

Design Analysis and Shape Optimization of Cabin to reduce Aerodynamic Drag in a Semi-Truck by using Computational Fluid Dynamics

Vignesh Banjan¹, Dattaraj Raikar²

¹Undergraduate, University of Wolverhampton

²Undergraduate, University of Wolverhampton

Abstract - Over recent years, commercial vehicles such as Trucks or Semi-trucks are in heavy demand as the need to transport goods from one place to another has increased. The commercial vehicles market is worried as fuel prices are rapidly increasing, they fear that they will have to increase commercial vehicles' traveling speed and stability, and one of the ways to achieve this is by decreasing the aerodynamic drag experienced by commercial vehicles. This project stresses the importance of the exterior design of a Semi-truck's Cabin by computationally analyzing the exterior aerodynamic drag experienced by a Semi-truck's Box-type Cabin via SST k-omega turbulence model and proposing an improved design of the same Cabin that has potentially lower drag force, capable of replacing the conventional box-type Cabin in a Semi-truck.

Key Words: Exterior Aerodynamics, Computational Fluid Dynamics, Semi-truck, Truck Cabin, Aerodynamic Drag Force, Truck Cabin Aerodynamic.

NOMENCLATURE

F_D – Drag Force

C_D – Drag Coefficient

ρ – Fluid Density

v – Object Velocity

A – Cross-sectional Area

1. INTRODUCTION

Box-type Cabin Semi-truck

The Box-type cabin of a semi-truck is known by many names such as Cab-over Engine, Forward Control, flat face, or flat nose, etc. This type of design was largely preferred by European and Asian truck manufacturers. Unlike the American cab design that has its engine placed in front of the cab, the forward control design has its cab on top of the engine. This type of cabin has a box shape design having a vertical frontal body which creates difficulty for the vehicle to speed up or be very stable while making turns at moderate speed. The box-type Cabin body style is quickly losing its value as the demands for quicker transportation and fuel prices are rapidly increasing [1].

Aim and Objectives

This project aims to design a CAD (Computer-Aided Design) model of a Semi-truck's Cabin had a lower drag force compared to a Box-type Cabin. A CAD model of a Box-type Cabin will be further improved by reconstructing based on exterior aerodynamics principles, strictly focused on the aim of reducing the aerodynamic drag force.

Considering the aim, below mentioned are the objectives to be achieved in this project: -



Fig - 1: Image of a Cab-over Engine Semi-truck

- o demonstrate extensive research on Semi-truck's Cabin and identify its influence on Aerodynamics of a Semi-truck.
- To identify research gap in a Box-type Cabin of Semi-truck concerning its exterior shape and explore areas of improvement.
 - To undertake modification concerning the exterior shape of a Box-type Cabin and construct an improved Cabin design.
 - To achieve significantly less aerodynamic drag force in the improved CAD model of the Cabin compared to the initial design of the Box-type Cabin.
 - To compare the results and highlight the difference between the initial design of a Semi-truck's Box-type Cabin and the improved design of the same Cabin.

3. Literature Review

Study of Aerodynamics of a Semi-truck

In this paper, the authors [2] have given a detailed overview of the aerodynamics of heavy vehicles and have also discussed the significance of aerodynamic flow-control devices which are utilized for reducing aerodynamic drag. In this paper, the author [3] has carried out an experimental analysis of an articulated model of a semi-truck to identify the effects of Cabin shape on the aerodynamic drag of a semi-truck. The authors [4] of this research paper have described the effects of aerodynamic devices on the external aerodynamics of a semi-truck. Various aerodynamic devices have been used, such as cabin roof fairing, side skirts, and gap fillers or flaps which are used to cover the gap between the cabin and the trailer of the semi-truck. The authors have briefly analyzed the results obtained and have highlighted the maximum drag reduction is obtained when a maximum number of aerodynamic devices are attached to the semi-truck. The authors [5] have discussed the two major styles of Cabin Design of semi-trucks i.e., Cab Over Engine Design and American Cabs Design. The authors have identified that COE configuration semi-trucks are majorly used in Europe and Conventional configuration semi-trucks are majorly used in North America. This paper focuses on the differences in Cabin Design and their effect on the Aerodynamics of semi-truck. The authors have also shared the future designs of Cab which can be useful considering the Aerodynamics of semi-truck, the designs such as the short nose and convoy driving, etc. have been explained in this paper.

Effects of Aerodynamic devices on Aerodynamics of a Semi-truck

In this paper, the authors [6] have stressed the under-body flow of a commercial vehicle and its significance towards aerodynamic drag. The authors have demonstrated aerodynamic drag results achieved using two variants of side skirts in a semi-truck. The authors [7] have used coarse Large Eddy Simulation to numerically investigate the exterior aerodynamics of a commercial vehicle consisting of a Cab-roof Fairing (CRF). The authors have also tested a 1/8 scaled-down vehicle model in a wind tunnel and achieved a 15% drag reduction. The authors have compared the results achieved in a CRF-equipped vehicle model and a vehicle model without CRF. The author [8] has used aerodynamic devices in a commercial vehicle model to interpret their effects on the aerodynamics of the vehicle. The author has got better results in two types of combination of aerodynamic devices, first one, deflector and frame extension, and secondly, deflector and vortex stabilizer. In both cases, the author has achieved results of approximately a 29% reduction in aerodynamic drag. In this paper, the authors [9], to analyze drag reduction, have utilized two aerodynamic devices, side skirts, and boat tail to the trailer. They have performed wind tunnel tests with a 1:14 scaled vehicle model and have also conducted road tests with full-scale prototypes that they had created. Wind tunnel test of side skirts with a yaw angle of 8° resulted in a 17% reduction in aerodynamic drag. Wind tunnel test of a combination of both the aerodynamic devices i.e., side skirts and boat tail resulted in a 20% reduction in aerodynamic drag. The author [10] has analyzed the external aerodynamics of a semi-truck tanker, attached with three types of aerodynamic devices – boattail, undercarriage skirt, and nose cone. Initially, the author has designed and analyzed a generic tanker truck to work on areas where improvement is needed. Then the designed model under four conditions – with boat-tail; with nose cone; with undercarriage side skirt and with all of the aerodynamic devices attached. For the model attached with a boat tail, around 4% drag reduction was obtained, for the model attached with a nose cone, around 3% drag reduction was obtained, for the model attached with an undercarriage skirt, around 7% drag reduction was obtained and for the model attached with all the aerodynamic devices, around 23% drag reduction has been obtained. These results are analysed by the author in this paper. The authors [11] have used aerodynamic devices in a semi-truck to achieve aerodynamic drag reduction, these devices are situated at the roof of the vehicle. Via Computational Fluid Dynamics (CFD) the authors have analyzed the effect of the aerofoil devices used on pitch and heave factors of the cabin of the vehicle. Three actuators/aerodynamic devices were used, and, in the paper, results are presented of only one of them. The results discuss amelioration achieved by just one controlled actuator and highlight the importance of its dimensions, control movement, and situation. This research paper is a detailed study of two drag reduction techniques that can be implemented in a semi-truck. Initially, the authors [12] have designed a simple model of a semi-truck to study the external aerodynamics of a semi-truck by using Computational Fluid Dynamics. The authors have then optimized the design of the semi-truck, related to the cabin and rear edges of the trailer. The optimization was performed at

various angles and the values were later compared and evaluated. The optimized design was then analyzed and compared with the simple design of the semi-truck which they created initially. The authors have obtained around 24% drag reduction in a semi-truck.

4. Proposed Methodology

To be more precise with the research area the trailer and tires, as well as the tire area, have been neglected and only the frontal exterior design of a Box-type Cabin has been considered. An initial design of the Forward Control Cabin of semi-truck was created using Computer-Aided Designing (CAD) software and was computationally analyzed via Ansys software. The initial CAD model was analyzed to obtain aerodynamic drag results for that particular model. The same CAD model was reconstructed considering aerodynamic principles to achieve significantly less aerodynamic drag compared to the initial CAD model. The results obtained by both the CAD models were compared to identify a potential improvement to the Box-type Cabin shape for drag reduction.

Throughout the project various software were used to complete the project: -

Table- 1: List of Software used for completion of the project

Purpose	Software
Designing	SolidWorks
Analysis	Ansys
Research Papers/Files Management	Mendeley
Project Tasks Planner	Microsoft Excel
Thesis Writing	Microsoft Word

1.1 Software Design and Analysis of Initial Box-Type Cabin

1.1.1 Initial Design

Initially, a Box-type Cabin was designed without considering the tires and the tire areas so that it doesn't complicate the research area. The geometry of the designed CAD model presented below (Figure 2) was kept simple by omitting all the aesthetic looks, side mirrors, interior design of the cabin, and unnecessary fillets as the main focus was to analyze exterior aerodynamics of the Cabin and ways to reduce aerodynamic drag. Standard dimension limits have been preferred while designing the model i.e., the height of the box-cabin is around 3 meters, the length is about 2.5 meters and the width is around 2.7 meters.

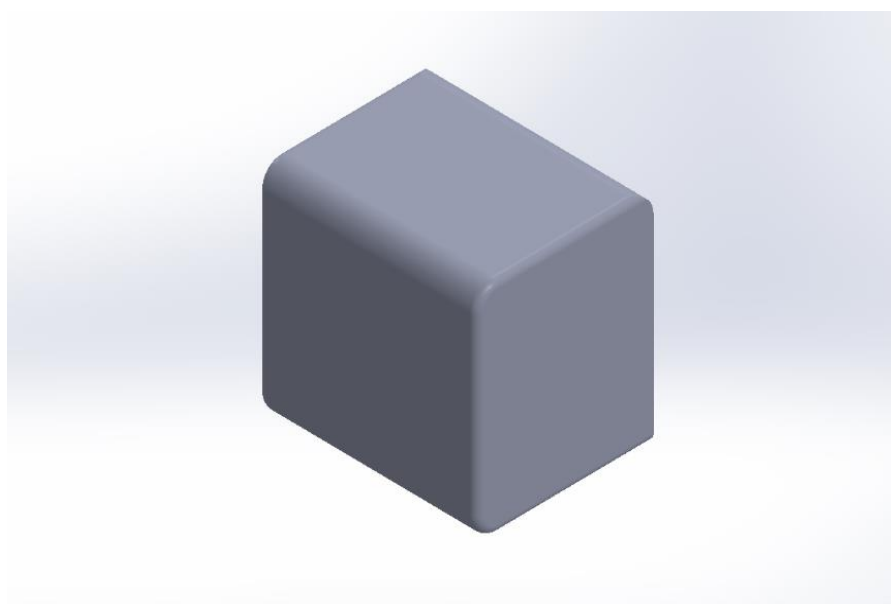


Fig - 2: Isometric view of Initial Design of Box-type Cabin of a Semi-truck

1.1.2 Analysis Setup

A flow domain was created to perform fluid flow analysis on the initial CAD model in Ansys software. Practically, the flow domain is known as Wind Tunnel where the product is kept inside and is subjected to a fluid to investigate its aerodynamics. Dimensions of the flow domain used in the analysis from the front – 5 times the length of the product, 20 times the length of the product from the rear, 5 times the length of the product from sideways, and 5 times the length of the product from the above direction (Figure 3). A ground clearance of 200 mm was considered.

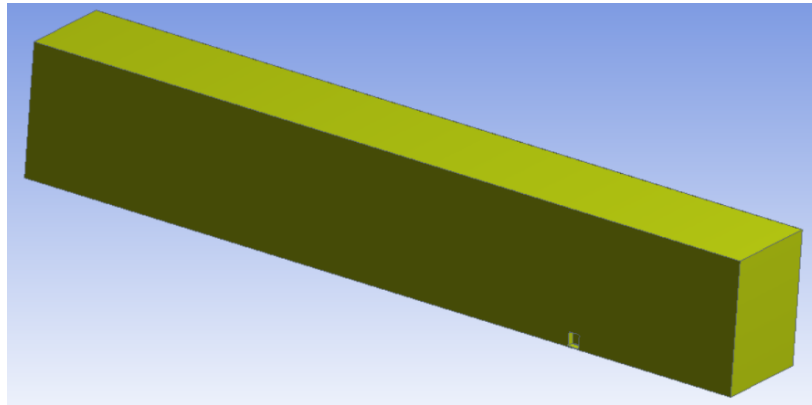


Fig - 3: A general view of Flow Domain after applying Symmetry

The symmetry of the entire geometry was taken for the analysis as it reduces the solving time by 50% and does not affect the overall results to a greater extent. The CAD model of the cabin was suppressed to create a void inside the flow domain. The entire geometry was named via the named selection operation to predefine the boundary conditions which automatically get identified by the solver tool of the Ansys software. The front face was named as Inlet through which the fluid enters, the rear face was named as an outlet as an exit point for the fluid, the bottom face was named as road – a boundary condition that moves during the simulation like a road based on the velocity and direction input provided in the solver. The symmetry side of the face was named as symmetry face and the remaining two faces were named as free slip walls so that the friction of the walls does not affect the results.

Figure 4 represents the **material properties** of the fluid that will be subjected to the CAD model. Constant Density and Viscosity of the air have been considered throughout the simulation as it is a steady transient analysis.

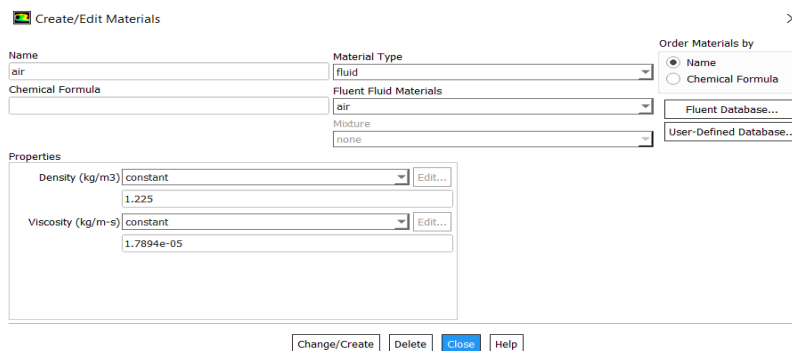


Fig - 4: Material properties of Fluid (Air) subjected to the CAD model

Aluminum material with constant density has been considered for the flow domain or the testing area. Figure 5 represents the material properties of the chosen material.

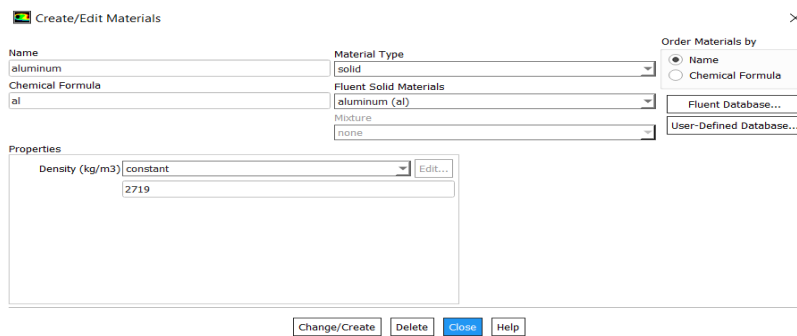


Fig - 5: Material properties of Flow Domain or Testing Section

Mesh is the process of breaking down the geometry into smaller elements and solving each of the elements. The results of all the elements are analyzed and put together to draft a final result. For engineers to manually perform this process is time-consuming and even a small human error while solving will give improper results. Therefore, this process is carried out by software that automatically breaks down the geometry into multiple smaller elements and solves each of the elements based on the input provided by the engineer to the software. In below (Figure 6) a zoomed-in view of the mesh created by the software around the cabin and illustrates that the elements in the void are precisely broken down into much smaller elements and the mesh around the contact region between the void and the flow domain is more uniformly created so that the when the cabin meets the fluid the results obtained will be more precise. The total number of nodes and elements are 213006 and 1142162 respectively.

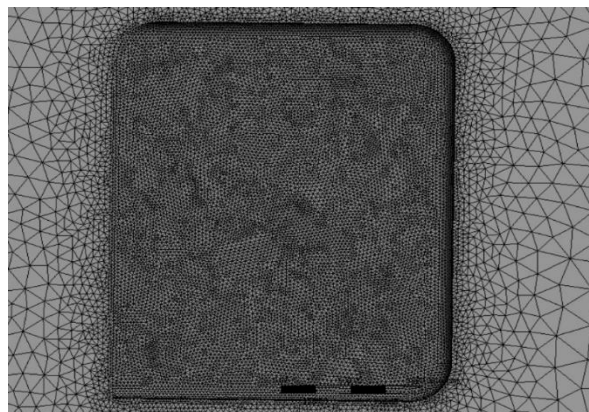
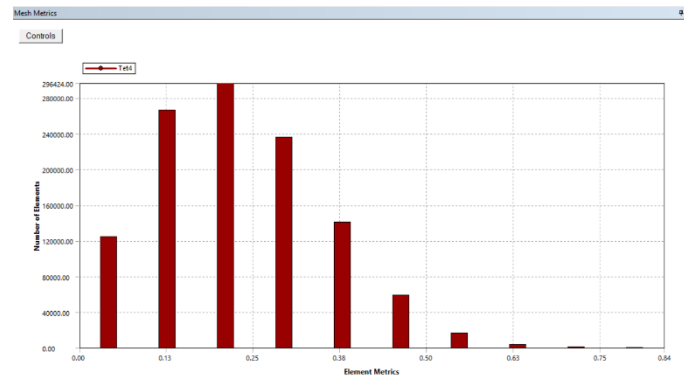


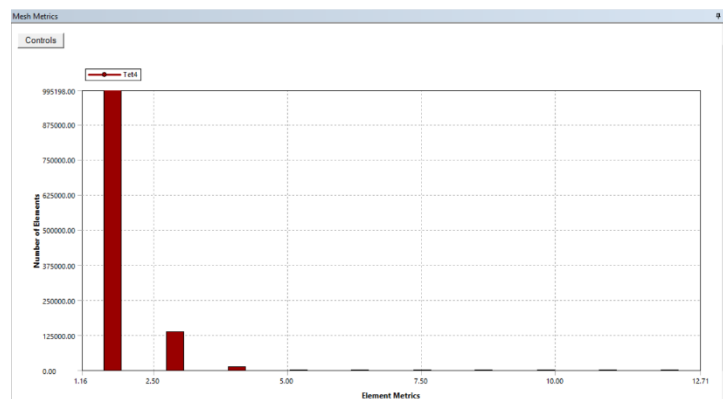
Fig - 6: Zoomed-in view of generated mesh around the initial cabin design

Mesh quality defines the software's ability to break down the geometry into smaller elements precisely so that proper results are achieved. The mesh quality is depended on the input provided to the software and also the nature of the geometry. Complicated geometry or geometry with a high number of fillets, holes, gaps, etc. affects the mesh to a greater extent which in turn affects the results obtained. Therefore, the simple geometry of the cabin was considered as the aim stresses the importance of the illustration of the shape and how it can be improved aerodynamically for the lower drag of the semi-truck. The mesh quality can be checked in the software so that the input can be altered several times to achieve the best mesh of the geometry. In below (Graph 1) the skewness quality check of the generated mesh is presented.



Graph - 1: Skewness quality of the generated mesh of initial cabin design

The above graph illustrates the skewness quality of the mesh generated. Skewness defines the property of the cell concerning its shape. An ideal cell or an equilateral cell has a skewness value of 0 and the worst shape of the cell has a skewness value of 1. The mesh generated has an average skewness value of 0.2. There are various parameters to evaluate mesh quality, skewness is one of the important parameters as the shape quality of each cell affects the results directly and even one of the cells is distorted then the result for that particular cell will be far more different from the result obtained for rest of the cells. Similarly, the aspect ratio is another important mesh quality parameter as it tells the ratio of the longest edge length of the cell to the shortest edge length of the cell. This ratio defines the shape of the cell, an aspect ratio of 1 is known as an ideal cell and an aspect ratio of more than 5 indicates that the cell has been stretched more than the limit and the results for that cell will differ highly from the results for rest of the cells. Below (Graph 2) illustrates the aspect ratio quality check for the generated mesh.



Graph - 2: Aspect Ratio of the generated mesh of initial cabin design

The generated mesh has an average aspect ratio of 1.8, a minimum aspect ratio of 1.1, and a maximum aspect ratio of 12. Though the maximum aspect ratio crosses the limit the number of cells with a maximum aspect ratio is very less with most of the cells having an aspect ratio around 1-2.

The **Turbulence model** for Computational Fluid Dynamics (CFD) Analysis is the two-equation Shear Stress Transport k-omega (k- ω) model. This turbulence model is one of the most commonly used models for the study or analysis of fluid flow around an object. In this turbulence model, “k” stands for turbulent kinetic energy and ‘omega’ or ‘ω’ stands for specific turbulent dissipation rate.

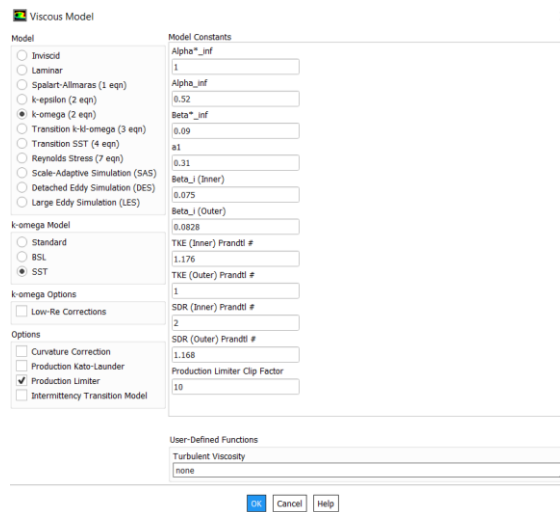


Fig - 7: An overview of SST k-omega turbulence model settings used for CFD Analysis

The Ansys Fluent Solver identified the **Boundary Conditions** which was predefined while setting up the geometry.

- **Free Slip Walls:** Shear condition – Specified Shear; Roughness model – Standard; Roughness Constant – 0
- **Inlet (Velocity):** Constant Velocity – 108 km/hr; Gauge Pressure – 0; Turbulent Intensity – 5% and Turbulent Viscosity Ratio – 10
- **Outlet (Pressure):** Gauge Pressure – 0; Backflow Turbulent Intensity – 5% and Backflow Turbulent Viscosity Ratio – 10
- **Road:** Moving Wall; Constant Velocity – 108 km/hr; Shear Condition – No-Slip; Roughness model – Standard
- **Semi-truck Cabin:** Stationary Wall; Shear Condition – No-Slip; Roughness model – Standard

Other boundary conditions include symmetry face and interior flow domain.

The solution Method used in CFD analysis is Second-order Pseudo Transient Method with coupled Pressure-Velocity.

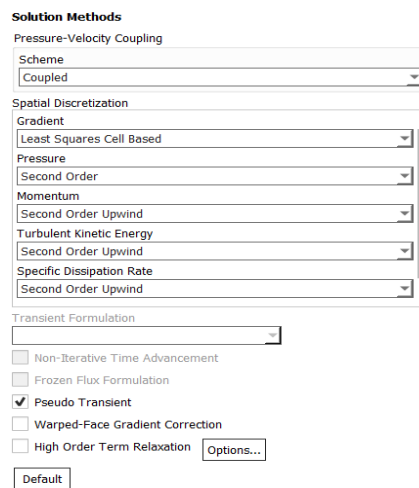


Fig - 8: An overview of Solution Method settings used for CFD Analysis

1.2 Software Design and Analysis of Final Box-Type Cabin

1.2.1 Final Design

By analyzing the results of the initial design, the same CAD model was redesigned considering the aerodynamic principles to achieve less aerodynamic drag force compared to the initial design. The vertical front of the initial design was given a curvature to make it more streamlined so that fluid could easily flow around the body which would create less aerodynamic drag force.

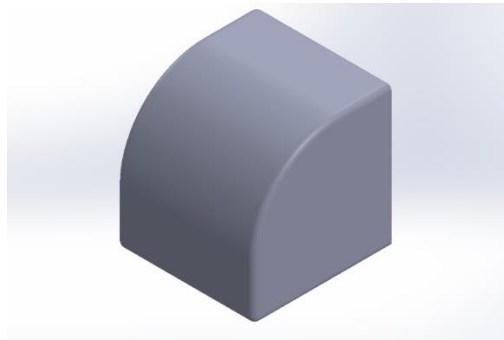


Fig - 9: Isometric view of Final Design of Box-type Cabin of a Semi-truck

1.2.2 Analysis Setup

The analysis setup for the final design of the Cabin CAD model is the same as that of the initial designed one. From setting up the geometry to setting up the solver the process followed is the same as that of the initial design of the Cabin CAD model.

However, due to a change in geometry of the CAD model the results of meshing were changed but not to a greater extent. In below (Figure 5) a zoomed-in view of the mesh created by the software around the final design of the cabin and illustrates that the elements in the void are precisely broken down into much smaller elements and the mesh around the contact region between the void and the flow domain is more uniformly created so that the when the cabin encounters the fluid the results obtained will be more precise. The total number of nodes and elements are 152274 and 810296 respectively.

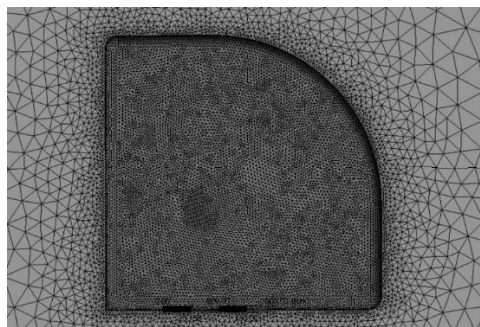
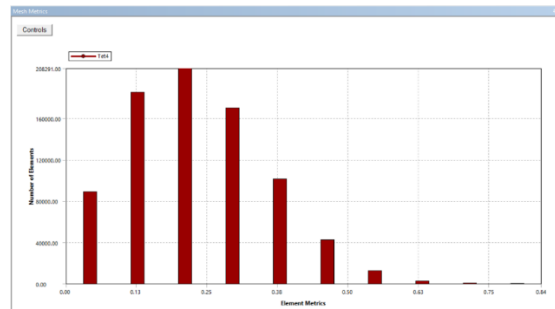


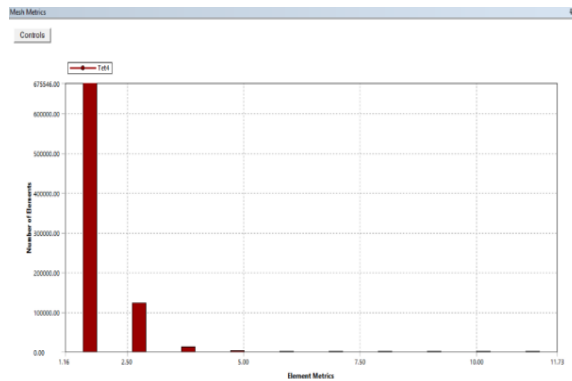
Fig - 10: Zoomed-in view of generated mesh around the final cabin design

In below (Graph 3) the skewness quality check of the generated mesh is presented. The graph illustrates the skewness quality of the mesh generated. Skewness defines the property of the cell concerning its shape. An ideal cell or an equilateral cell has a skewness value of 0 and the worst shape of the cell has a skewness value of 1. The mesh generated has an average skewness value of 0.2.



Graph - 4: Skewness quality of the generated mesh of final cabin design

Below (Graph 4) illustrates the aspect ratio quality check for the generated mesh. The generated mesh has an average aspect ratio of 1.8, a minimum aspect ratio of 1.1, and a maximum aspect ratio of 11. Though the maximum aspect ratio crosses the limit the number of cells with a maximum aspect ratio is very less with most of the cells having an aspect ratio around 1-2.



Graph - 3: Aspect Ratio of the generated mesh of final cabin design

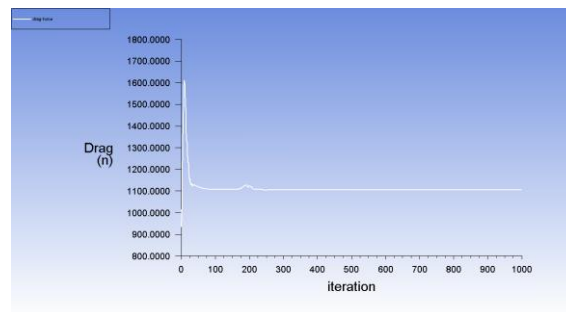
5. RESULTS AND DISCUSSION

5.1 Aerodynamic Drag, Convergence and Mathematical Evaluation

After setting up the Computational Fluid Dynamics (CFD) