Pickup Truck Wake Analysis

Richard D. Leyton-Romero

December, 2022

1 Abstract

Aerodynamic drag is one of the largest contributors to the low fuel economy of pickup trucks. An area for optimization lies in the bed configuration of the vehicle. Three pickup truck bed geometries are tested in Star CCM+ and a wind tunnel experimental procedure is planned. The geometry tested is the Lokhande model which is a 1/12 scale generic pickup truck model. Bed lengths are decided to be the three deliverable configurations available on the current generation Ford F-150 which are 5.5 ft, 6.5 ft, and 8 ft. An analysis of their flow and wake patterns is conducted and the three geometries are compared to learn how bed length affects lift and drag performance. Furthermore, for the best performing of the three configurations, a tonneau cover and spoiler are placed to begin understanding how a NASCAR Craftsman Series race truck performs aerodynamically and if there any applications here to use for road vehicles.

2 Introduction

According to Edmunds (2022), the Ford F-Series pickup truck continues to be the number one most sold vehicle in the United States, followed closely in second by the Chevrolet Silverado. These vehicles are known for their utilitarian advantages at the sacrifice of fuel economy. Although the United States is currently seeing falling prices of fuel in late 2022, the issue of efficiency remains important as humanity seeks to confront climate change. Thankfully, the combination of development in the vehicle's powertrain performance, tires and aerodynamics have made significant improvements over time. Given that new models tend to get larger frontal areas with respect to year, it is of great importance to continue to understand the airflow around the pickup to optimize the drag and gain fuel economy.

Initial aerodynamic research on pickup truck bodies was done with clay modeling and the original focus was to enhance airflow to the heat exchangers within the vehicle. Butz et al., (1987) is credited for being one of the foundational studies for aerodynamic pickup truck development. Their parametric study assessed almost 200 configurations of body shape including the addition of a tonneau cover to reduce drag.

Simple body shapes such as the Ahmed model have been in use for aerodynamic development since the 1980's. One of the key benefits of this model was the ability to gain an understanding of the effects of rear-end configuration to the wake of a vehicle (Le Good & Garry 2004). However, in recent decades, pickup trucks have dominated new car sales. This means that the most ubiquitous vehicle on the road cannot be modeled with the standard notchback, fastback, or squareback configurations. Lokhande et al. (2003) introduced a generalized pickup model that has been used several times in research. The form is a 1/12 scale of generalized pickup truck dimensions. Usage of this model quickly led to further understandings of the complexity of the flow interaction between the cabin and the bed. Al-Garni et al. (2003) discovered the symmetrical nature of the flow over the Lokhande model and in concurrence confirmed the flow pattern over the cabin into a vortex over the bed surface, as well as the downwash from the tailgate to the ground. Later, Cooper (2004) formulated a study to test different tailgate configurations. Many people had been removing their tailgates or driving with them in a lowered position to get better fuel economy. Cooper's series of experiments at the National Research Council of Canada's wind tunnel determined that the best configuration is with the tailgate up.

With pickup trucks being delivered in several different configurations such as crew cab, extended cab, short bed and long bed, it has become increasingly more important to understand how these configurations affect air flow. Ha et al. (2010) used a generic model similar to the Lokhande to test various aerodynamic measures with respect to bed length and bed height. They found that each bed height has its own optimal length. With a gained understanding of how flow reattachment at the tailgate controls the drag, the same group published a study experimenting with the addition of a rear downward flap placed at the location of the third brake light. The results of their study are promising, and they do manage to find and optimize a length and angle for their respective bed configuration (Ha et al. 2011).

Recently, engineers at Ford have published and made a new open-source general bluff body geometry for CFD studies called the GTU. The body features several different configurations of bed and cabin length as well as bed covers and toppers. This geometry can accurately describe most body-on-frame vehicle models sold in North America (Woodiga et al. 2020).

Carrying on, the aim of this study is focused on utilizing different available bed configurations to quantify and optimize aerodynamic performance. This will be achieved by:

- Visualizing the flow into, out of, and around the bed.
- Numerically analyzing the lift and drag data for the whole vehicle in different configurations.

3 Simulation Methods

This study uses Star CCM+ for its simulation code. Star CCM+ is a simple to use computational fluid dynamics (CFD) code and is the preferred tool of choice at Oxford Brookes. The geometry utilized (Figure 1) is like the Lokhande model with few changes, namely the vehicle is kept as two-dimensional as possible except for the wheels which are placed as would normally be in a vehicle. The purpose of this is to minimize the impact of the front geometry before the flow reaches the bed. A detailed drawing is provided in the appendix.

