ultimately decreasing the drag of the vehicle. To demonstrate this, Computational Fluid Dynamics (CFD) was used to analyse the aerodynamics of the original model, then improvements were made onto a new model, and analysed to be compared. Both models were created on SolidWorks and analysed using ANSYS CFX.

Geometry

The geometry of the model T was determined using the actual drawing and dimension of the Model T (Figure 1). When designing the model on SolidWorks, the model was simplified by removing negligible parts. The roof was altered, as the design was convertible, it was simplified to be a solid roof. The elimination that were made, include parts such as the door and interior of the vehicle. This also reduces the meshing complexity of the simulation and will also quicken the solving time of the mesh with no significant impact to the results.

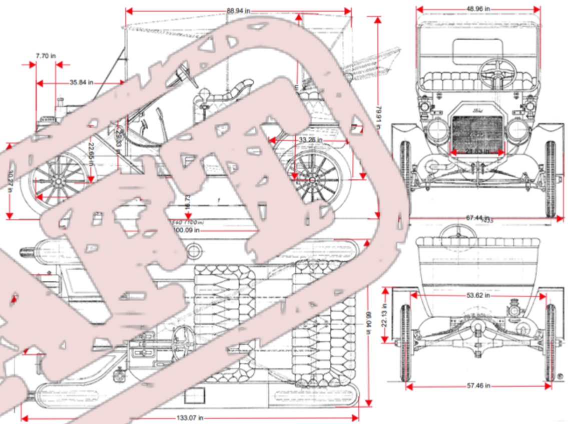

Figure 1. Drawing of the ford model T

Mesh, Solver & Related Computational Parameters

The computational domain was created using SolidWorks and then transferred into Ansys to simulate the physical air flow in a wind tunnel. The parameter of the enclosure was determined through

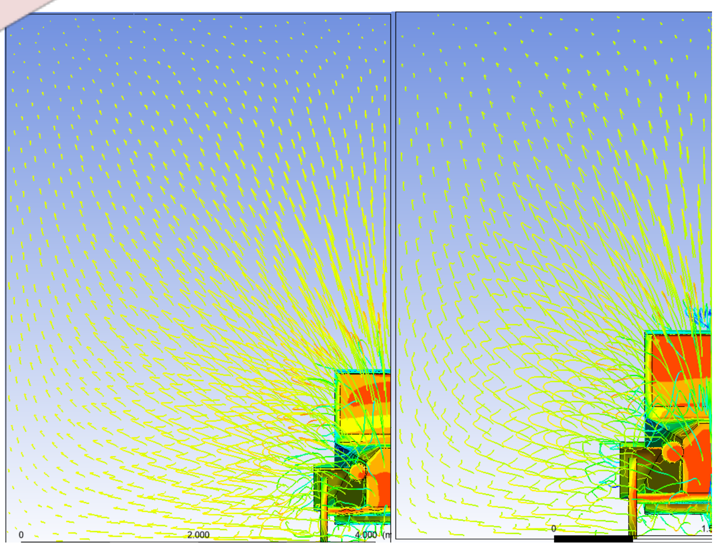

Figure 2. New computational domain (left) and Old computational domain (right)

using guidelines stated by Lanfrit [1], which stated that the domain should be approximately 3 car length in front of the car, and 5 lengths behind. The height and width were chosen so that the free-slip conditions did not affect the flow on the car. When performing the first iteration for the CFD analysis, it was found that the enclosure was too small creating the venturi effect. This resulted in a 14.2% increase to the drag force on the car. Therefore, the height and width were revised and increased to 3 times the height of the car and 5 times the car width. SolidWorks was also used to subtract the enclosure from the car. The design including the enclosure is halved, this is because the design is symmetrical therefore the mesh will the symmetrical as well. This will reduce computational time and more elements can be used as to make a much better mesh.

Subsequently, a mesh was automatically created using Ansys. The initial quality of the mesh created was low producing only 367579 elements. Therefore, to improve the mesh on the car for better air flow, an inflation was used. A Program Controlled automatic inflation was selected with the option 'First Aspect Ratio' on the car. The reason was because it produces layers of high-quality prismatic mesh around areas where turbulent flow excessive. This will also decrease the density form around the enclosure where laminar flow is more present, keeping the number of elements the same. The growth rate was kept at default, but the layer was increased to 10 layers. To further improve the quality of the mesh, face sizing was applied to the car decreasing the element size to 0.08 m which still obeyed the limit of 512k nodes. Furthermore, a volume control box was sketched around the vehicle where the elements can be limited to a size of 0.075 m, using the body sizing function.

