SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

One-Way Fluid Structure Interaction of a Utility Vehicle Splitter using CFD Modelling

Connor Wright[1], Mohammad Al-Rawi[1,2,*]

[1] Centre for Engineering and Industrial Design (CEID), Waikato institute of Technology-Te Pūkenga, Hamilton, New Zealand
[2] Chemical and Materials Engineering, The University of Auckland, Auckland, New Zealand

ARTICLE INFO

Article history:
Received 6 May 2023
Received in revised form 8 June 2023
Accepted 7 July 2023
Available online 1 December 2023

Keywords:
Finite Element Analysis; One-Way Fluid Structure Interaction; Ute Aerodynamics

ABSTRACT

The Utility Vehicle, commonly known as a “ute” in New Zealand, and a “pickup” in other countries, is very popular for work, recreation and motorsport. This report investigates how the addition of a front splitter influences the handling ability on a Ford Courier Ute, for better aerodynamics during racing. Computational fluid dynamics (CFD) methods and Finite Element Analysis (FEA) were used to investigate the one-way fluid structure interaction of the speed of the Ute with and without the splitter modification. Three configurations of the splitter were modelled and analysed using SolidWorks 2022, CFD and FEA packages. The ground clearance between the splitter and the road was set at three heights, 200 mm, 150 mm and 50 mm. These designs were compared with the control, in terms of the amount of downforce, drag and aerodynamic efficiency. The ground clearance of 150 mm was optimal, in terms of the parameters investigated. The findings from this paper may inspire Ute enthusiasts to fit a splitter to their Ute for racing applications.

1. Introduction

Computational modelling approaches are applied to determine aerodynamic efficiency of vehicles’ panelling with the aim of reducing fuel consumption [1]. Computational fluid dynamics (CFD) models can help to optimize the aerodynamic shape of vehicles to reduce the drag coefficient [1-3]. Hachimy *et al.,* [2] presented a computational model focusing on three turbulent regimes to predict the drag coefficient for a vehicle body using ANSYS Fluent analysis. They investigated the k-epsilon, the shear stress transfer (SST) and the k-omega SST to minimise the drag coefficient. The study found that the total drag coefficient value in the SST agrees with the experimental data with a difference of 0.82%. Alshqirate *et al.,* [3] analysed the flow over a small truck (a “pickup”) with and without a covered cargo area as an extra accessory using OpenFOAM. Their turbulence model was based on k-omega SST and showed that covering the truck results in a reduced drag coefficient due to reduction of recirculation zones compared to the open cargo truck. Yang and Khalighi [4] assessed the assumption of a steady state flow around a small truck to validate the aerodynamic shape, which

* *Corresponding author.*
E-mail address: mohammad.al-rawi@wintec.ac.nz (Mohammad AL-Rawi)

https://doi.org/10.37934/cfdl.15.12.3443

is also compared experimentally. The CFD steady state results provided the same qualitative outcomes to the experimental scenario, assuring that this simulation method provides the same quality data as the transient analysis for this type of problem.

Car enthusiasts frequently modify their own vehicles with accessories intended to enhance the aerodynamics and give a similar outcome as racing cars. Therefore, there is a significant demand for improvements to the different types of accessories, such as front and rear spoilers, to achieve an acceptable drag coefficient [5]. Previous research has investigated the optimally aerodynamic shapes for racing vehicles, from those that are not used on public roads, such as Go-Karts [5], to those that are roadworthy and high-performance, such as Mercedes-AMG GT [6], and Porsche 911 [7]. Computational simulation has been used in small vehicles, to enhance the design of the chassis in a Go-Kart [8] and in large vehicles, such as modelling the aerodynamic response for a large bus when overtaking, taking into account the interference of crosswind [9]. CFD analysis can also be presented as one-way fluid structure interaction (FSI), as shown in [5], for spoiler optimization or to model a sedan car operating at different cruise speeds [10]. Also, the one-way FSI model was used to investigate the aerodynamic shape for the Shell Eco-marathon (SEM) vehicle [11].

