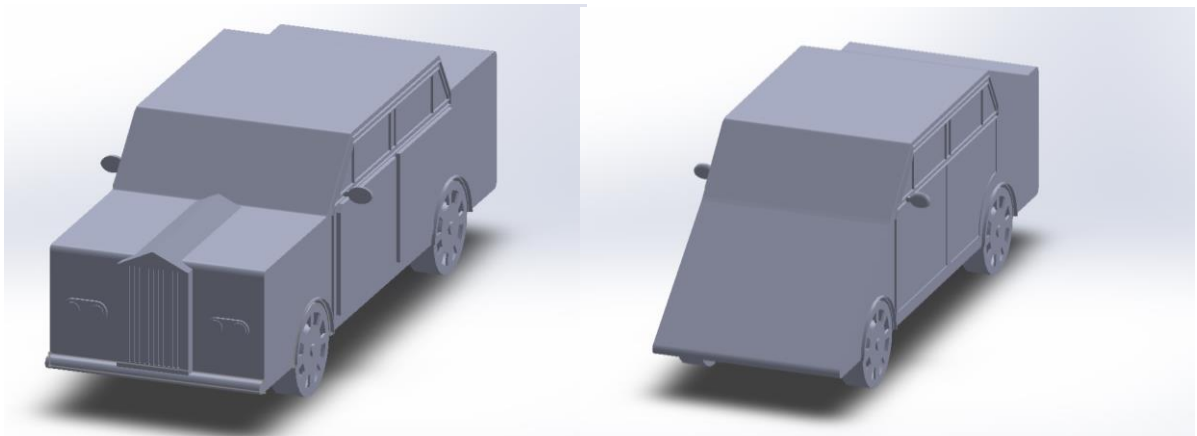




Mechanical Engineering (BEng)
Computational Fluid Dynamics
DR. Daniel Espino



Emma Howard
1896525

Introduction: In this report the classic Rolls Royce Silver shadow was redesigned to become more aerodynamic. *Figure 1* provides a sketch of the original car model with it's dimensions and *Figure 2* shows the simplified computer-aided-design (CAD) model that was made on Solidworks. Both models were assessed using Ansys 2020 R1, using the CFX solver. Certain parts of the model were simplified to aid the computational analysis. It was decided to include certain main features such as: the wheels, side-view mirrors and details of the front bumper of the car, as these would have a significant impact on the drag having been some of the first contact points with air and having a relatively large surface area. However more discreet features were ignored such as the exhaust systems, logos etc to reduce complications within the simulations. The same conditions in the simulations were applied to both models to show how the design has improved.

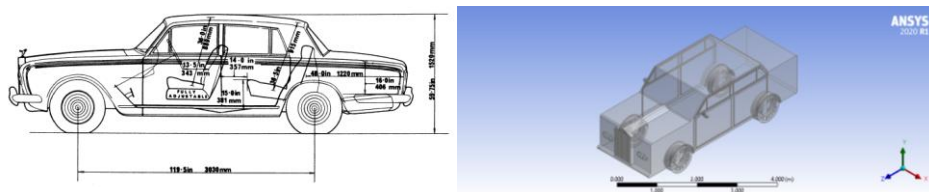


Figure 1 (LEFT): Silver shadow model sketch and dimensions (Rijkers 1998). Figure 2 (RIGHT): CAD model of the Silver shadow.

Geometry: The flow of fluid was not bounded within a domain of rigid walls, it flowed around the car, making it an external flow problem. 2 planes of symmetry were made. On the XZ axis, a symmetry had to be made to form the enclosure. This defined that the base of the wheels are in contact with the floor. Then, through the centre of the car, a plane of symmetry was created in the XY direction which slices the car in half. This reduces computational time and meant more mesh was available for the other half of the car. For the analysis, a computational domain was created to show how the air flow disperses around the vehicle. According to (Lanfrit, 2005) the dimensions of the computational domain should be at least 3 car lengths in front of the car, and 5 lengths behind therefore this was replicated. These specifications of the enclosure allow the wake of the car to eventually return to a free-stream condition and this means the exit length is long enough to make the boundary conditions valid. The boolean operation was then performed to separate the car body from the volume of fluid.