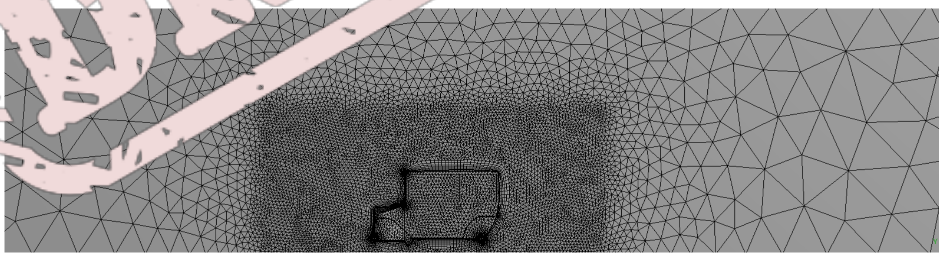

Figure 3. the final mesh created for the initial design

Fluid properties & boundary condition

Boundaries were created for all the selected names. The fluid coming through the inlet is set to be travelling at a speed of 42mph which around 18.8ms^{-1}, this because the top speed was around 40 - 45 mph for the model T [2]. For the outlet boundary settings, the Average static pressure was set to zero. The walls are 'free from slip' which means that there is no friction between the wall and the fluid. Final, the boundary for the car was set to 'no slip' which means the fluid velocity is zero on the car. The fluid used is air with properties of 1.229 kg/m^3 for density at 15°C, with a dynamic viscosity of 1.73 x 10^5 kgm^{-1}s^{-1} [3]. The final change was the initialisation, which was set to 42 mph in only the U direction. The realizable k-epsilon turbulent model was used as the solver to simulate mean flow for turbulent conditions [4].

Results

The drag force is measured using the function calculator, this is the force acting on the car as air is flowing at the opposite direction. The force that was calculated is 439.314 N. Subsequently, the cross-sectional area of the car which is 2.53 m^2 was found using SolidWorks, which is used to obtain the coefficient of drag C_D. The equation is

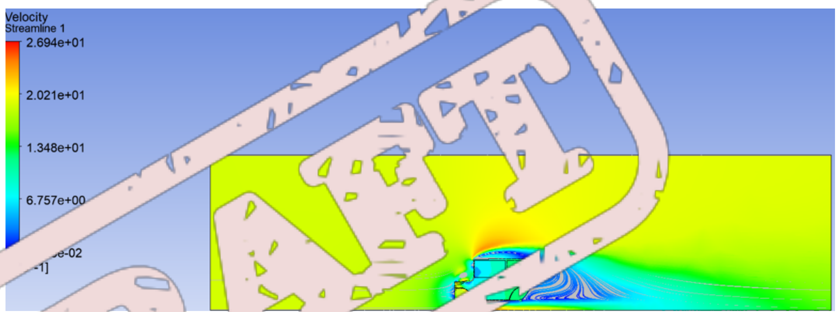

Figure 4. velocity streamlines on the initial design

$$C_D = \frac{2F}{\rho A U^2} = \frac{2 \times 439.314}{1.229 \times 2.53 \times 18.8^2} = 0.8$$

Equation 1 shows the equation used to obtain the C_D, where F is the drag, A is the cross-sectional area, ρ is the fluid density and U is the speed of the fluid. It also shows the relatively high C_D found on the model T.

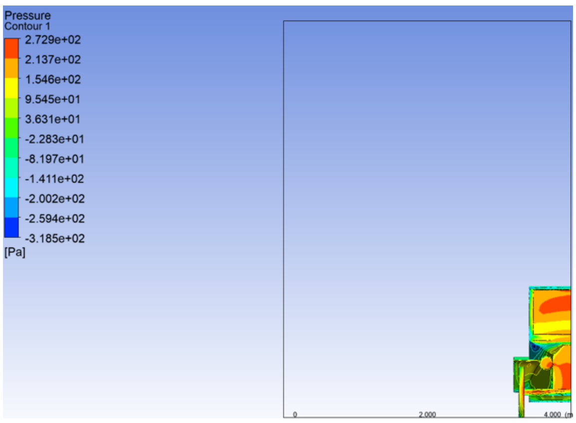

Figure 5. the pressure drag on the initial design

Using Ansys, I was able to analysis the results found when running the program. Creating point cloud starting from the inlet allowed me to create velocity streamlines shown in figure 4, where the Air velocity at the front is 19.417 ms^{-1} at the front, 22.725 ms^{-1} at the top and 11.352 ms^{-1} at the back of the car. Figure 4 also shows air being recirculated at the front and rear of the car, which represents turbulent flow of the vehicle. The change in pressure and velocity at the edge of the rear vehicle, is the cause of tripping of the boundary layer, forming eddies which is shown in figure 4. However, there is laminar flow seen at the beginning and end of the enclosure means that the parameter of the domain is suitable. Using the contour feature, the pressure change can be determined which is shown in figure 5.