Two options were available for flow domain modeling. Either model it after a wind tunnel or capture an open road scenario. Multiple studies involving the Lokhande model chose the first method. These were typically characterized by a model that was already scheduled to be tested in a particular wind tunnel. Furthermore, it can be observed that under high blockage conditions, the air artificially accelerates around the body leading to inaccuracies in flow structure (Cooper 2004). For these reasons, an "open road" configuration was simulated which featured a 2 m wide by 3 m tall and 8 m deep volume.

Given the large flow domain, modifications needed to be made to bring computation time to a manageable scale. It was decided to simulate only half the vehicle along a symmetrical plane. This

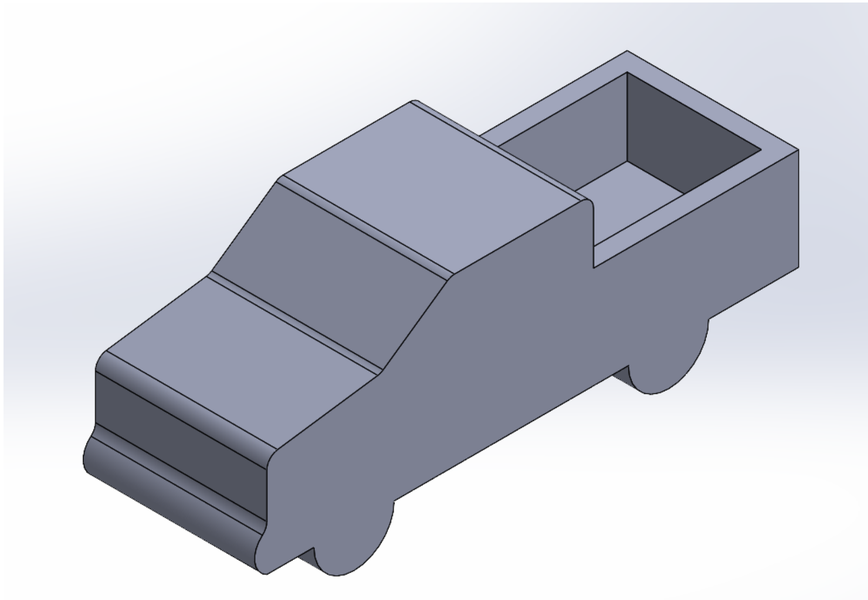

Figure 1: 6.5ft baseline model

proves to still be an accurate representation of the flow (Yang & Khalighi 2005, Holloway et al. 2009, Mokhtar et al. 2009, Gamil et al. 2021, Howard et al. 2021).

Meshing was completed with triangular surface elements, tetrahedral volumetric cells, and multiple prism layers. This pattern best complements complex shapes and flows while still being computationally feasible for a modern 12 core processor. An inner domain, two vehicle lengths (2L) long, 1L tall and 1L wide is used to make a very fine mesh. A wake refinement is also undertaken to best capture peak areas of interest. Similar meshing patterns are seen by (Lokhande et al. 2003, Yang & Khalighi 2005, Holloway et al. 2009, Ha et al. 2011, Gamil et al. 2021). Detailed views of the meshing pattern are seen in Figures 2 and 3

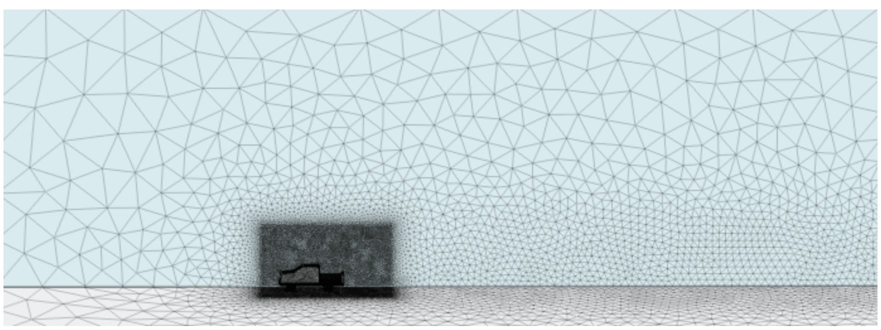

Figure 2: Flow Domain Mesh

In selection of physics models to be used, for open road domains, the k-omega turbulence model was used by (Ha et al. 2011) for its better operation in adverse pressure gradients and separating flows. However, in our initial pilot studies it was unable to fully characterize the vortex inside the bed of the model. For this study, Realizable k-epsilon was used as it provided the best convergence

