© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

http://penerbit.uthm.edu.my/ojs/index.php/ijie
ISSN : 2229-838X e-ISSN : 2600-7916

The International Journal of Integrated Engineering

Aerodynamic Analysis on the Effects of Frontal Deflector on a Truck by using Ansys Software

Pavin Kumaar Varadarajoo[1*], Izuan Amin Ishak[1*], Daniel Syafiq Baharol Maji[1], Nurshafinaz Mohd Maruai[2], Amir Khalid[1], Norirda Mohamed[1]

[1]Department of Mechanical Engineering Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

[2]Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia (UTM), Jalan Sultan Yahya Petra, 54100, Kuala Lumpur, MALAYSIA

*Corresponding author

DOI: https://doi.org/10.30880/ijie.2022.14.06.008
Received 7 March 2021; Accepted 27 February 2022; Available online 10 November 2022

Abstract: The main causes of aerodynamic drag for automotive vehicles are the flow separation at the rear end of the vehicles. By reducing the drag force, it is possible to increase the fuel economy. Aerodynamic component i.e. Frontal Deflectors (FD) commonly used on trucks to prevent the flow separation. Frontal Deflectors themselves do create the drag, but they also reduce drags by preventing flow separation at downstream. The main aim of this paper is to quantify the effect of frontal deflectors on improving trucks aerodynamics. In this study, the simulation were ran for 6 different shapes of FD which acquires different height and different placement of FD that is mounted on the truck from the frontal roof by using ANSYS Fluent software. The design of the truck has been done in SOLIDWORK 2018 and the same design is used for analysis in ANSYS (Fluent). The two equation models used in this study are $k-\varepsilon$ with applying the Reynolds-averaged Navier Stokes (RANS) equations for the behaviour of fluid flow around the truck. The Reynolds number used is $Re = 1.1 \times 10^6$. Based on the result, all the FD's resulted in reduction of C_d. The drag coefficient of all FD models differs. The velocity streamline acquired is different between the Frontal Deflector models mounted on the truck and the flow structure and vortex formation differs in various pattern formation. FD 4 produces the least value of drag. Hence, the efficiency of the truck improves. Therefore, FD 4 is the best model as the C_d acquired is 0.508 with the height (15 mm) and placement of (230 mm) is the best FD to be used on a truck. Consequently, the drag reduction percentage of FD 4 compared to the truck without a FD is 32.2%.

Keywords: Aerodynamics, coefficient of drag (C_d), CFD, Frontal Deflector (FD), trucks

1. Introduction

A variety of add-on devices for tractor-trailer vehicles had been suggested to reduce the aerodynamic drag of these vehicles. The aerodynamic drag plays an important role in fuel consumption and the performance of a truck. Increasing fuel prices and awareness of the environment, the industry is striving to design more fuel-efficient vehicles and to improve existing technologies. Drag force is one of the essential aspects of vehicle design [1]. Nowadays, most trucks are designed to achieve ideal aerodynamic performance i.e. low in drag force, so that the trucks will perform well. It is reported that a typical truck with an average drag coefficient of 0.6 and driving at 110 km/h spends 65% of its fuel overcoming aerodynamic drag [2], [3]. It is also pointed out that 70% of the brake power of a vehicle engine is consumed to overcome the aerodynamic drag generation of the vehicle at 100 km/h [2].

*Corresponding author: *pavin_kumaar@yahoo.com*

The main problem faced by truck manufacturers is the air resistance associated with high-speed movement. Since the trucks have a large frontal area and the presence of a trailer also leads to the truck experiencing significant resistance, where it is needed to overcome. This can be attained by several strategies where it can be recognized that one of the most effective ways to reduce drag is to change the body vehicle geometry i.e. adding Frontal Deflectors (FD). FD streamline is the step between the top part of a cab and the front part of a container [4]. The results indicate that aerodynamic performance might well be enhanced by alerting the vehicle's bodylines, i.e. by appropriately designing various component profiles, the vehicle's drag coefficient may be kept at a minimum value. A well-designed FD i.e wide enough to extend across the front of the trailer can reduce wind resistance to a certain extent. Thus, optimizing the angle of the FD can reduce the drag force acting on the vehicle, thereby reducing fuel consumption [5].

A Frontal Deflector (FD) affixed to the cab of a truck or tractor is the most commonly used drag-reducing device to regulate the forebody flow. Although FD is being used commercially nowadays, the design used by truck owners differs as they tend to modify it without any proper knowledge of aerodynamic drag. There are even trucks being used without an FD whereby the truck owners believe it is not needed. In this paper, the importance of using a proper and simple design of FD which can result in multiple advantages had been shown. By installing an FD on a truck, fuel consumption can be utilised while also conserving natural resources. It can be seen that the FDs used in most Malaysian trucks manufactured locally are not aerodynamically efficient as they can increase the overall aerodynamic drag.

1.1 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is a branch of fluid dynamics that uses computational methods and algorithms to solve and evaluate a fluid flow problem. CFD modelling is based on fundamental governing equations of fluid dynamics like the conservation of mass, momentum, and energy. CFD helps forecast fluid movement behaviour based on statistical models using analytical methods. It is now widely used and is acceptable as a valid engineering tool in the industry. The field of CFD became a commonly applied tool for generating solutions for fluid flows with or without solid interaction. In an exceedingly CFD analysis, the examination of fluid flow under its physical properties like velocity, pressure, temperature, density, and viscosity is conducted. The CFD simulation needs to be applied on a truck without an FD to verify the streamline of aerodynamic drag distribution. To generate a virtual solution for a natural phenomenon related to fluid flow, without compromise on accuracy these properties need to be considered at the same time [6]. Besides, the determination of proper numerical methods to generate a path through the solution is as important as a mathematical model. The software, which the analysis is conducted is one of the key elements in generating a sustainable product development process, as the number of physical prototypes can be reduced drastically. There are several simulation software that can be used to conduct the CFD analysis such as ANSYS Fluent, OpenFoam, PowerFlow, Star CCM+, Autodesk CFD, SimScale, etc. All software is available to carry out CFD simulations. The mathematical model differs depending on the nature of the issue, such as heat transfer, mass transfer, phase transition, chemical reaction, etc. [7]. The turbulence model consists of several models i.e. RANS, URANS, LES, DES, DNS, etc.