Redesign, Validation and Recommendation

A redesign was made to reduce the coefficient of friction by incorporating more aerodynamic designs to the car. The windshield of the car was slanted firstly, this allowed air to flow smoothly over which is represented in figure 7, the velocity streamlines are flowing over the car. The high pressure drag that is experienced at the flat windscreen on the initial design is also drastically decreased due to the curve shaped windscreen shown in figure 8. This is because air is able to escape allowing the area to slip. The front of the car bonnet was slightly declined, and the flat facing shape of the bonnet was redesign giving it a triangular shape. The decline of the bonnet again allows for smooth transition of air flow, this change allowed the turbulent flow at the front change to transitional flow seen in figure 7. The triangular shaped bonnet allows the air to be sliced, letting air slip out to the side of the car eventually decreasing the pressure drag at that specific area. Both the roof and the bumper were slanted creating an Aerofoil shape at the back of the car. This effectively

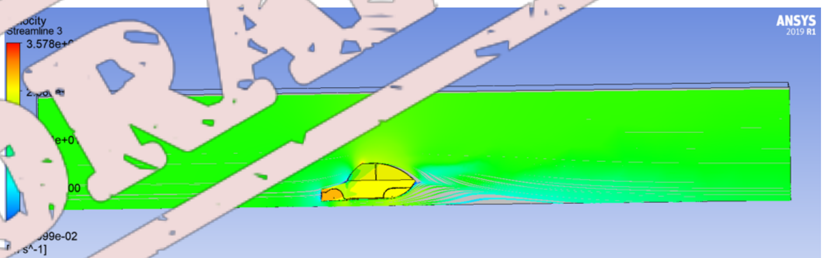

Figure 7. velocity streamlines on the redesign

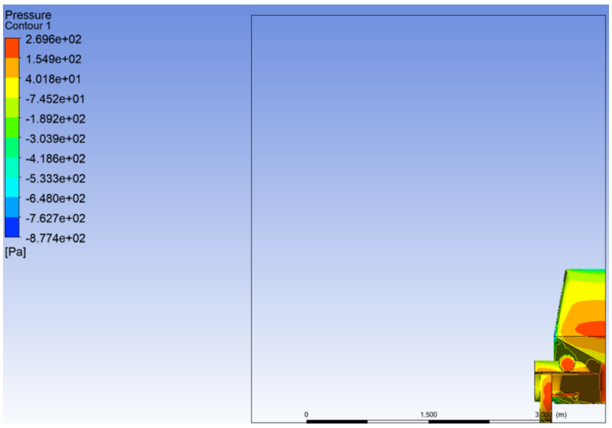

Figure 8. the Pressure drag on the initial design

minimises the separation of flow seen in figure 7. Turbulent flow is almost eliminated at the back of the car and fewer eddies are visible in figure 7. The car had to be made longer because the changes performed to it compromised the capacity of the car, therefore the new length of the car is 4.01 m. There are some little changes done to help the redesign such as creating new fenders. However, the new fenders due help with parting the air flow in a smooth manner underneath the car. The tyres are larger than the initial design to compensate the additional weight of the car. The top width of the car was also lightly reduced, decreasing the cross-sectional area to 2.284 m^2. Finally, any edges were chamfered just to faintly help the flow of air.

A CFD simulation was performed on the redesign using the same quality mesh and boundary condition used on the initial design. The new force acting on the redesign was 341.2 N, using this value and the new value of the cross-sectional area of the redesign the C_D for the new design is 0.685. There is 13.8% reduction of the C_D, meaning that the redesign has effective improved the initial design. In addition, the initial design had a lift of 192.4 N, where the redesign achieved a lift of [illegible] N. There is [illegible]% reduction, this provides downwards force which gives the car more traction control making it easier to handle the car. Still, there is not a significant reduction; this may be due to increasing the length which increases the weight of the car. The slight decrease to the width of the car is also a factor why the drag was only slightly improved. So, further improvement that could be done in the future is an increase to the area of the car and more aerodynamic designs to compensate the length of the car. Based on the CFD analysis, performed on the initial design and the redesign the results show that the use of aerodynamic design has reduced the model's coefficient of drag, making the redesign an excellent recommendation to replace the model T.