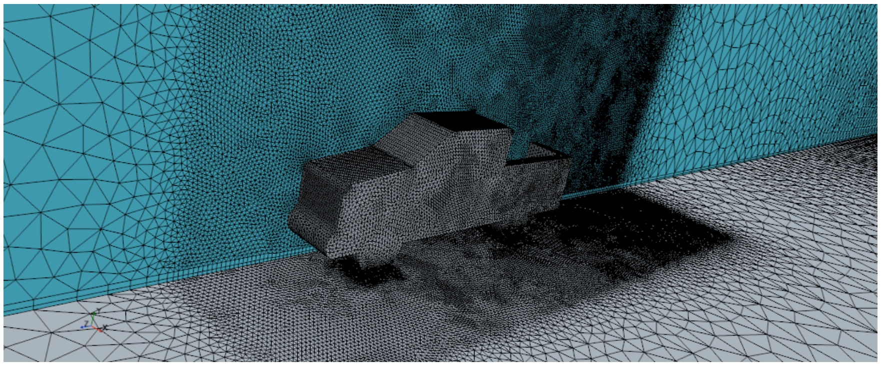

Figure 3: Detailed Surface Mesh

characteristics during operation. Detailed simulation parameters are shown in Table 1

Simulation Settings	
Code	Star CCM+
Flow Domain	Open Road
Volumetric Cell	Tetrahedral
Surface Cell	Triangular
Prism Layers	12
Prism Layer Thickness	1.0E-5 m
Growth Rate	1.3
Cell Count	$\sim$ 4 million
Space	Three Dimensional
Material	Gas
Flow	Segregated Flow
Equation of State	Constant Density
Time	Steady
Viscous Regime	Turbulent
RANS	Realizable k-epsilon
Truck Frontal Area (m^2)	0.029146744
Air Density (kg/m^3)	1.18415
Free Stream (m/s)	25

Table 1: Simulation Settings

4 Proposed Experimental Methods

To validate these findings, an experimental test must be conducted. Previous studies on Lokhande models were done on wind tunnels with high blockage ratios (Al-Garni et al. 2003). This should be avoided as it causes air around the body to accelerate and disrupt the natural behavior of the vortices we wish to inspect. A mid-level approach would be to use an open section wind tunnel of similar size. This will allow the air to go as wide as necessary without having wall and roof effects. The best approach for our model would be to utilize a wind tunnel that can best accommodate our "open road" style flow domain.

A model needs to be produced that is 1/12 standard size. This can be done rapidly by 3D printing a core and using clay to smoothen the surface. The model should be modular to incorporate different bed lengths. With pressure tap locations exactly where Lokhande specifies in Figure 4. The NASCAR style rear spoiler does not need to be an airfoil shape. It is a simple triangular extrusion, and the exact details are in the appendix drawing.

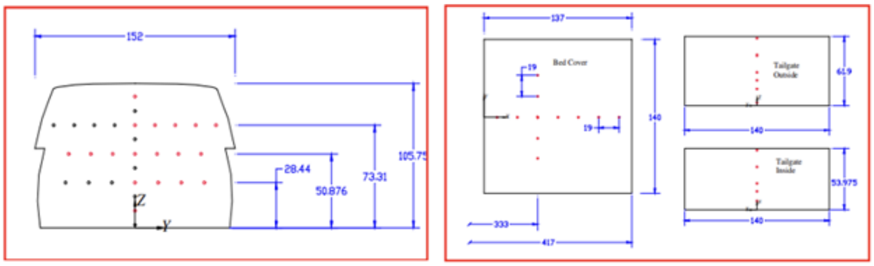

Figure 4: Pressure Tap Locations Lokhande et al. (2003)

Particle Image Velocimetry (PIV) is the flow visualization technique of choice. Images should be taken every 0.01 m along the length of the body. Moreover, an appropriate balance gathering force and moment data in all six degrees of freedom would be optimal for future data analysis. 5 trials should be run: 3 for each configuration, 1 for the 8 ft NASCAR, and a final return to baseline to quantify error.