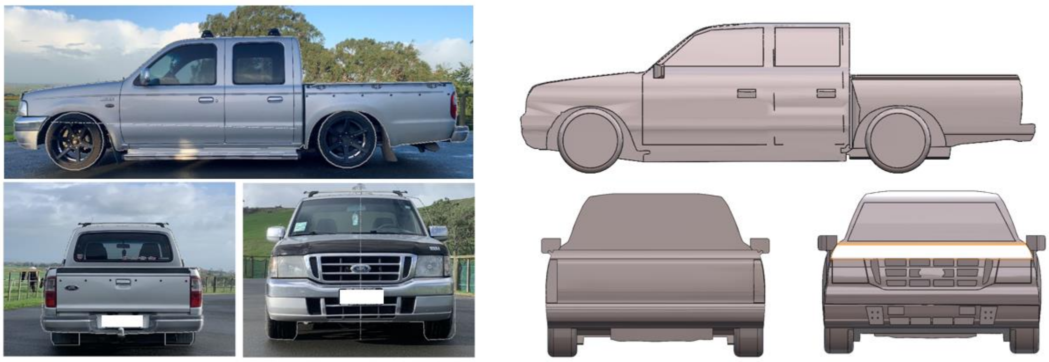
Racing teams across the world utilise vehicle aerodynamics to maximise negative lift (downforce) while minimising drag. Downforce is a force that increases and decreases with the properties of air flowing over the vehicle. Downforce increases available friction at the tyres. Friction is a product the normal force applied to the road, where normal force comprises the downforce and the force due to the mass of a vehicle [12-15]. Downforce is preferred as inertia is a product of mass. The higher the inertia the more the vehicle will oppose the change of direction, which decreases the vehicle's handling ability [15]. Most production vehicles come from the factory without the ability to develop downforce. This is due to their shape acting like a plane wing and providing positive lift at high speeds. Downforce is developed with the addition of components such as Rear Wings, Diffusers and Front Splitters.

This report investigates the effects of frontal aerodynamics on Utility Vehicles (Utes) used in the racing scene. Utes are very popular in New Zealand. According to vehicle monitoring website Car jam, the Ute, and specifically the Ford Ranger make and model, is the most popular vehicle in New Zealand [16]. Utes are very versatile, due to their seating capacity suitable for families, their towing capacity, suitable for boats and most caravans, as well as the carrying capacity of the tray. However, Utes are not known for their aerodynamic ability. Their larger size and optimisation for pulling power means they are not highly streamlined vehicles. Additionally, the tray acts similarly to a parachute.

This study investigates a front splitter modification to a Ute. A front splitter is a component added to the front of the vehicle, designed to split the oncoming air, keeping the high-pressure, low velocity air above the vehicle and low-pressure, high velocity air below the vehicle. The aspects of a front splitter that contribute to its ability to achieve this outcome include its length, width, angle of attack and ground clearance. This study focuses on the effect of adjusting the ground clearance, setting this at 200 mm, 150 mm and 50 mm, whilst holding the other aspects constant.

2. Methodology

The study investigates the most suitable ground clearance, to achieve the highest amount of downforce while also looking at the how each configuration affects drag across the Ute. The Ute analysed in this study has 2-Wheel Drive (2WD) 2004 Ford Courier as shown Figure 1 (a). These Utes are smaller in size compared to today's standards and perform well on a racetrack. They can reach speeds of 150 km/h, therefore the study was conducted with air speeds matching this speed. This study looked at pressures and velocities to find drag and lift coefficients using CFD methods, and a stress analysis was evaluated using Finite Element Analysis (FEA). The images of the Ute shown in Figure 1 (a) were converted to the STL (stereolithography) file using SolidWorks 2022 based on the realistic dimensions of the vehicle to achieve the Ute model as shown in Figure 1 (b). The dimensions of the Ute were obtained physically from the Ute and with the right ratio are implemented in SolidWorks. The combination of photographs, as below, and physically-measured dimensions were entered into SolidWorks 2022. We then use the sketch command to convert the 2D images to a 3D model.



(a) (b)

Fig. 1. (a) A 2004 Ford Courier Ute (b) The 3D model of the 2004 Ford Courier Ute

2.1 Computational Modelling Analysis

The computational domain was set as an external analysis type, excluding cavities and without flow conditions in SolidWorks Flow Simulation, as shown in Figure 2. The gravity was assumed in the negative direction of y -axis, with the flow type set as laminar to turbulent, with velocity in the negative direction of the Z-axis at a speed of 150 km/h.

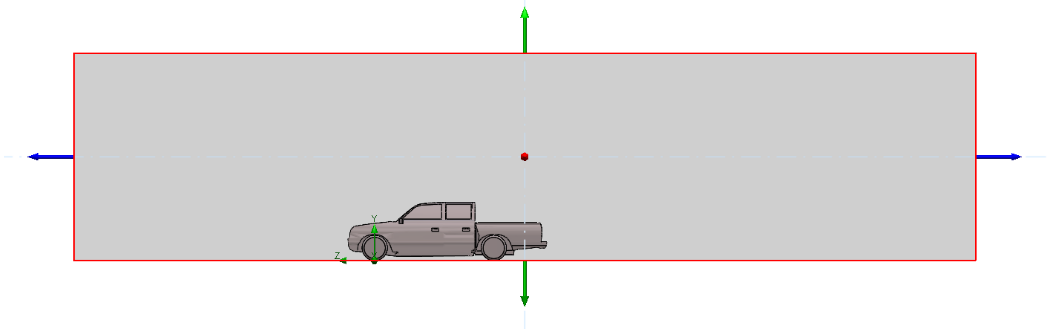


Fig. 2. The computational domain for the Ute showing the direction of the velocity

Flow Simulation was used to simulate real world events, calculating pressures, velocities, and forces when fluids are introduced to a part. The model for the replica Ford Courier is shown Figure 1 (b). The Splitter with components and dimensions (in mm) is shown Figure 3.