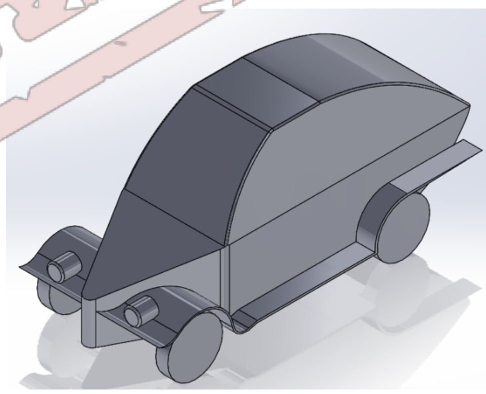

Figure 6. The redesign

Reference

1. Lanfrit, M. (2005),' Best practice guidelines for handling Automotive External Aerodynamics with FLUENT'. Available at: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf
2. Ford model T (2020). Available at: https://en.wikipedia.org/wiki/Ford_Model_T
3. Hall (2015),' Air properties definition'. Available at: https://www.grc.nasa.gov/www/k-12/airplane/airprop.html
4. k-epsilon turbulent model (2019). Available at: https://en.wikipedia.org/wiki/K-epsilon_turbulence_model
5. Austin, R, 'Formula One racing'. Available at: http://www.formula1-istanbul.com/7/how-do-these-three-forces-affect-a-formula-one-racing-car

DEPARTMENT OF MECHANICAL ENGINEERING

Small-Scale Engine

Mechanical Design A Project

Version [illegible]

Department of Mechanical Engineering
The University of Birmingham

PROJECT BRIEF

Your design consultancy has been commissioned to design a small-scale two stroke, compression ignition engine for use in an expendable Unmanned Arial Vehicle (UVA) to be used in search and rescue missions. The engine must power the aircraft for least 15 minutes of flight

The engine must be capable of producing around 0.4 brake horse power (bhp) or $300\ W$ at a speed of approximately $16{,}000\ rev\ min^{-1}$. The engine, which should be air cooled shall be capable of driving a propeller of diameter $200\ mm$ x $100\ mm$ pitch at the required power and speed. The drive shaft should be a minimum of 5 mm in diameter.

The engine should be flexible and easy to start and operate. Thus it is suggested that you choose a side port layout rather than front or rear disc rotary valve arrangements as this configuration will give the characteristics required. You will probably need to make your design a short stroke engine, but this, of course, is up to you. You should also design a test bed, on which the engine can be fitted and connected to a pre-supplied dynamometer such that test engine data can be acquired

You are required to produce a series of component drawings (one per group member), a General Arrangement (GA) drawing of your design and also provide a validation report. The arrangement drawing should include a parts list containing materials specification and any treatments required. Where possible certain items (such as nuts and bolts) should be chosen from standard parts.

The company are particularly looking for good innovative design e.g. is there an alternative way to adjust compression, can the engine benefit from a form of [illegible]charging. They are projecting a production run of initially of 5000 engines per year.

ASSIGNMENT

You are required to submit solutions for the above two-stroke engine and you must include the following:

1. A fully detailed and dimensioned components drawing selected from your design (one from each member of your group).

2. A short report validating your design, this may include [10.5 pages max]:
    a. An engine design summary [2-page max], including
        i. Basic engine parameters such as weight, stroke/ bore ratio, stroke length, bore and displacements
        ii. Details of porting arrangements
        iii. Details of any important aesthetic or ergonomic features
        iv. Engine mounting details
        v. Test frame and dynamometer connecting hub
        vi. Maintenance schedules etc.

    Think of this as the leaflet that would be included in the box!

    b. Engine and porting calculations [4 pages max], viz:
        i. Stroke/ bore ratio
        ii. Cylinder bore
        iii. Stroke length
        iv. Displacement
        v. Porting arrangements
        vi. Timing diagram
    c. Design calculation[illegible] pages max]
        i. Sha[illegible]alysis
        ii. Loads – st[illegible]ate
        iii. Stea[illegible]te [illegible]um diam[illegible] calculations
        iv. [illegible]ue [illegible]nd [illegible] of safety
        v. Be[illegible]g sp[illegible]fication
        vi. Stre[illegible]alculatio[illegible]or the test rig and hub coupling
    d. Feedback
        i. Inc[illegible]page description of how feedback was used to develop your design further

3. A general assembly drawing to BS 8888 of your design (in third angle projection) containing:
    a. At least internal and external orthographic views, showing clearly all the main components, with sections as needed for detail and relative positions.
    b. Notes for layout and installation, service supply details, testing, relevant codes etc. so that a prospective buyer (an engineer) would have sufficient information to make a decision to order.
    c. Overall/ leading dimensions and engine weight.
    d. Parts list including component details – quantities, materials and supply.
    e. Separate items identified with leader lines to balloons that include the item reference number linking to the parts list.

N.B. Part 2 of the assignment does not require a lengthy report discussing why you chose the final design; rather you should focus on the technical aspects of your design, such as the assumptions you have made, the calculations and any comments on results. There is also no limit on the number of drawings that you choose to submit.

ASSESSMENT CRITERIA

Assessment of submissions will take account of the following factors:

Engineering integrity and the quality of design decisions

- Quality and practicality of design (location etc.)
- Engine parameters
- Porting and timing
- Fuel line
- Test bed and coupling hub

Application and understanding of engineering technology

- Material selection
- Manufacturability

Engineering Communication

- Quality of drawings, including:
  - Parts list
  - Adequacy of vie[illegible]
  - Dimensio[illegible]
  - Note[illegible]
- Report pre[illegible]ntatio[illegible]

SUBMISSIONS

1. The componen[illegible] dra[illegible] [illegible]re to be [illegible]mitted through CANVAS on Friday 16th November 2018 by 23:59
2. The validation rep[illegible] [illegible] [illegible]e submitted through CANVAS on Friday 30th November 2018 by 23:59
3. The design drawings are to be submitted through CANVAS on Friday 14th December 2018 by 23:59

Also, it is worth noting presentation quality is considered in the assessment.

RESOURCES

During the course formal inputs are offered on the topics of mechanical design techniques, engine theory, manufacturing and materials and their effects on component design. Informal inputs can also be sought from the course tutors who are to be considered as consultants. Drawings should be produced using the SolidWorks software suit or similar software. Please refer to the introductory presentation for further details

RECOMMENDED TEXT

Simmons, C H. and Maguire, D E. (2007). Manual of Engineering drawing to British and International Standards, Elsevier, ISBN: 0-7506-5120-2

Budynas, R G and Nisbett, J K (2015). Shigley's Mechanical Engineering Design (Si), 10Ed, McGraw Hill, ISBN: 933922163X

Website: http://www.roymech.co.uk

ADDITIONAL INFORMATION

This information is offered to assist you in understanding what is required and should be read carefully.

Drawings should be in third angle projection and contain all parts identifying bought-out as well as in-house (designed and manufactured) parts.

There are a number of ways in which the required engine can be achieved. The 'best' solution will only emerge when you draw a few ideas. Even th[illegible] outcome will depend on what additional requirements (if any) you think are needed or would [illegible] the product. It is very important to take into account the manufacturing costs associated with materials and their respective processing technologies in order to produce the components/sub-assemblies cost-effectively in the required quantities (batch sizes).

Get used to drawing your ideas [illegible] drawing board preferably (although these are limited in Mechanical Engineering, and sketch pads can be very useful however), so corrections can be easily be rubbed out. This is the professional approach – hand sketches done in biro on A4 paper and not to scale cannot give the 'feel' necessary to determine whether the design or even a small design feature is a good idea or not.

Always draw in full size; by doing this you will better appreciate the various design problems as they emerge as well as showing the client exactly what they are getting. In addition the tutors can only be of assistant if you have something positive to show them each week.

You should aim to have your preliminary designs completed by the midpoint of the semester. You should do this to enable the course tutors to give you feedback on your design and the 'go-ahead' for the final design work and the individual material/manufacturing reports. To be at this point at week 7 will require assembly drawings that show all the necessary views such that a professional engineer can understand exactly how your design operates and also how key engine components can be manufactured.