1.2 Aerodynamics

Aerodynamics is the study of how air is associated with moving bodies. This is the study of forces and the consequent movement of objects through the air. Understanding the airflow around an object allows for the calculation of forces and moments acting on the object. In the automotive sector, aerodynamics is the study of street vehicle streamlined features. Its main goals are to reduce drag and wind noise, mitigate noise pollution, and also avoid undesired lifting forces and other causes of highspeed aerodynamic instability. Aerodynamics is primarily concerned with the drag force, which is caused by air passing over and around a solid body [8], [9]. For certain classes of trucks, it is also important to produce down-force, hence, it will improve the traction [10]. Aerodynamic drag consists of two main components which are skin friction drag and pressure drag. The most prominent of the two is the pressure drag. The pressure drag is caused due to the shear forces acting between the two layers of fluid [11]. Pressure drag constitutes more than 80% of the total drag and is highly dependent on vehicle geometry due to the separation of the boundary layer from the rear window surface and the creation of a wake region behind the vehicle [12]. Skin friction drag is the resistant force exerted on an object moving in a fluid and is caused by the viscosity of fluids and developed from laminar drag to turbulent drag as a fluid moves on the surface of an object [13]. Aerodynamic drag is a mechanical force generated by the movement of the truck through the air as it accelerates forward. It can be thought of as air resistance. Essentially, it is an opposing force that your truck needs to overcome to move forward. The stronger the effects of drag, the more energy the truck requires to move at the desired speed. Between these three forces, one can describe most of the interactions of the airflow with a vehicle body. The equation of coefficient of drag C_d is shown in equation 1.

$$C_d = \frac{F_d}{0.5\,\rho V^2 A} \qquad (1)$$

where, F_d is the drag force, ρ is the air density, A is the effective frontal area and V is the flow of velocity.

1.3 Reynolds Number

The Reynolds number (Re) is used to study fluids as they flow. It determines whether fluid flow is laminar or turbulent [14]. Usually, flowing fluids go along streamlines. If a flow is laminar, it will pass along smooth streamlines fluids. These streamlines break up if the flow is turbulent and the fluid moves irregularly. Turbulent flow creates greater friction drag onto an aircraft. However, it also keeps the flow attached over its surface. In general, Re can be expressed as equation 2:

$$Re = \frac{V \times L}{\upsilon} \qquad (2)$$

where V is the velocity of the fluid, L is the characteristic length of the vehicle and υ is the kinematic viscosity of air. The Re and aerodynamics relationship is the parameter used for viscosity. The exact values for when the Re are 'large' or 'small' is not well defined but change depending on an actual problem.

1.4 Aerodynamics On Trucks

The need to make trucks more aerodynamically resulted in higher fuel prices and the need for owners to remain competitive in costs. Improving typical current-generation truck aerodynamics is not easy. It requires an understanding of the aerodynamics involved, the tests required to determine the changes in vehicle shape, and the cost savings of the vehicle service reasonable. On the contrary, the shape of a road vehicle is primarily determined by functional, economic, and last but not least, aesthetic arguments. Aerodynamic characteristics are not usually intentionally generated. Depending on the specific purpose of each type of vehicle, the objectives of aerodynamics differ widely according to its model [5]. While the process of weighing the relative importance of a set of needs from different disciplines is generally comparable to that of other branches of applied fluid mechanics.

1.5 Frontal Deflector

The frontal deflector (FD) is an aerodynamics device that modifies the boundary layer of the fluid motion by bringing momentum from the outer flow region into the inner flow region of the wall-bounded flow. It consists of a large and broad vane commonly attached to the vehicle surface to reduce the aerodynamic drag. Fitted to the top of the truck, this flat or contoured plate can be placed at various angles to match the body and is perhaps the most effective single add-on for trucks with bodies of differing heights. The more the body extends, the more a well-adjusted FD can deliver possible advantages in reducing drag. The drag of the truck's cab would be much higher when the FD is reduced to its horizontal position [15]. There is a high degree of airflow isolation between the cab and the container. To reduce the airflow resistance and to produce much finer airflow, a drag-reduction system is mounted on the cab as a reference to the pressure contour of the alignment plane of the truck, which indicates a smoother airflow between the container and the driver's cabin of the strengthened vehicle as well as a reduced vortex.

2. Methods

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that analyzes and solves problems involving fluid dynamics and heat transfer using numerical analysis and data structures. CFD is mainly the practice of replacing the governing partial differential fluid flow equations with numbers and advancing these numbers in space and/or time in order to provide a final numerical definition of the whole flow area of interest [16].

2.1 Model Description

In this analysis, the model used is a generic conventional model (GCM) truck built on the geometry of modern current truck generation [17], [18]. Using SOLIDWORKS software, this model is designed by referring to the modern current truck generation. Study of the flow pattern around a truck model, its drag coefficient when installed with and without a frontal deflector is used as a guideline for this analysis.

The generic conventional truck model without a frontal deflector is shown in Fig. 1, and the size for this truck model is shown in Table 1. The design of the truck used is a one-eighth-scaled model from the original size.

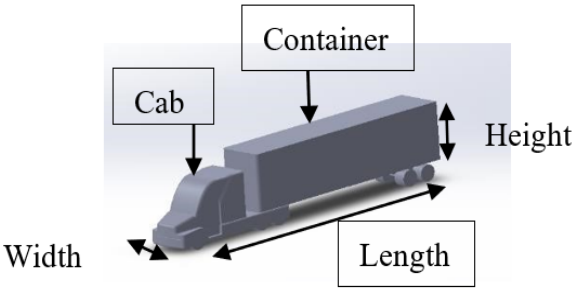


Fig. 1 - Generic conventional model of the truck

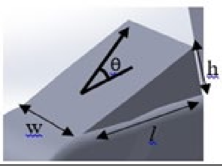
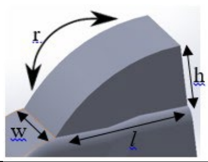
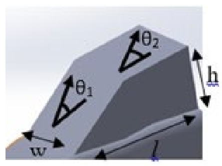
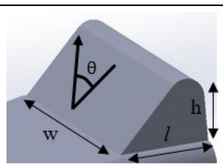
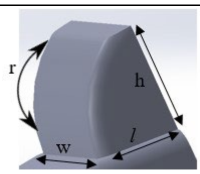
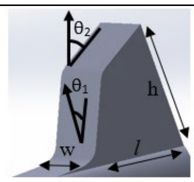
Table 1 - Dimension of the generic conventional model of the truck

Configuration of the generic conventional model	Dimension (mm)
Length	2500
Width	340
Height	485

2.2 Design of Frontal Deflector

Six distinct frontal deflectors are used in this analysis namely FD 1, FD 2, FD 3, FD 4, FD 5 and FD 6. Concerning the height, the boundary layer's thickness is measured on the basis that the optimum height of the FD would be almost equal to the thickness of the boundary layer at the rear end of the cab roof, which is located in front of the separation point about 15 to 25 mm. The optimum height for the FD is found to be up to approximately 30 mm. Table 2 shows six frontal deflectors different in shape and angle of fillet i.e. θ = 65º, radius = 45, θ = 55º. The width is constant where w = 74 mm meanwhile the length varies where from FD 1 to FD 3 is l = 25 mm and FD 4 to FD 6 is l = 40 mm. This is due to the change of size of the cab on the vehicle model.

Table 2 - Design of frontal deflectors

FD 1	FD 2	FD 3
		
Dimensions (mm)	Dimensions (mm)	Dimensions (mm)
Height = 30 **Thickness** = 10 **Angle, $\boldsymbol{\theta}$** = 65º	**Height** = 30 **Thickness** = 15 **Radius** = 45	**Height** = 30 **Thickness** = 10 **Angle** = θ_1= 55º, θ_2 = 30º
FD 4	**FD 5**	**FD 6**
		
Dimensions (mm)	Dimensions (mm)	Dimensions (mm)
Height = 15 **Thickness** =10 **Angle, $\boldsymbol{\theta}$** = 45º	**Height** = 25 **Thickness** = 15 **Radius** = 30	**Height** = 25 **Thickness** = 10 **Angle** = θ_1= 20º, θ_2 = 60º

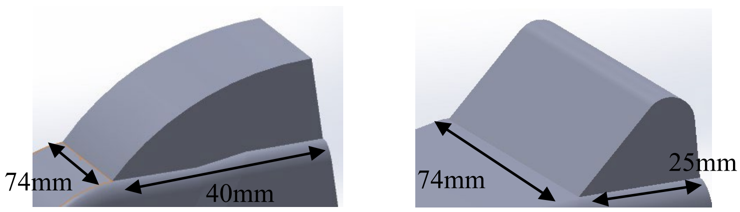
2.3 Placement of Frontal Deflector

The frontal deflector should be placed at the most suitable point which is on the roof of the cab and in between the cab and the trailer, a point where immediate upstream of the flow separation point exists and a point at an optimum distance of between 215 mm and 230 mm from the frontal roof as shown in Fig. 2. Where FD 1, FD 2, and FD 3 are placed at 215 mm from the frontal area and FD 4, FD 5, and FD 6 are placed at 230 mm from the frontal area.

In this study, six different types of the frontal deflector are placed on top of the roof at a time with two different amounts of spacing from the frontal area as seen in Fig. 2 and Fig. 3. The placement of designs 1 to 3 (FD 1, FD 2, FD 3) differs from the placement of designs 4 to 6 (FD 4, FD 5, FD 6). Fig. 4 illustrates the truck mounted with a frontal deflector on top of the cab's roof with two different placements.

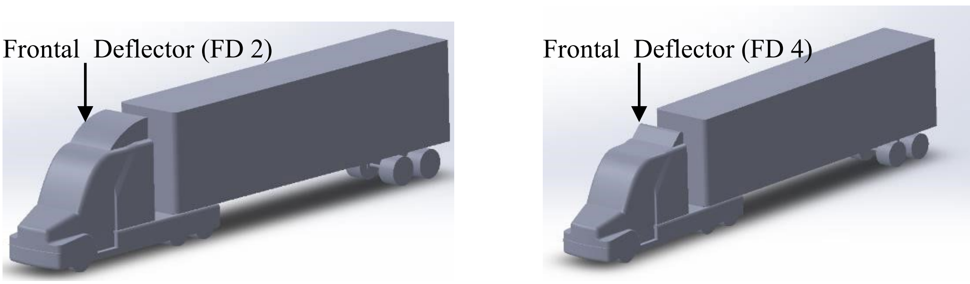
(a) (b)