5 Results and Discussion

The wake of the pickup truck body can be characterized by two distinct vortices shown in Figure 5. The first has its centroid in the bed and the second is immediately behind the tailgate. The vortex within the bed spans the area of the floor up to the height of the top edge of the cabin and rotates three dimensionally as seen in Figure 6. The air within this vortex travels at low speeds and circulates several times before reattaching to the top of the tailgate and finally escaping. This reattachment has been linked to an increase in drag (Ha et al. 2011). The reattached air then immediately separates again and downwashes either into or past the second vortex behind the tailgate. Like the first vortex, the second vortex traps air behind the tailgate. The behavior of this circulation is very similar to the squareback condition characterized in Schuetz (2016).

Katz (2006) states that vortices are driven by energy. Thus, the reduction of these large vortices are the simplest effort that can be done to reduce drag and improve fuel efficiency. From a motorsports perspective, this ideology can be used to convert that energy from drag into lift.

As we explore the three main configurations of bed length (Figure 7), we can certainly see the trend of reducing/tapering first and second vortex volume as bed length is increased. This effect is at a maximum with the 8 ft configuration. This results in less downwash and the air travels a path much more in line with the vehicle axis of direction. Subsequently, as bed length is increased, drag and lift are reduced.

The data in Table 2 shows that between the baseline 6.5 ft configuration and the 8 ft configuration, the decrease in drag is unnoticeable but the decrease in overall lift coefficient is substantial at

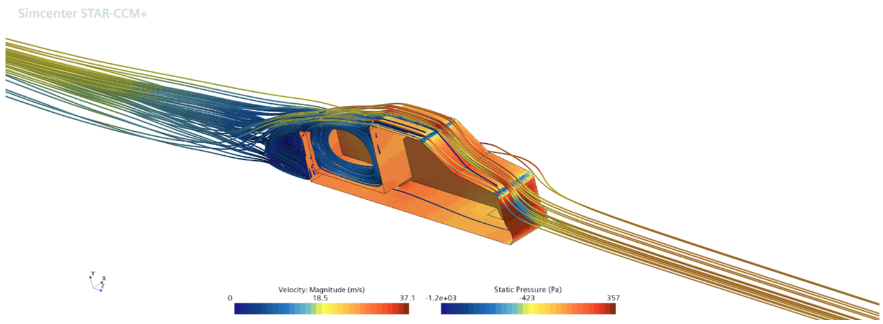

Figure 5: 3D Vortex Flow 6.5ft Baseline

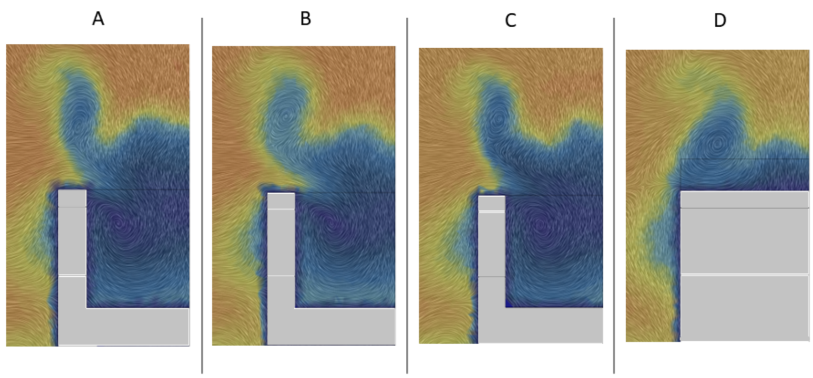

Figure 6: A. 5.5ft, B. 6.5ft, C. 8ft, D. NASCAR 8ft.

14.3 percent change. Certainly, the lowest performing configuration was the short 5.5 ft bed. This bed configuration sees a 1.6 percent and 6.4 percent gain in drag and lift coefficients respectively over baseline.

For a road vehicle, the more fuel-efficient option may be the 6.5 ft bed given the increased mass associated with the longer configuration. However, a longer bed will have better high-speed stability given the large reduction in lift coefficient. Compared to baseline, the short bed has a mildly concerning gain in lift coefficient and would perhaps be best suited for low-speed applications.