Material properties: The fluid being modelled was air at 25°C, representing the environment of an average summer day in the UK. The density of air at this temperature is 1.1644 kg/ m³ and viscosity is 1.86x10⁻⁵ kg/ms (Engineers edge, 2020). Flow around cars usually have a high Reynolds number and therefore it was assumed the flow was in a turbulent state which was modelled using the k-epsilon function. For turbulent flows the viscosity is important, therefore the flow was modelled as being viscous. After using the following equations EQ.1. and EQ.2 a Mach number of 0.25 was found. Fluids with low velocity and those that move slower than Mach 0.3 are generally considered as incompressible therefore this assumption was used in this study. EQ.1: $a = \sqrt{\gamma RT}$

EQ.2: $V = aM$ **Boundary conditions:** The frontal area was labelled as the inlet and a velocity boundary was applied here. A velocity of 15m/s (roughly 30mph) was chosen as this is an expected speed a user would be driving at in their day-day travels. The rear area was labelled as the outlet which was assigned an average static pressure of 0 (gauge). This is because the gauge pressure is only required for subsonic flows (ANSYS Inc, 2009) therefore it sets the pressure as absolute at a value of 1 atm. The floor was labelled as a non-slip wall as it was assumed that the wheels are locked and that the car is sliding since modelling rotating tyres requires accuracy in geometry and boundary conditions which can be very complex and time-consuming. The top and sides of the enclosure were also labelled as a non-slip wall as this would be the common default for all walls in viscous flows (ANSYS, Inc 2009). It was decided to use the k-epsilon turbulence model as this model is good for external flows and simulations involving low velocity therefore matching the criteria of this study. For CFX it required an input for turbulent kinetic energy and turbulent eddy dissipation which was calculated using the following equations:

$$\text{EQ.3: } K = \frac{3}{2}(UI)^2 \approx 1 \frac{m^2}{s^2} \quad \text{EQ.4: } \epsilon = C_v \frac{\frac{3K}{4} \frac{2}{l}}{l} \approx 1 \frac{m^2}{s^3}$$

A Medium-turbulence of 5% was chosen which is commonly simulated for flow in not-so-complex geometries or for low speed flows (SimScale, 2020).

Where C is the turbulence model constant, which usually takes the value 0.09, K is the turbulent energy, L is the turbulent length scale, U is the mean flow velocity, I is the turbulence intensity.

Mesh: Whilst generating the mesh for the computational domain, it had to be taken into consideration on how many cells the mesh would contain as this strongly affects the computational cost and time it would take in order to solve the fluid flow problem. When creating the mesh, default options for element sizing were chosen due to the limitations of Ansys student having a maximum mesh size of 512000 cells. The mesh was then refined based on proximity and curvature, this combination of settings captures all the geometric features of the vehicle. Then, to further improve the mesh quality, Inflation layers were added. Layered elements provide good alignment with the flow near wall boundaries (Lanfrit, 2005) and therefore this improved the reliability of the results. By using the first aspect ratio function, prismatic layers were created as seen in *Figure 3/4*.

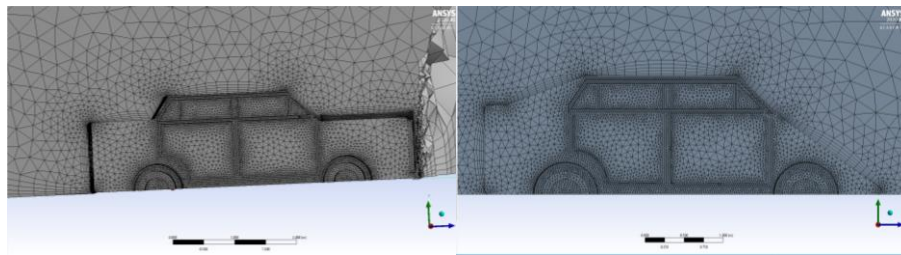


Figure 3 (LEFT): Mesh around the original car model showing the prism layer growth.

Figure 4(RIGHT): Mesh around new car model showing prism layer growth.

Solver: For the convergence criteria in CFD, levels of $1E-5$ are considered as well converged (Kuron,2020) therefore this value was chosen. It was chosen to use a steady-state simulation as it computes the fully developed solution that does not change in time whereas a transient simulation would compute the instantaneous values in each time for each quantity (CFD Support, 2020). Despite the transient model being more comprehensive it has much higher requirements compared to steady-state i.e. CPU time, disk storage etc. Therefore, as the steady-state simulation is faster, it is more desirable for this project.

Results and analysis: To improve the aerodynamics, the classical large and boxy frontal area of the car was significantly reduced by forming a curved edge up to the windscreen. The boot was then extruded up to the roof to direct the flow of air past the car better. By reducing the surface area that is perpendicular to the flow, it saw significant reduction in pressure. Just by reviewing the maximum

values of pressure acting on the car from *figure 5/6* (1.753×10^2 and 1.135×10^2 pa respectively) it showed a reduction of pressure by 35%, highlighting how the flow of air is being significantly less disrupted. Another improvement that was made was on the side-view mirrors, originally, they were typically very boxy and almost rectangular meaning air didn't flow around it well. Therefore, they were redesigned as a spherical shape which also saw a significant drop in pressure which can be seen by the reduced brightness in the contours.

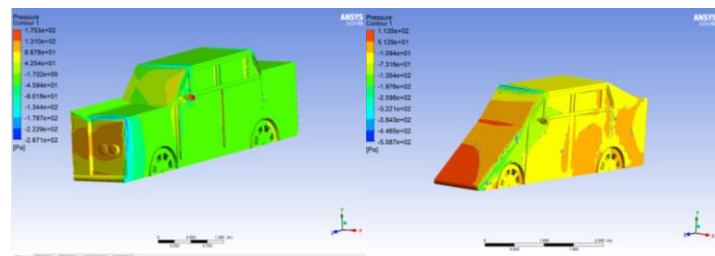


Figure 5(LEFT): Pressure contour of original Rolls Royce. Figure 6(RIGHT): Pressure contour of new design.

Another factor that shows improvements of the aerodynamics is the velocity streamline, in *Figure 7/8*. The distortion of velocity is represented by colour change. It can be seen that in the original design in *Figure 7* the streamlines near the car go from orange to a green/blue showing a great change in velocity whereas in *Figure 8* only little disruption can be seen as the flow near the car shows little colour change as it turns slightly from green to blue. This suggests that the new design is smoother and allows air to pass over the car better.

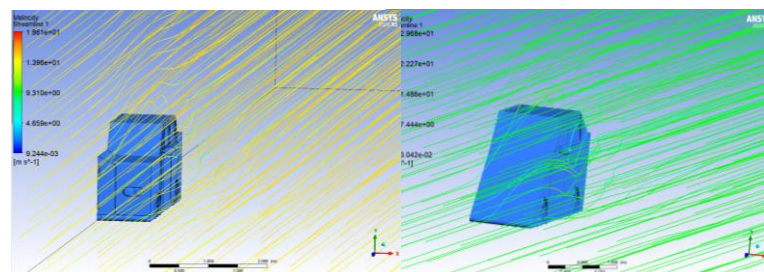


Figure 7(LEFT): Velocity streamline around original design. Figure 8(RIGHT): Velocity streamline around new design.

Finally using the following equation **EQ.5**, the drag coefficients were found.

$$\text{EQ.5: } C_D = \frac{2F_d}{\rho v^2 A}$$

CD=Coefficient of Drag. Fd= Drag Force ,Newtons. ρ=Density of fluid,1.1644 kg/m3. v=velocity, 15m/s. A=Cross sectional area, 3 m2

The original model achieved a CD of 0.3 whereas the new, improved model computed a value of 0.28. These values verified that the new model is more aerodynamic. By having a new lower CD and better performance characteristics ultimately, this would lead to improvements in the performance of the vehicle in terms of pertaining it's speed and fuel efficiency.

