UNIVERSITY OF BIRMINGHAM

Computational Fluid Dynamics

Mechanical Engineering BEng

Module co-ordinator: Dr Leung Soo & Daniel Espino

Report by: William Onyema (ID no.: 1826926)

Personal 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 action reduces the

Figure 1. Drawing of the ford model T

meshing complexity of the simulation and will also quicken the solving time of the mesh with no significant impact to the results.
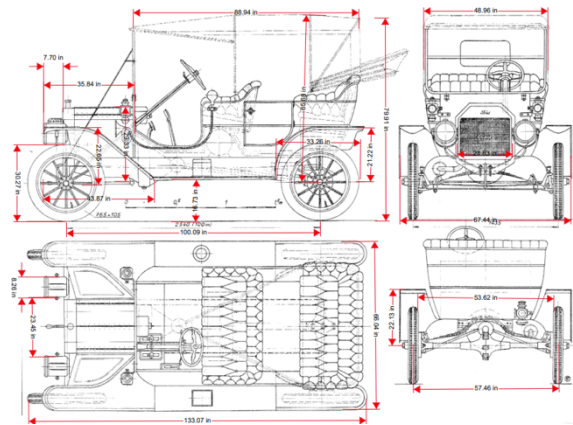
Mesh, Solver & Related Computational Parameters

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
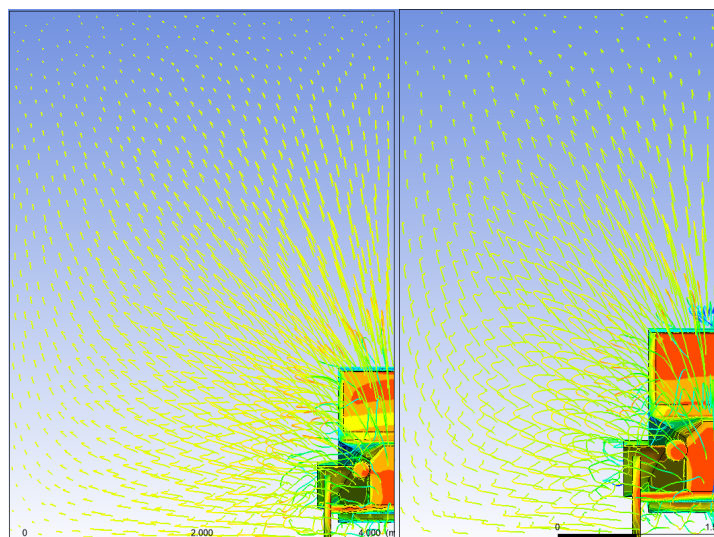

Figure 2. New computational domain (left) and Old computational domain (right)

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 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 is because it produces layers of high-quality prismatic mesh around areas where is turbulent flow excessive. This will also decrease the density from 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
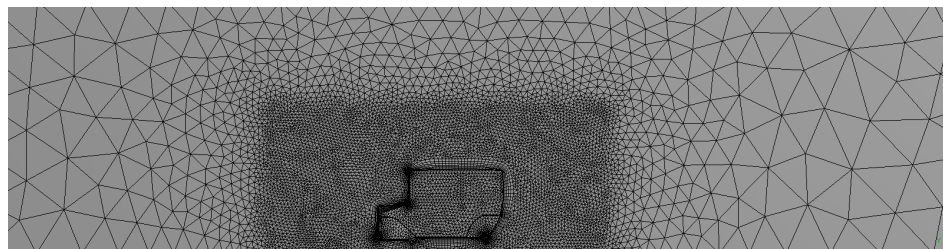

Figure 3. the final mesh created for the initial design

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.

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

Figure 4. velocity streamlines on the initial design

car which is 2.535 m^2 was found using SolidWorks, which is used to obtain the coefficient of drag C$_D$. The equation is

$$C_D = \frac{2F}{\rho A U^2} = \frac{2 \times 439.314}{1.229 \times 2.535 \times 18.8^2} = 0.8$$

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.
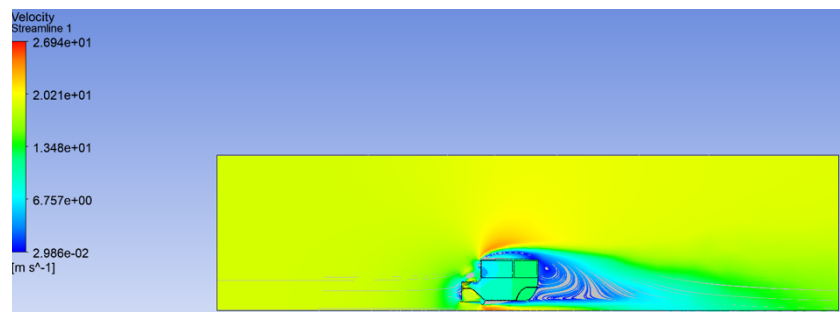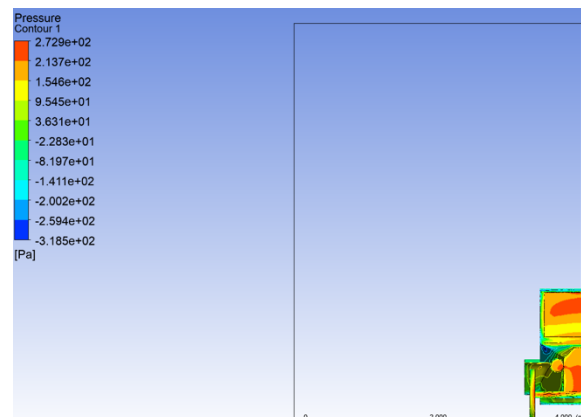

Figure 5. the pressure drag on the initial design

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 in the initial design is also drastically decreased due to the curved shaped windscreen seen in figure 8. This is because air is able to escape allowing the are to slip. The height 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 always for
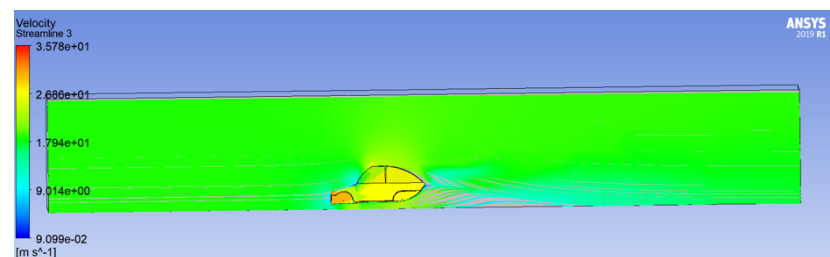

Figure 7. velocity streamlines on the redesign

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
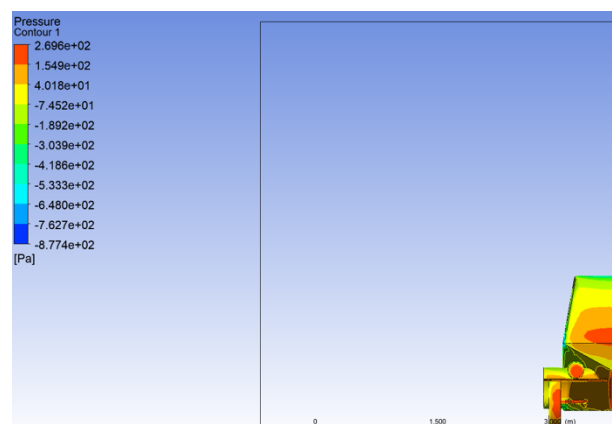

Figure 8. the Pressure drag on the initial design

bumper were slanted creating an Aerofoil shape at the back of the car. This effectively

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 175.2 N. There is an 8.94% reduction, this provides downwards force which gives the car more traction control making it easier to handle car [5]. 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
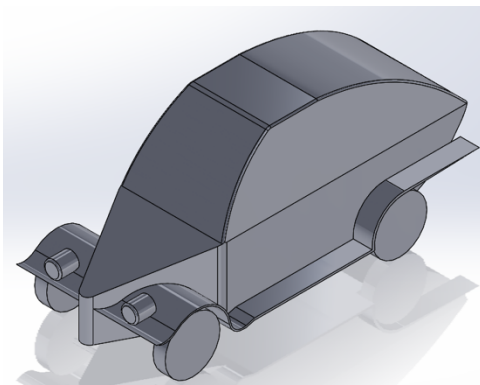
Figure 6. The redesign

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.

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