Finally, as seen in Figure 7D and Figure 8, the addition of the rear tonneau cover significantly reduces the volume of the first vortex. Given enough length, the flow will escape the first vortex and reorient itself along the bed cover. Then the effect of the NASCAR style rear spoiler is twofold: first it increases the volume of the second vortex generating more drag and second it generates significant downforce by creating an upwash effect from the underbody flow. This is a highly advantageous effect for racing applications. This configuration applied to the 8 ft model generated a drag coefficient gain of 24.5 percent and a lift coefficient reduction of 184 percent. A small drag penalty for a large increase in downforce.

Continued work should be done to see how the effects of the NASCAR style additions affect the flow of the shorter bed configuration. Shorter beds tend to be more economical and their reduced mass results in better city MPG. North America is characterized by long, straight-line highway driving, and a short bed pick-up should be optimized to perform well in these scenarios.

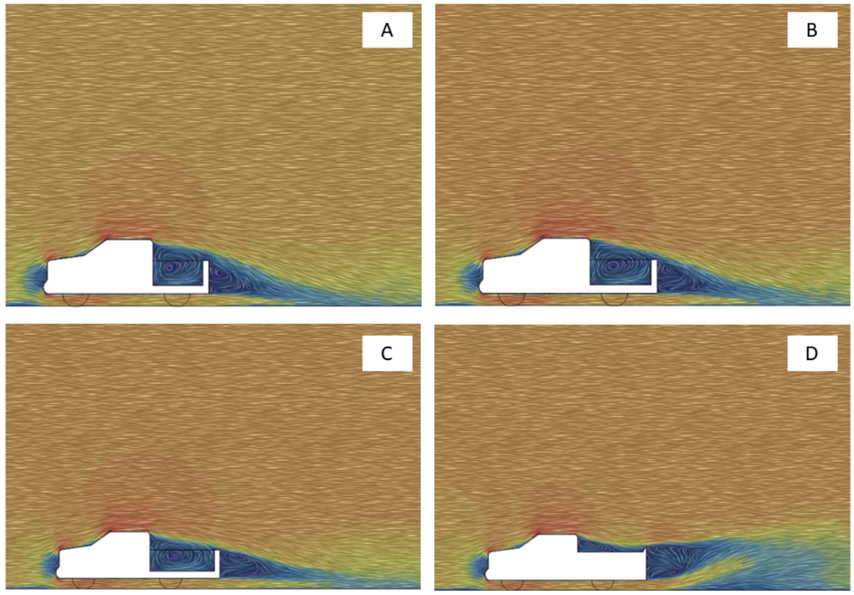

Figure 7: A. 5.5ft, B. 6.5ft, C. 8ft, D. NASCAR 8ft.

Results	5.5'	6.5'	8'	NASCAR
Cd	0.364	0.359	0.358	0.446
Cl	0.123	0.115	0.099	-0.083
l/d	0.337	0.321	0.275	-0.186

Table 2: Aerodynamic Results

6 Conclusion

A CFD study was undertaken to quantify the differences in pickup truck bed length to drag and lift coefficients. Geometries were modified from a Lokhande model and bed lengths were chosen based on what is available to purchase in 2022. Results show that longer beds are generally more efficient and shorter beds are at risk of high-speed instability. Afterwards, a brief discussion at how a NASCAR Craftsman Series style bed affects the flow pattern and a mention of how these results can be applied to optimize short-bed stability. Furthermore, an experimental method is planned to confirm these findings.

6.1 Reflection

This study provided an excellent training ground for hands-on experience working with published literature and CFD studies. In the process, much was learned about the flow fields of a very popular vehicle body as well as best practices for simulation. Currently, the best recommendation for anyone with a pickup truck is to aim for a longer bed and use a cover. There are many opportunities to continue this research. For example, with more advanced computing power, more realistic