Validation: To ensure the simulation that was created was valid, any sources of error in computational simulations had to be considered and controlled. Discretisation errors occur due to difference in results for real-life and computational simulations due to limited time and resolution that a computer model has. One way this was controlled was by refining the mesh as much as possible, this was done by reaching the limits of ANSYS student mesh in order to achieve the most reliable result as possible and by adding extra refinements to make the near-wall boundary conditions more valid. Round-off errors depend on computational power and how many figures the computer can store. To try and reduce these errors double precision was used in the solver. Iteration or convergence error can lead to an incomplete result, to prevent these errors the model was left to complete to the final converged state. Also, the convergence criteria that was chosen was relatively low improving the numerical accuracy of the solution. Physical modelling errors occur due to turbulence models uncertainties however, it can be trusted that physical processes on CFD are known to high accuracy and extensive research was conducted on the turbulence model that was chosen to ensure it well-suited the scenario of the simulation. Finally, human errors can occur and the model created may not simulate real-life conditions. To prevent this, thorough research was conducted into finding the most appropriate input conditions. Finally, to show the accuracy of results in this report, the drag coefficient (CD) of the CAD Silver Shadow was found and compared to the real value. Although research into the CD of the Silver Shadow remains unclear, (Pakwheels, 2017) plotted a chart of the average CD of cars against year, and for the 80's the average is 0.33 and in the simulation a value of 0.3 was found. This suggests that the CAD model is a reasonable representation of the real model and therefore the inputs and results of this simulation can be trusted to be accurate and close to real-life results.

References:

- ANSYS FLUENT 12.0 User's guide. 2009. *7.3.8 Pressure Outlet Boundary Conditions*. [online] Available at: <<https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node244.htm>> [Accessed 17 November 2020].
- ANSYS FLUENT 12.0 User's guide. 2009. *7.3.14 Wall Boundary Conditions*. [online] Available at: <<https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node250.htm>> [Accessed 17 November 2020].
- Cfdsupport.com. *Transient Or Steady-State?*. [online] Available at: <<https://www.cfdsupport.com/OpenFOAM-Training-by-CFD-Support/node356.html>> [Accessed 17 November 2020].
- Engineersedge.com. 2020. *Viscosity Of Air, Dynamic And Kinematic | Engineers Edge | Www.Engineersedge.Com*. [online] Available at: <https://www.engineersedge.com/physics/viscosity_of_air_dynamic_and_kinematic_14483.htm> [Accessed 19 November 2020].
- Kuron, M., Engineering.com. 2020. *3 Criteria For Assessing CFD Convergence*. [online] Available at: <<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/9296/3-Criteria-for-Assessing-CFD-Convergence.aspx>> [Accessed 17 November 2020].
- Lanfrit, M., 2005. *Automotive External Aerodynamics With FLUENT*. [ebook] Available at: <https://southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf> [Accessed 17 November 2020].
- PakWheels Blog. 2017. *History Of The Drag Coefficients In Cars*. [online] Available at: <<https://www.pakwheels.com/blog/history-drag-coefficients-cars/>> [Accessed 20 November 2020].
- Rijkers, M., 1998. Rrsilvershadow.com. *Rolls-Royce Silver Shadow - Sizes*. [online] Available at: <<https://www.rrsilvershadow.com/EMaten/Maten.htm>> [Accessed 17 November 2020].
- SimScale. *K-Epsilon | Global Settings | Simscale Documentation*. [online] Available at: <<https://www.simscale.com/docs/simulation-setup/global-settings/k-epsilon/>> [Accessed 17 November 2020].
- Symsscape.com. 2009. *Reynolds-Averaged Navier-Stokes Equations | Symsscape*. [online] Available at: <<https://www.symsscape.com/reynolds-averaged-navier-stokes-equations>> [Accessed 17 November 2020].
- *Vehicle Aerodynamics*. [ebook] G. Dimitriadis. Available at: <<http://www.ltas-aea.ulg.ac.be/cms/uploads/VehicleAerodynamics02.pdf>> [Accessed 17 November 2020].