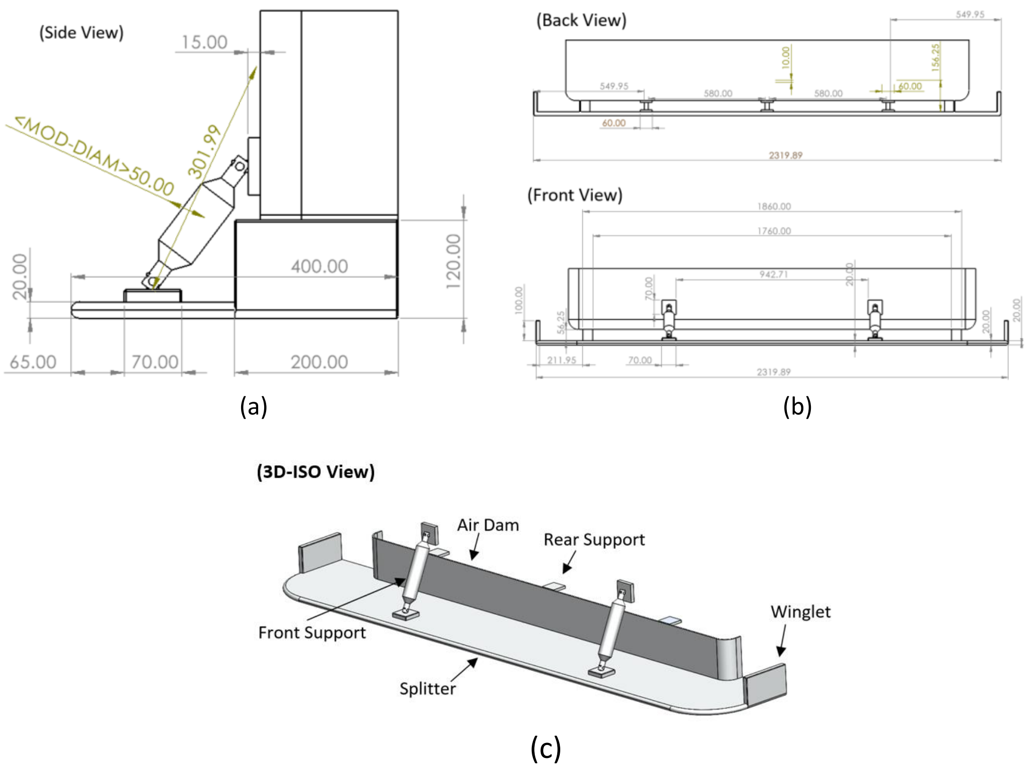


Fig. 3. The drawings for the Splitter showing (a) the side view, (b) front view, back view, and (c) the 3D ISO-view (dimensions in mm)

The height of the air dam and rear supports was changed from 6.25 (High Splitter) to 56.25 (Medium Splitter) and 156.25 (Low Splitter) as shown in Figure 4. This splitter was designed to bolt onto the Ute with minimal customisation, using existing chassis rails. A MATLAB 2022a code was developed to ensure the front supports were set at the correct length and angle.

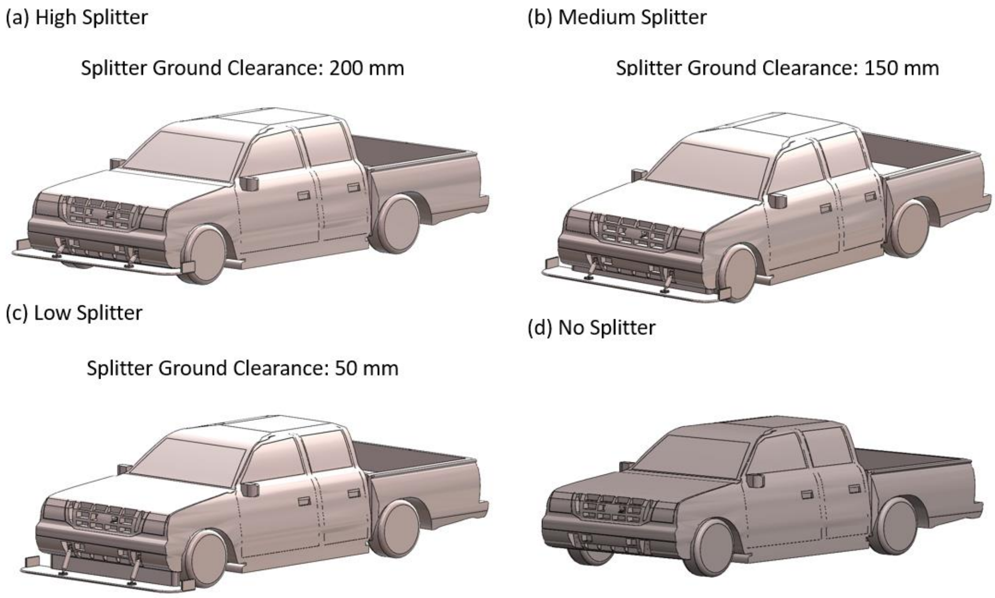


Fig. 4. The height of the air dam and rear supports set at (a) 6.25 (High Splitter), (b) 56.25 (Medium Splitter), (c) 156.25 (Low Splitter) and (d) no Splitter