LEARNING OUTCOMES

1. Understanding the technical requirements of a project brief, i.e.
    a. the operating cycle of a two stroke engine
    b. the piston/cylinder porting requirements
    c. Awareness of the influence of stroke/bore ratios in engine design
2. Designing components to specific requirements, i.e.
    a. Appreciating the tolerances needed in small engine design
    b. Appreciating the cooling and lubrication requirements of small two-stroke engines
    c. Understanding the fuel oil mixtures required by two stroke engines
    d. Understanding a simple carburettor
3. Designing with consideration of the technology of manufacturing processes and material choice, i.e.
    a. Appreciating the choice of materials available and using this information to make good decisions regarding materials selection
    b. Appreciating the limitations of manufacturing processes and materials in achieving satisfactory product performance
    c. Assessing the environmental impact of different phases in the engine live-cycle (energy requirements and carbon emissions/footprint)
4. Presenting a case for a chosen design neatly and persuasively, including the use of British Standards
5. Interfacing design and manufacturing stages in product development with viable process planning decisions
6. Understanding the communication needed between engineers in integrating component design.

OPPORTUNITIES FOR FEEDBACK

Formative feedback/ assessment

(definition: https://en.wikipedia.org/wiki/Formative_assessment)

1. Product Design Specification [See Dr Dearn when it is done]
2. During *Design Forums*
3. *'Flash Feedback'*

Summative feedback/ Assessment
(definition: https://en.wikipedia.org/wiki/Summative_assessment)

1. Feedback on submissions
2. Video and provided through CANVAS, 15 working days after submission.

ACADEMIC INTEGRITY

Plagiarism will not be tolerated. It is the act of a Student claiming as their own, intentionally or by omission, work which was not done by that Student. Plagiarism also includes a Student deliberately claiming to have done work submitted by the Student for assessment which was never undertaken by that Student, including self-plagiarism and the other breaches. Sanctions of a plagiarism include the Student failing the Programme of study

Department of Mechanical Engineering
The University of Birmingham

UNIVERSITY OF BIRMINGHAM ASSESSMENT AND FEEDBACK STUDENT TEMPLATE

Section One

Reflecting on the feedback that I have received on previous assessments, the following issues/topics have been identified as areas for improvement: (add 3 bullet points). NB – for first year students in the first term, this may refer to assessments in their previous institution, or verbal feedback given in a teaching session.

Section Two

In this assignment, I have attempted to act on previous feedback in the following ways (3 bullet points).

Section Three

Feedback on the following aspects of this assignment (i.e. content/style/approach) would be particularly helpful to me: (3 bullet points).

minimises the separation of flow seen in figure 7. Turbulent flow is almost eliminated at the back of the car and fewer eddies are visible in figure 7. The car had to be made longer because the changes performed to it compromised the capacity of the car, therefore the new length of the car is 4.01 m. There are some little changes done to help the redesign such as creating new fenders. However, the new fenders due help with parting the air flow in a smooth manner underneath the car. The tyres are larger than the initial design to compensate the additional weight of the car. The top width of the car was also lightly reduced, decreasing the cross-sectional area to 2.284 m^2. Finally, any edges were chamfered just to faintly help the flow of air.

A CFD simulation was performed on the redesign using the same quality mesh and boundary condition used on the initial design. The new force acting on the redesign was 341.2 N, using this value and the new value of the cross-sectional area of the redesign, the C_D for the new design is 0.685. There is 13.8% reduction of the C_D, meaning that the redesign has effective improved the initial design. In addition, the initial design had a lift of 192.4 N, where the redesign achieved a lift of [illegible] N. There is [illegible]% reduction, this provides downwards force which gives the car more traction control making it easier to handle the car. Still, there is not a significant reduction, this may be due to increasing the length which increases the weight of the car. The slight decrease to the width of the car is also a factor why the drag was only slightly improved. So, further improvement that could be done in the future is an increase to the area of the car and more aerodynamic designs to compensate the length of the car. Based on the CFD analysis, performed on the initial design and the redesign the results show that the use of aerodynamic design has reduced the model's coefficient of drag, making the redesign an excellent recommendation to replace the model T.

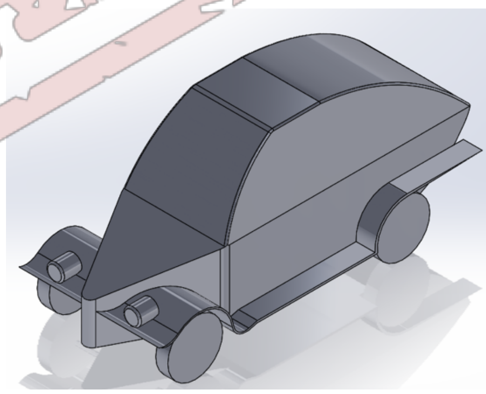
Figure 6. The redesign