Fig. 2 - Two different locations of the frontal deflector to be mounted (a) location for FD 1, FD 2 and FD 3; (b) location for FD 4, FD 5 and FD 6



(a) (b)

Fig. 3 - Example of the basic dimensions of the frontal deflectors (a) FD 2; (b) FD 4

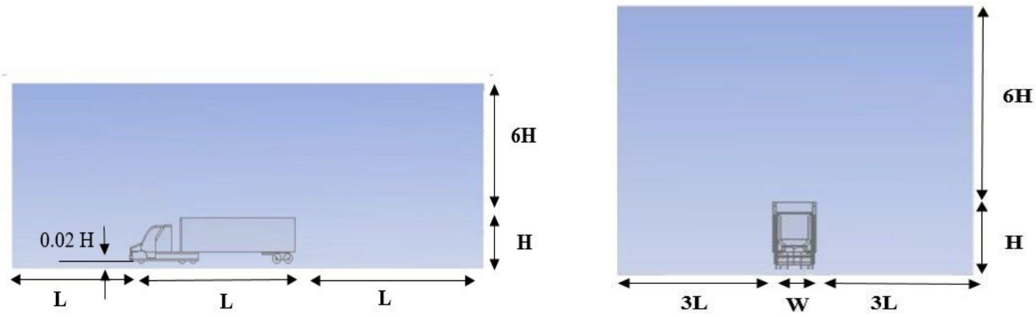


(a) (b)

Fig. 4 - Example of the truck models with frontal deflector mounted on (a) FD 2; (b) FD 4

2.4 Steps Performing the Simulation

The first step is to construct a 3D model of trucks with different FD before the simulation runs. A 3D model is imported from SOLIDWORKS into ANSYS within this project. The creation of model is generated in ANSYS and the fluid domain is created prior to the meshing phase. A bigger enclosure has been created to ensure a better mesh and more accurate result in the later stage. Analytical tools utilize the fluid domain or enclosure to simulate fluid. The size of enclosure as shown in Fig. 5 i.e. H = 0.52 m, L = 2.5 m, W = 0.34 m.



(a) (b)

Fig. 5 - Size of enclosure (a) side view; (b) front view

The model used an automatic global mesh which is a combination of tetrahedron patch conforming and sweep methods. The model's position of the inlet, outlet, and wall was specified in the meshing process. The total number of meshes is 150,585 elements and the nodes are 806,783. Once the meshing process has been completed, the model's boundary conditions such as velocity inlet, pressure outlet, vehicle surface, wall roughness, and flow form should be set as seen in Fig. 6 in the setup process. The Reynolds number used based on the width of the truck is $Re = 1.1 \times 10^6$ as

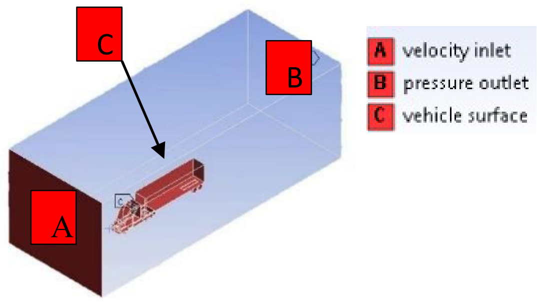


Fig. 6 - The boundary conditions used in the numerical investigation

referred to in the validation paper by Pointer *et al.* [19]. The fluid medium is air which is based on the actual situation. The details of the boundary conditions applied can be seen in Table 3.

Table 3 - Details of the boundary conditions

Details	Boundary Conditions	Value
Inlet	Velocity Inlet	51.5 m/s
Outlet	Pressure Outlet	0 Pa (gauge)
Wall	Wall Boundary	Non-equilibrium wall
Vehicle Surface	Wall Boundary	No-slip
Reference	Ambient	101.325 kPa

3. Results and Discussion

3.1 Quantitative Effect of FD on Aerodynamics

The results of drag coefficient (C_d) for trucks that are mounted with different types of frontal deflectors were presented and visualized in Fig. 7. The FDs used in this study are quite similar to each other but varies in the terms of placement. The truck without an FD shows the highest value of C_d which is 0.703. The best FD design is the 4th design (FD 4) where the C_d acquired is the lowest compared to other designs. In this study, the efficiency had increased to 72.3% by mounting FDs on the truck with regards to FD 4 where the C_d is 0.508. The FD 2 holds the highest C_d amongst the model with FD which is 0.582 due to its forepart being highly exposed to the air where its forepart structure is high compared to other designs.

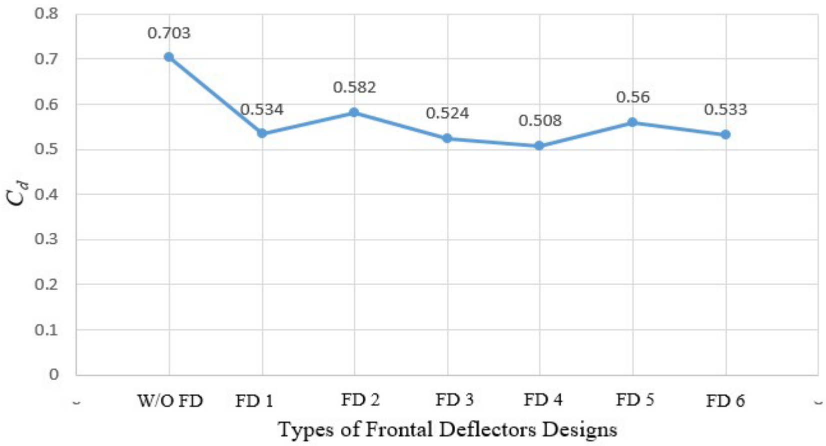
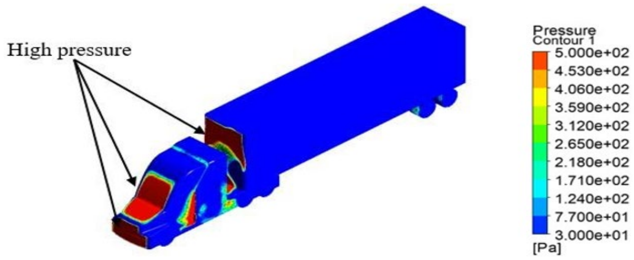