This study used an external flow computational simulation. The fluid, Air, is set at -150 km/h in the z-direction. Gravity is considered at -9.81 m/s^2. The computational domain was set at, x = 3.0, y= 5.18, z = 25m. A road was placed at a tangent to the bottom of the tyres to assist in finding an accurate result. Goals were set to find the results required. They are: Global average velocities and pressures, surface goals were placed on the front, rear, top and bottom of the Ute to measure the normal force applied to those regions. These forces were used to find the drag coefficient (Cd) using Eq. (1), lift coefficient (Cl) using Eq. (2) and aerodynamic efficiency (η) using Eq. (3). Negative (Cl) values represent negative lift (downforce), resulting in negative (η) values, as they are a product of (Cl).

$$Cd = \frac{2(F_{Front}+F_{Back})}{\rho A v^2} \tag{1}$$

$$Cl = \frac{2(F_{Top}+F_{Bottom})}{\rho A v^2} \tag{2}$$

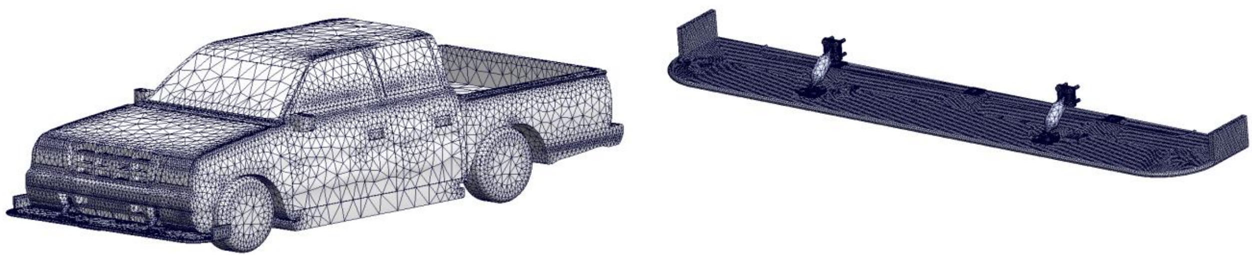
$$\eta = \frac{C_l}{C_d} \tag{3}$$

Where, force (F) is in N, fluid density (ρ) kg/m^3, average velocity (v) is in m/s and the reference area (A) is in m^2. A global mesh is used, with a minimal gap size of 1.60 m, the ratio factor was set to 1 and initial mesh is set to 5. The CFD analysis was used to find the force applied to a model and the results were exported for use in the FEA study.

2.2 Finite Element Analysis

Finite Element Analysis (FEA) is also a tool available in SolidWorks. FEA was used to determine the stress and strain within the splitter due to the results calculated in the CFD analysis. The air flow was imported using the flow effects option found under the force selection tab.

The splitter was made from 1060 Aluminium Alloy (Elastic modulus 69 GPa, Poisson's Ration 0.33 and mass density 2700 kg/m^3) which was used for its light weight and rigidity. Aerodynamic components added to a vehicle need to be light weight and rigid. The splitter must be rigid: if it moves, it becomes unpredictable, which may result in a crash. The splitter was fixed at the three supports behind the air dam and by the two brackets located on the front supports. A fine standard type of mesh was used with a maximum element size value of 60 mm and a minimum of 3 mm with high quality mesh using 16 Jacobian points for the splitter and the Ute, as shown in Figure 5. The mesh quality presented in this study was based on the recommendation in the literature [3] through attempting three different element sizes: 100 mm to generate (839,231), 80 mm to generate (1,061,326) and 60 mm to generate total element (1,441,085) using blended curvature-based mesh which leaded to the mesh convergence.



(a) (b)

Fig. 5. Mesh generation (a) for the Ute and (b) for the splitter

3. Results and Discussion

3.1 Pressure and Velocity Distribution

The results for pressure and velocity are shown in Figure 6. The cut plots show how the air starts to pool above the splitter creating a high - pressure zone above and a low-pressure zone below. This is something not seen on the Ute without a splitter. The splitter affects the wake generated, the lower the splitter is to the ground, the narrower the trailing wake is. It is also seen that the no splitter model has the wake sticking to the ground and creates a larger pool around the bottom rear of the Ute. Figure 7 also demonstrates how each splitter slows and redirects the air flow around the front and along the Ute based on the flow trajectory profiles. Figure 8 demonstrates how the low-pressure forms around the bottom of the splitter, low splitter design had the greatest area of pressure drop. Although, the high splitter had both the highest and lowest pressures across the model. With a high pressure of 105.186 kPa and a low pressure of 100.075 kPa.