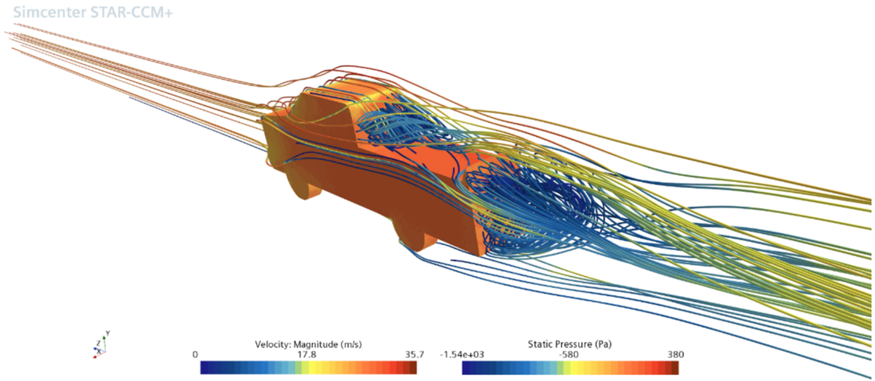

Figure 8: 3D Vortex Flow NASCAR

bluff bodies such as the GTU can be used and parametric studies involving small yaw angles and moving ground simulation can be conducted. At a more basic level, this experimental data can be compared better with the output data of previously published work. Moreover, these future studies can be carried out experimentally at the proposed site. Doing so would yield one of the most advanced studies in pickup truck flow characteristics.

References

Al-Garni, A. M., Bernal, L. P. & Khalighi, B. (2003), 'Experimental Investigation of the Near Wake of a Pick-up Truck', pp. 2003–01–0651.
URL: *https://www.sae.org/content/2003-01-0651/*

Butz, L. A., Donavan, P. R., Gondert, T. R., MacDonald, R. A. & Wood, D. H. (1987), '1988 Chevrolet/GMC Full-Size Pickup Truck Aerodynamics', p. 872274.
URL: *https://www.sae.org/content/872274/*

Cooper, K. R. (2004), 'Pickup Truck Aerodynamics - Keep Your Tailgate Up', pp. 2004–01–1146.
URL: *https://www.sae.org/content/2004-01-1146/*

edmunds (2022), 'Most Popular Cars in America', *Edmunds* .
URL: *https://www.edmunds.com/most-popular-cars/*

Gamil, A. I., Syam, T., Arab, M. A. & Ghani, S. (2021), 'Areadynamic Analysis of GMC Sierra Pickup Truck', pp. 535–545.

Ha, J., Jeong, S. & Obayashi, S. (2010), 'Flow characteristics of a pickup truck with regard to the bed geometry variation', *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* **224**(7), 881–891.

Ha, J., Jeong, S. & Obayashi, S. (2011), 'Drag reduction of a pickup truck by a rear downward flap', *International Journal of Automotive Technology* **12**(3), 369–374.

Holloway, S., Leylek, J. H., York, W. D. & Khalighi, B. (2009), 'Aerodynamics of a Pickup Truck: Combined CFD and Experimental Study', *SAE International Journal of Commercial Vehicles* **2**(1), 88–100.
URL: *https://www.sae.org/content/2009-01-1167/*

Howard, K., Norman, P., Woodiga, S., Lewington, N. & Hupertz, B. (2021), 'A Detailed Aerodynamics Investigation of Three Variants of the Generic Truck Utility', pp. 2021–01–0950.
URL: *https://www.sae.org/content/2021-01-0950/*

Katz, J. (2006), *Race car aerodynamics: designing for speed*, rev. 2nd ed edn, R. Bentley, Cambridge, MA. Section: viii, 307 pages : illustrations ; 27 cm.

Le Good, G. M. & Garry, K. P. (2004), 'On the Use of Reference Models in Automotive Aerodynamics', pp. 2004–01–1308.
URL: *https://www.sae.org/content/2004-01-1308/*

Lokhande, B., Sovani, S. & Khalighi, B. (2003), 'Transient Simulation of the Flow Field Around a Generic Pickup Truck', pp. 2003–01–1313.
URL: *https://www.sae.org/content/2003-01-1313/*

Mokhtar, W. A., Britcher, C. P. & Camp, R. E. (2009), 'Further Analysis of Pickup Trucks Aerodynamics', pp. 2009–01–1161.
URL: *https://www.sae.org/content/2009-01-1161/*

Schuetz, T. (2016), *Aerodynamics of road vehicles.*, fifth edition / edn, SAE International, Warrendale, Pennsylvania. Section: xix, 1289 pages : illustrations ; 27 cm.

Woodiga, S., Howard, K., Norman, P., Lewington, N., Carstairs, R., Hupertz, B. & Chalupa, K. (2020), 'The GTU: A New Realistic Generic Pickup Truck and SUV Model', pp. 2020–01–0664. **URL:** *https://www.sae.org/content/2020-01-0664/*

Yang, Z. & Khalighi, B. (2005), 'CFD Simulations for Flow Over Pickup Trucks', pp. 2005–01–0547. **URL:** *https://www.sae.org/content/2005-01-0547/*