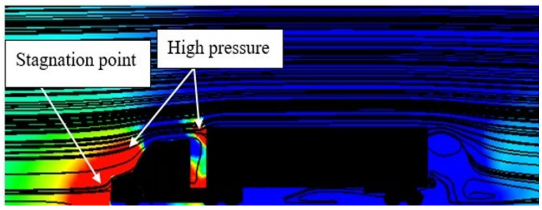
Fig. 7 - Coefficient of drag (C_d) for trucks mounted with different designs of frontal deflectors and without frontal deflector

3.2 Flow Structure Analysis on the Base Case Model

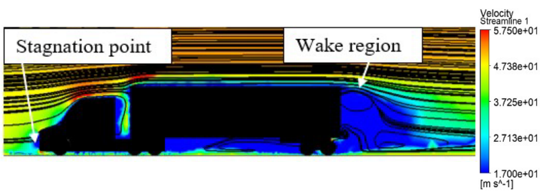
As shown, Fig. 8 (a) – (b) represents the pressure and streamline on the truck and surrounding without an FD being placed on it. Without FD, the pressure exerted on the truck and the surrounding area is massively disrupted. This disrupted flow with high pressure being exerted on the truck will cause an increase in C_d. As can be seen, the stagnation point appears as a result of flow separation and when the local velocity of the fluid is zero. In Fig. 8 (b) and (c) it can be seen once the flow advance towards the truck's frontal region, the flow breaks and divides with some going over and some under the surface of the truck. The frontal area of the truck indicates a low-velocity flow as that particular area has a high-pressure region which can be considered as the stagnation point. High pressure at the frontal area of the container is due to a direct impact on the large area and flow separation occurs. When there is no FD is mounted on a truck’s cab, the separation develops on a truck’s frontal area, i.e. cab and container where it rises the average pressure occurring on the front surface, resulting in an increased force countering the motion of the truck.



(a)



(b)



(c)

Fig. 8 - (a) Pressure contour on vehicle body surface without FD; (b) streamline superimposed with pressure contour surrounding the vehicle body without fd; (c) streamline superimposed with velocity contour on the truck without FD

3.3 Velocity Streamline

Streamlines are known for their instantaneously tangent to the velocity vector of flow. The streamlines were acquired on a plane placed horizontally across the truck which originates from the plane’s edge. The comparison of velocity streamlines of all models which use different shapes of FDs is shown in Fig. 9 (a) – (g). Fig. 9 (a) shows high-velocity streamlines in front of the container due to flow separation. The high pressure mainly occurs at the frontal area of the cab and container. The air even gets trapped in between the cab and container which may cause swirled airflow in that region which may result in a high-pressure region particularly in that area as shown in Fig. 9 (b).

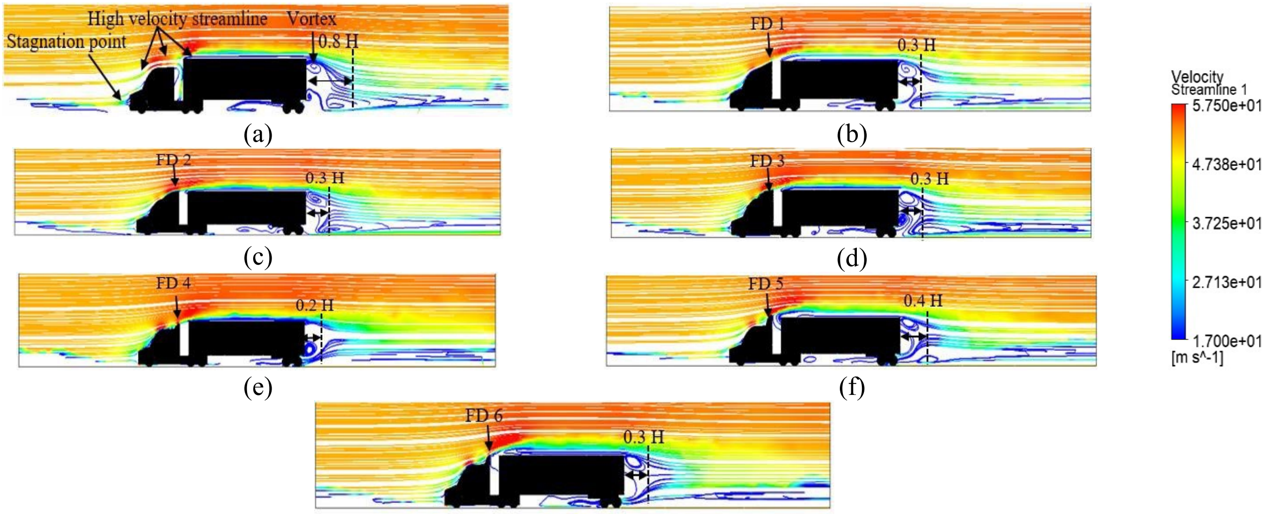


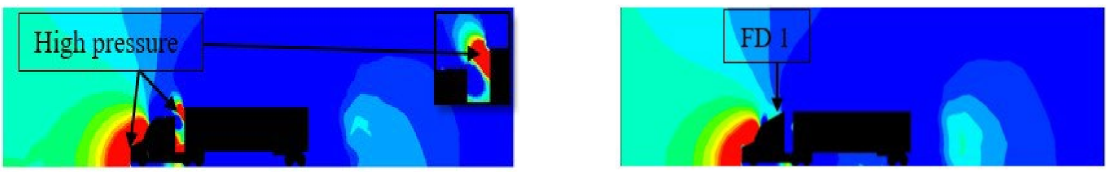
Fig. 9 - Side view of velocity streamlines on the models mounted with different FD's