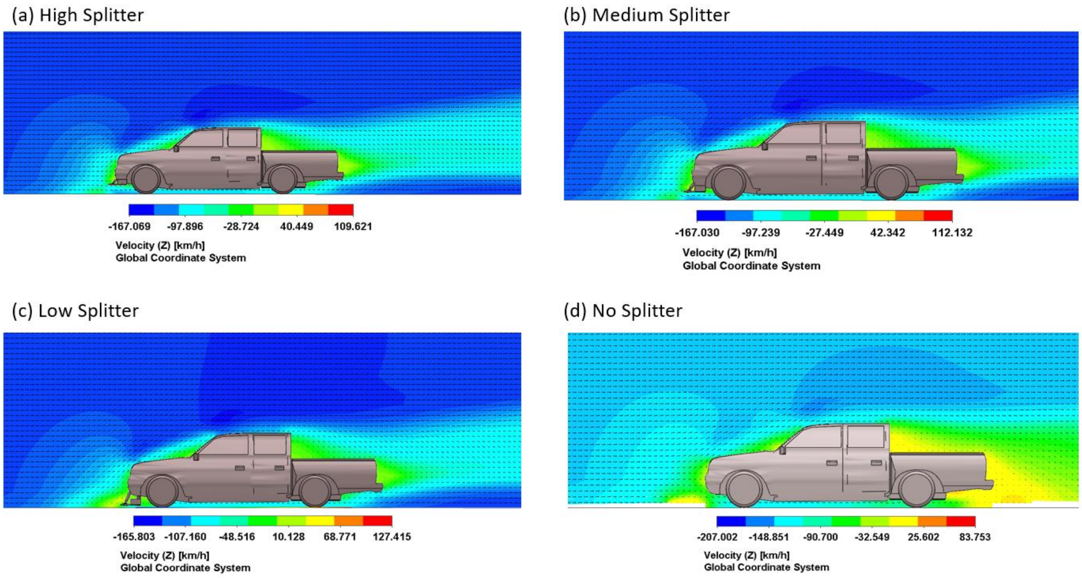


Fig. 6. The velocity cut section in the Z-axes showing (a) the high splitter, (b) medium splitter, (c) low splitter and (d) no splitter

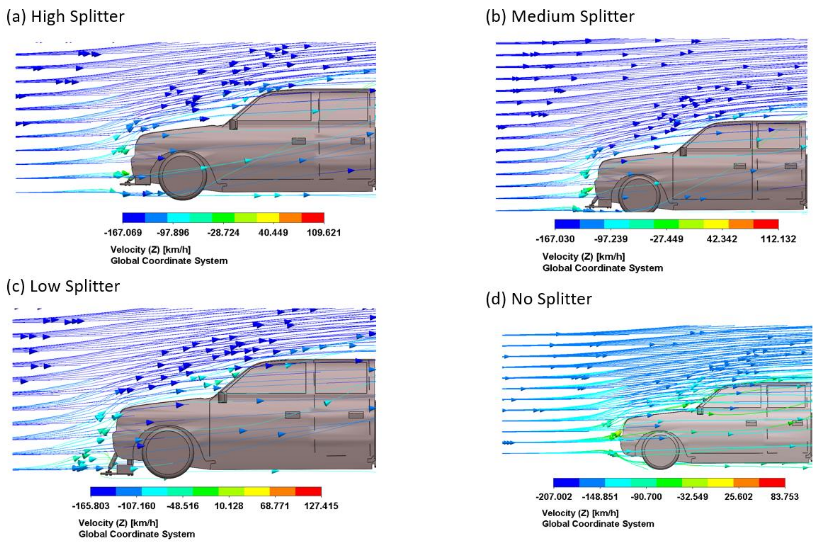


Fig. 7. The velocity flow trajectory showing the (a) high splitter, (b) medium splitter, (c) low splitter and (d) no splitter

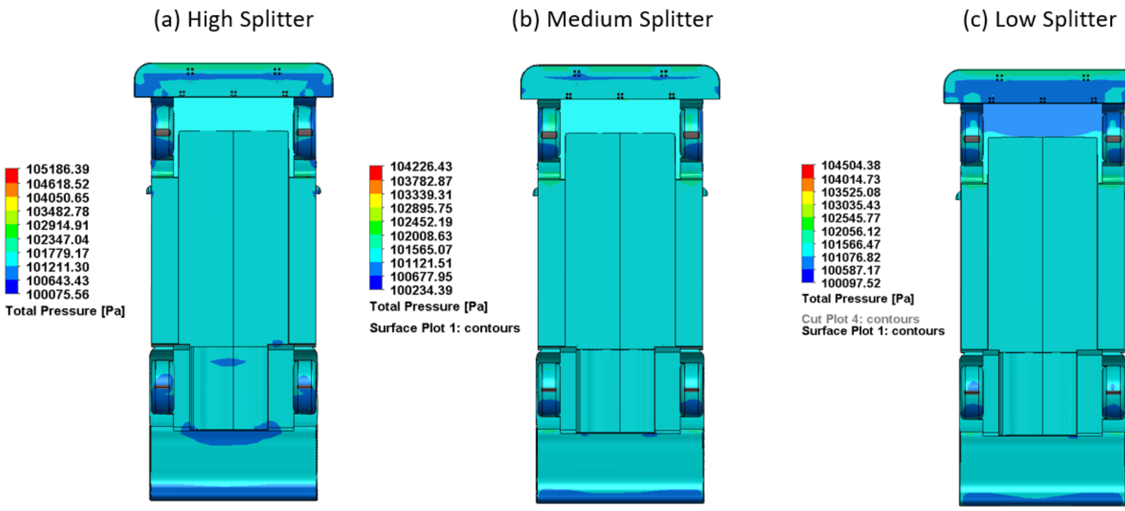


Fig. 8. The pressure contours under the Ute showing the (a) high splitter, (b) medium splitter and (c) low splitter

The drag coefficient (Cd), lift coefficient (Cl) and aerodynamic efficiency (η) was investigated. The results were exported into the FEA analysis, where they were set as an external force impacting the splitter from the negative z direction.

Figure 9 illustrates the drag coefficient (Cd), lift coefficient (Cl) and aerodynamic efficiency (η) across all four designs. The graph demonstrates that all the values for Cl were positive, this shows that none of the designs develop negative lift (downforce) at 150 km/h.

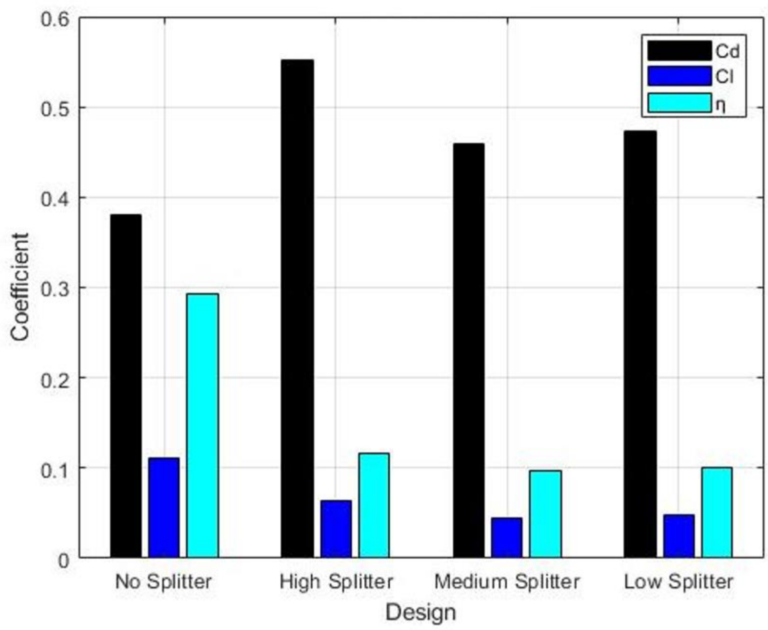


Fig. 9. Coefficient results for the drag coefficient (Cd), lift coefficient (Cl) and aerodynamic efficiency (η) for the four designs

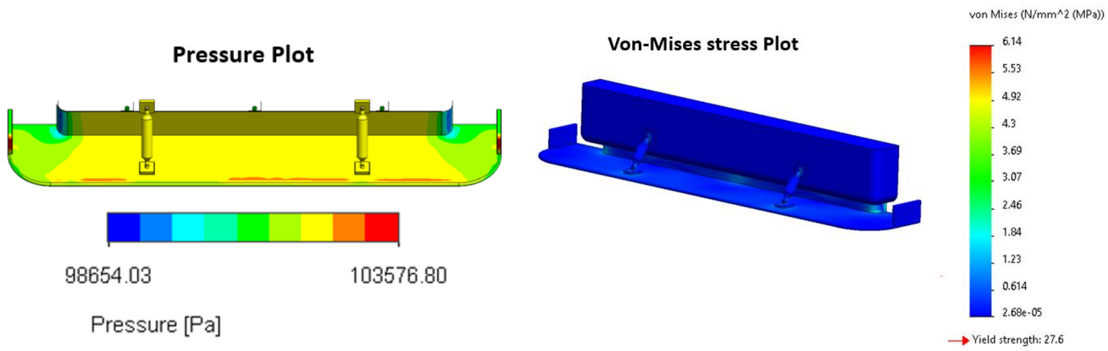
The Medium design (as shown in Figure 4 (b)) with 150 mm of ground clearance had a drag coefficient of 0.459 and a lift coefficient value of 0.044. This design opposed the greatest amount of lift, resulting in the best aerodynamic efficiency, with a value of 0.096.

The design with the worst aerodynamic efficiency was the Ute without a splitter (as shown in Figure 4 (d)) which had a value of 0.294. This design had a drag coefficient of 0.381 and a lift coefficient of 0.112. This demonstrates that the splitter increased drag force, whilst opposing lift.

In the middle of the data set was the high splitter design, with 200 mm of ground clearance and the low splitter design with 50 mm of ground clearance. The high splitter design had an aerodynamic efficiency of 0.116 compared to 0.101 for the low splitter design. The high splitter design had the greatest coefficient of drag, 0.553, compared to 0.473 for the low splitter design. The coefficient of lift was in the middle of the data set for both designs, that being 0.064 for the high splitter design and 0.048 for the low splitter design.

3.2 One-Way Fluid Structure Interaction

The one-way FSI model assessed based on the von Mises stresses and the yield point of the 1060 Aluminium Alloy material was used for the medium splitter design. The model was fixed at the front bumper to form the most accurate result. The 1060 Aluminium Alloy had a yield stress of 27.57 MPa, as shown in Figure 10. The von Mises results demonstrated that yield was not exceeded throughout the model. High stress areas are shown in Figure 10 (b). The highest stress can be seen on the upper front support reaching a maximum of 6.1 MPa. The lower front support reached a stress level of 5.8 MPa. The maximum deflection was shown to be 0.0571 mm located on the outer edges of the splitter. This is ideal to ensure rigidity of the splitter so as to reduce the likelihood of a crash. The maximum force applied to the splitter was 558.9 N, which equated to 216 N across both the front supports. The maximum force on the air dam was 63.5 N.



(a) (b)

Fig. 10. (a) the Pressure contours and (b) the von-Mises stress the medium splitter design

3.3 Results Validation and Limitation

The computational modelling for this study assisted in choosing the right splitter with respect to the ground clearance. However, this type of analysis requires a verification of the computational model to validate the drag coefficient compared to previous studies. Experimental studies [17,18] using industrial wind tunnel testing for the range for 100 km/h, found the drag coefficient to be between 0.41 and 0.51. For our study, the drag coefficient is 0.459, which is within the acceptable range. In addition, the factor of safety (FOS) for the splitter was equal to 4.5, based on automatic criterion as maximum. This value is within the acceptable range for ensuring safety of the design. One of the limitations of the design presented in this study was that, whilst the splitter did improve the handling and stability of the Ute during the high speed, it did not increase the top speed, as its main function was to create a downforce compared to the spoiler.

4. Conclusions

This study investigated four Ute designs applied to a Ford Courier make and model. Three of the designs involved installing a splitter under different configurations, each with different ground clearances, whilst the fourth design was a Ford Courier without a splitter, which was used as a control to compare each splitter design. The design with the most promising results was the medium height, with a ground clearance of 150 mm, and an aerodynamic efficiency of 0.096. This design contains a maximum of 6.1 MPa of stress with a maximum deflection of 0.0571 mm. This was optimal, as the design is well under the yield of 27.57 MPa and avoids deflection, which would cause the Ute to handle unpredictably. It was found that none of the four designs develop downforce at 150 km/h, although the designs with a splitter do oppose a significant amount of the lift developed at this speed. It is rare to see the application of one aerodynamic component in motor sport, usually several components are used in conjunction with a splitter to maximise downforce. These extra components may assist with developing downforce on the Ford Courier Ute. Reducing lift is desired to ensure the vehicle under assessment shows improved handling of corners in a racing setting. Future work will investigate the addition of side modifications to the Ute, such as adding a square back on the performance parameters addressed in this study.

Acknowledgement
The authors would like to acknowledge the financial contribution made by the Centre for Engineering and Industrial Design, Waikato Institute of Technology towards the research in this paper. This research was not funded by any grant.