The flow separation on the edges of the foreword of the trucks can be reduced by attaching FDs, which also greatly decreases the drag of the vehicle. As seen in Fig. 9, the air flowing past the cab of the truck can affect the bare front surface of the container of the truck. The air that affects the truck above the cab is splitting up, some flowing over the upper edge and around the edge of the top part of the container and some seeping down the gap where it flows under the lower part of the container and throughout the bottom sides. It can be seen that there are regions of high and low-velocity streamlines. The flow starts to separate at the frontal area of the truck and as soon as it flow's constantly above the cab, vortices regions were formed in between the cab and the container. The vortices which form below the truck's container is due to the rear front area of the truck which causes a flow separation and resulting in the formation of vortices and causing a low-pressure region in that area. Hence, resulting in downforce as the pressure above the truck is higher compared to underneath. The flow then separates at the rear end of the truck whereby vortices region is formed.

The vortex formed for each model differs as each FD's shapes hold different characteristics in terms of design and placement. As shown in Fig. 9 (b), (c) and (d) a quite similar vortex formation can be seen which is 0.3 H. The FD 4 has the shortest vortex formation which holds the length of 0.3 H compared to the truck without FD which is 0.8 H. Meanwhile, FD 5 has the largest vortex formation of 0.4 H. Besides, FD 5 and 6 has a high velocity starting from the frontal area of FD which is due to high-end design resulting in a similar vortex formation which is larger i.e. 0.4 H and 0.3 H respectively. FD 4, 5 and 6 acquires different flow pattern at the frontal area of the truck due to the thin structure of FD. Due to this, a low-velocity region can be seen near FD as a result of a direct hit of flow on the FD which causes separation to occur.

3.4 Surrounding Pressure Contour

As shown in Fig. 10, a high-pressure zone forms at the frontal portion of the truck model due to a strong effect of the incoming flow. Due to the high velocity of flow over the truck, a low-pressure zone can be seen on the top of the truck's cab. Somewhere at a truck rear end container has a low-pressure zone which means that the flow rate is high in that zone.



(a) (b)

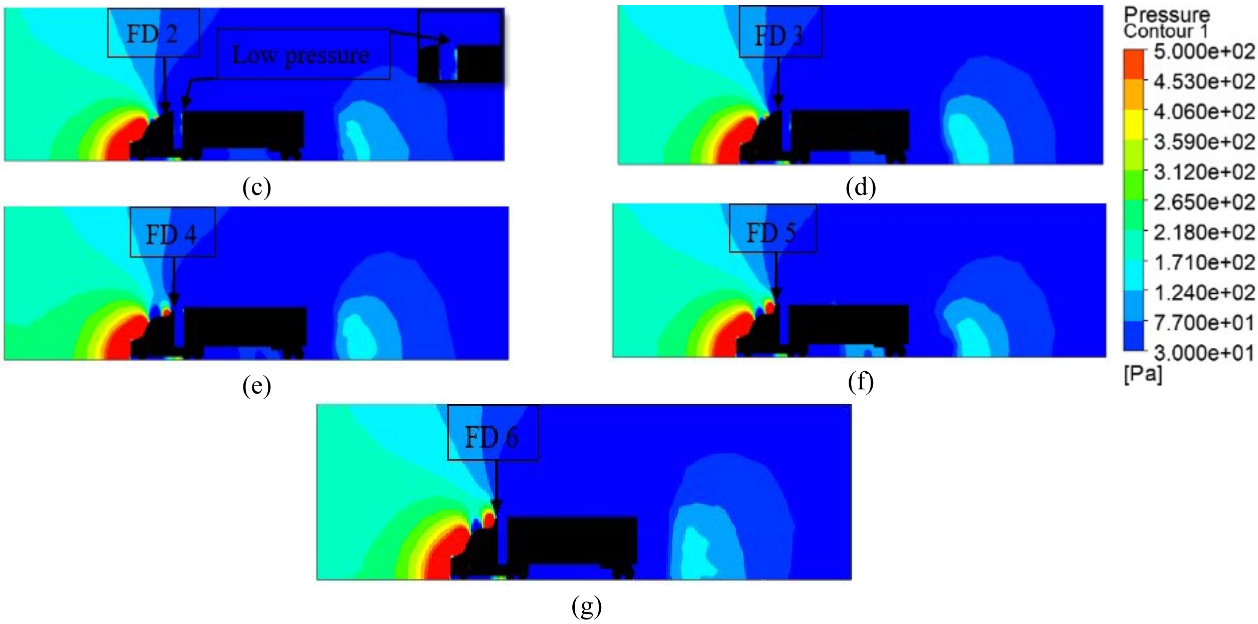
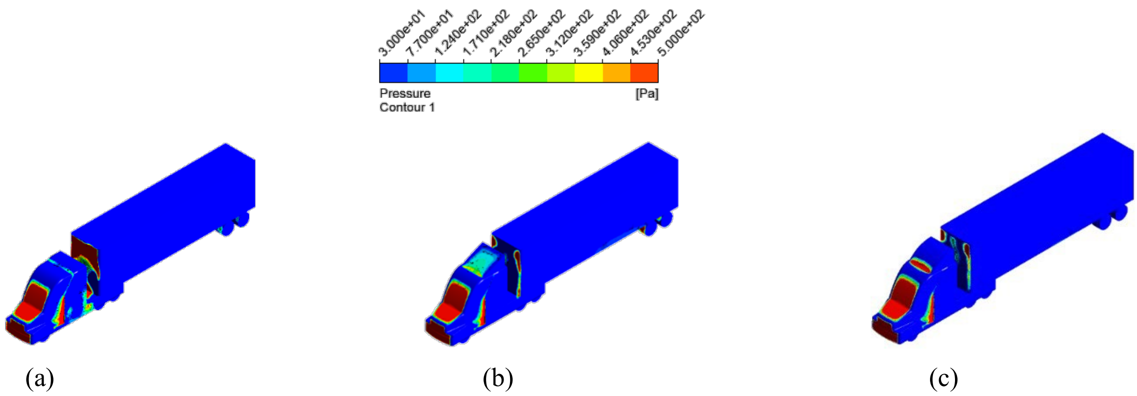


Fig. 10 - Side view of pressure contour on the vehicle surroundings for models mounted with different FD's

From Fig. 10 it can be seen that the effectiveness of cab mounted with FD has a better and uniform pressure distribution throughout the truck's surface which can be seen on the truck's surface and also surrounding of the truck with a frontal deflector. FD 4, FD 5 and FD 6 as shown in Fig. 10 (e) – (g) marks a second large high-pressure area which is near the surface of FD where this is due to the FD's shape blocking the flow structures movement.

3.5 Surrounding Pressure Contour

The pressure exerted on the truck differs due to the different shapes of frontal deflectors. As can be seen in Fig. 11, the main area which is the frontal area gets hit directly by high pressure compared to other areas. The red region on the frontal area of the truck is the stagnation point where it causes a high-pressure region. Starting from the top region, the pressure gets much lower as the flow separates which is represented by the blue pressure region. There is a slight pressure exerted on the side surface of the truck due to the separation towards the side truck structure. On the top surface of the container's frontal area of the original model truck (without FD) as shown in Fig. 11 (a) higher pressures at the front forward edge are measured. This phenomenon can be seen in Fig. 11 (a) as well, which is the pressure at the gap between cab and container. There is a slight pressure distribution that indicates a low-pressure area on the sides of the truck which is due to air that curves around. This can be theoretically proven through Bernoulli's equation where when the airspeed increases, the pressure goes down. Hence, low-pressure zones are formed. The lesser the pressure area formed in front of the container, the lesser the value of C_d. The pressure area in front of the container rapidly decreases as soon as an FD is placed on top of the cab.



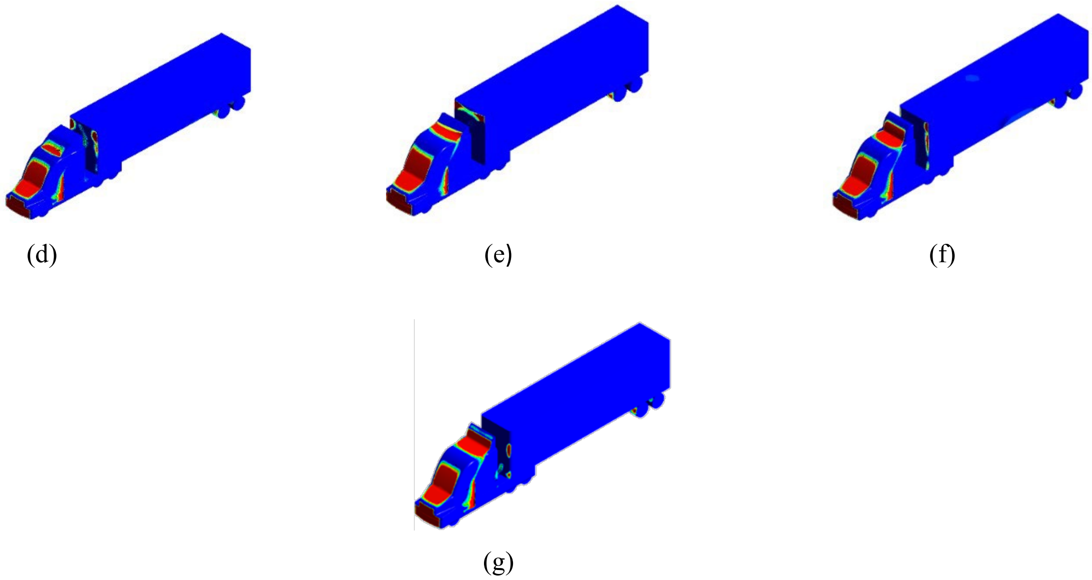


Fig. 11 - Isometric view of pressure contour on the truck surface for models mounted with different FD's

4. Conclusion

A detailed three-dimensional CFD analysis was conducted on a simplified one-eighth-scaled model of a generic truck at a specific speed. The research was analysed based on the quantitative effect i.e. coefficient of drag and the qualitative effects i.e. pressure contour and velocity streamlines on aerodynamics. This study focused on the effects of frontal deflectors on aerodynamics by incorporating an enhancement component. As for the main findings, this study on frontal deflectors (FD) has proven that the optimal placement and the best configuration of FD will aerodynamically minimise the drag coefficient (C_d) of trucks. Different shapes and locations of FD mounted on the trucks has a significant impact on the aerodynamics of the vehicle. Adding an FD lowers the pressure by reducing the high-pressure area at the rear front of the container. By applying the conceptualization of adding an FD to the truck's framework, the drag force acting on heavy vehicles has been significantly decreased. This study helps in improving and developing the aerodynamic aspects in trucks which could favour both the user and industry under certain aspects i.e. high performance, vehicles stability and fuel consumption. In this CFD analysis, the drag coefficient (C_d) can be reduced by as much as 32.2% when FD 4 is introduced. In the future, the topic of this study can be enhanced in terms of the parametric study on the FD4 configurations such as the length and angle of the FD in order to further optimize the aerodynamic performance of the trucks. In conclusion, by assessing the effect of each parameter as mentioned, this analysis can be further extended.

Acknowledgement

The author would also like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for providing the necessary research facility for this study.

References

[1] H. Chowdhury, R. Juwono, M. Zaid, R. Islam, B. Loganathan, and F. Alam, "An experimental study on of the effect of various deflectors used for light trucks in Indian subcontinent," *Energy Procedia*, vol. 160, no. February, pp. 34–39, 2019, doi: 10.1016/j.egypro.2019.02.115.

[2] P. Das, M. Tsubokura, T. Matsuuki, N. Oshima, and K. Kitoh, "Large Eddy Simulation of the flow-field around a full-scale heavy-duty truck," *Procedia Eng.*, vol. 56, pp. 521–530, 2013, doi: 10.1016/j.proeng.2013.03.155.

[3] R. M. Wood and S. X. S. Bauer, "Simple and low-cost aerodynamic drag reduction devices for tractor-trailer trucks," *SAE Int. Truck Bus Meet. Exhib.*, pp. 1–18, 2003, [Online]. Available: http://dx.doi.org/10.4271/2003-01-3377.

[4] X. Wang, X. Hu, L. Liao, and T. Li, "Numerical simulation to investigate influence of additional devices on aerodynamic drag for heavy-duty commercial truck," *Appl. Mech. Mater.*, vol. 209–211, pp. 2089–2093, 2012, doi: 10.4028/www.scientific.net/AMM.209-211.2089.

[5] S. A. Bhat, "Aerodynamic analysis and optimization of wind deflector in a Commercial load transport vehicle," *Int. J. Innov. Res. Adv. Eng.*, vol. 4, no. 04, pp. 86–89, 2017.

[6] S. Hetawal, M. Gophane, B. K. Ajay, and Y. Mukkamala, "Aerodynamic study of formula SAE car," *Procedia Eng.*, vol. 97, pp. 1198–1207, 2014, doi: 10.1016/j.proeng.2014.12.398.

[7] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. M. Tahir, "Review of research on vehicles aerodynamic drag reduction methods," *Int. J. Mech. Mechatronics Eng.*, vol. 14, no. 2, pp. 35–47, 2014.
[8] M. N. . Kamal *et al.*, "Effect of Crosswinds on Aerodynamic Characteristics of a Generic Train Model Using ANSYS," *J. Ind. Eng. Innov.*, vol. 2, no. 1, pp. 1–12, 2020.
[9] I. A. Ishak, M. S. Mat Ali, M. F. Mohd Yakub, and S. A. Z. Shaikh Salim, "Effect of crosswinds on aerodynamic characteristics around a generic train model," *Int. J. Rail Transp.*, vol. 7, no. 1, pp. 23–54, 2019, doi: 10.1080/23248378.2018.1424573.
[10] G. M. Le Good and K. P. Garry, "On the use of reference models in automotive aerodynamics," *SAE Tech. Pap.*, vol. 2004, no. 724, 2004, doi: 10.4271/2004-01-1308.
[11] M. Rashaduddin and A. Waheedullah, "A Study on Airflow over a Car," *Int. J. Sci. Eng. Technol.*, vol. 5, no. 3, pp. 21–28, 2017, doi: 10.2348/ijset0517021.
[12] R. A. Drollinger, "Heavy duty truck aerodynamics," *SAE Tech. Pap.*, 1987, doi: 10.4271/870001.
[13] W. H. Hucho and G. Sovran, "Aerodynamics of road vehicles," *Annu. Rev. Fluid Mech.*, no. 25, pp. 485–537, 1993.
[14] M. Kumar, A. Dubey, and A. Arun Jadhav, "Effect of vortex generators on aerodynamic characteristics of a car," *Adv. Mater. Res.*, vol. 418–420, no. November 2018, pp. 1873–1877, 2012, doi: 10.4028/www.scientific.net/AMR.418-420.1873.
[15] C. Ahmedabad, "CFD Analysis of Aerodynamic and Aeroacoustics in Truck Trailer body," *Int. J. Res. Eng. Sci. Manag.*, vol. 1, no. 11, pp. 119–122, 2019.
[16] I. A. Ishak, M. S. M. Ali, and S. A. Z. Shaikh Salim, "Mesh size refining for a simulation of flow around a generic train model," *Wind Struct. An Int. J.*, vol. 24, no. 3, 2017, doi: 10.12989/was.2017.24.3.223.
[17] B. Storms, D. Satran, J. Heineck, and S. Walker, "A Summary of the Experimental Results for a Generic Tractor-Trailer in the Ames Research Center 7- by 10-Foot and 12-Foot Wind Tunnels," *NASA*, no. July, 2006.
[18] R. C. Mccallen, K. Salari, J. Ortega, and P. Castellucci, "DOE Project on Heavy Vehicle Aerodynamic Drag," *FY 2006 Annu. R ep. Heavy Veh. Aerodyn.*, no. January, 2007, doi: 10.2172/1036846.
[19] W. D. Pointer, T. Sofu, and D. Weber, "Commercial CFD Code Validation for Simulation of Heavy-Vehicle External Aerodynamics," *4th ASME_JSME Jt. Fluids Eng. Conf.*, pp. 1–6, 2018.