References
[1] Mariaprakasam, Rafiqi Daniel Ramadhan, Shabudin Mat, Pakharuddin Mohd Samin, Norazila Othman, Mastura Ab Wahid, and Mazuriah Said. "Review on Flow Controls for Vehicles Aerodynamic Drag Reduction." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 101, no. 1 (2023): 11-36. https://doi.org/10.37934/arfmts.101.1.1136.
[2] Hachimy, Fatima-zahra, Ashraf A. Omar, and Omar Elsayed. "The accuracy of the numerical solution in predicting Ahmed body components drag coefficients." *CFD Letters* 14, no. 5 (2022): 24-32. https://doi.org/10.37934/cfdl.14.5.2432.
[3] Alshqirate, Abed Alrzaq, Dastan Zrar Ghafoor, and Sachin L. Borse. "CFD Analysis of Flow Over Pickup Truck with And Without Covering Cargo Area Using OpenFOAM." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 90, no. 1 (2022): 40-51. https://doi.org/10.37934/arfmts.90.1.4051
[4] Yang, Zhigang, and Bahram Khalighi. "CFD simulations for flow over pickup trucks." *SAE paper* 2005-01 (2005): 0547. https://doi.org/10.4271/2005-01-0547
[5] Al-Rawi, Mohammad, and Abderrahmane Oumssount. "One-Way Fluid Structure Interaction of a Go-Kart Spoiler Using CFD Analysis." In *Proceedings*, vol. 49, no. 1, p. 51. MDPI, 2020. https://doi.org/10.3390/proceedings2020049051
[6] Estrada, Gustavo. "Mercedes-AMG GTR: Aerodynamics for the Record." In *Progress in Vehicle Aerodynamics and Thermal Management: 11th FKFS Conference, Stuttgart, September 26-27, 2017 11*, pp. 135-144. Springer International Publishing, 2018. https://doi.org/10.1007/978-3-319-67822-1_8
[7] Gönüldinc, Okan, and Steffen Hölzel. "Adaptive Aerodynamik–Innovation des Porsche 911 Turbo." In *Karosseriebautage Hamburg: 13. ATZ-Fachtagung*, pp. 159-174. Springer Fachmedien Wiesbaden, 2014. https://doi.org/10.1007/978-3-658-05980-4_18
[8] Raghuvanshi, Akash Chaudhary, and Tushar Srivastav. "Design and development of foldable kart chassis." *Materials Today: Proceedings* 2, no. 4-5 (2015): 1707-1713. https://doi.org/10.1016/j.matpr.2015.07.004
[9] Yudianto, Aan, Herminarto Sofyan, and Naufal Annas Fauzi. "Aerodynamic Characteristics of Overtaking Bus under Crosswind: CFD Investigation." *CFD Letters* 14, no. 8 (2022): 20-32. https://doi.org/10.37934/cfdl.14.8.2032
[10] Zakher, Bassem Nashaat, Mostafa El-Hadary, Mohammed Abd Elfatah Elgohary, and Ibrahim M. El Fahham. "A Comparison Between Experimental Life Road Simulation and Computational Fluid Dynamics and Fluid Structure Interaction for Sedan Car." *CFD Letters* 14, no. 2 (2022): 81-97. https://doi.org/10.37934/cfdl.14.2.8197

[11] Tiew, Hedy Soon Keey, Ming Wei Lee, Wei Shyang Chang, Mohammad Hafifi Hafiz Ishak, and Farzad Ismail. "Fluid-structure interaction on the design of fully assembled Shell Eco-Marathon (SEM) Prototype Car." *CFD Letters* 12, no. 12 (2020): 115-136. https://doi.org/10.37934/cfdl.12.12.115136
[12] Gilliéron, Patrick, and Azeddine Kourta. "Aerodynamic drag reduction by vertical splitter plates." Experiments in fluids 48, no. 1 (2010): 1-16. https://doi.org/10.1007/s00348-009-0705-7
[13] Szudarek, Maciej, Konrad Kamieniecki, Sylwester Tudruj, and Janusz Piechna. "Towards Balanced Aerodynamic Axle Loading of a Car with Covered Wheels—Inflatable Splitter." *Energies* 15, no. 15 (2022): 5543. https://doi.org/10.3390/en15155543
[14] Bhattacharjee, Sourajit, B. B. Arora, and Vishesh Kashyap. Optimization of race car front splitter placement using CFD. No. 2019-01-5097. SAE Technical Paper, 2019. https://doi.org/ 10.4271/2019-01-5097
[15] Formula 1 Dictionary. (n.d.). Downforce. https://www.formula1-dictionary.net/downforce.html [Accessed 6 May 2023]
[16] Car Jam. (2022). *Popular New Zealand Vehicles*. https://www.carjam.co.nz/nz-fleet/ [Accessed 6 May 2023].
[17] Chowdhury, Harun, Raymond Juwono, Mohamed Zaid, Rezaul Islam, Bavin Loganathan, and Firoz Alam. "An experimental study on of the effect of various deflectors used for light trucks in Indian subcontinent." *Energy Procedia* 160 (2019): 34-39. https://doi.org/10.1016/j.egypro.2019.02.115
[18] Chowdhury, Harun, Bavin Loganathan, Israt Mustary, Hazim Moria, and Firoz Alam. "Effect of various deflectors on drag reduction for trucks." Energy Procedia 110 (2017): 561-566. https://doi.org/10.1016/j.egypro.2017.03.185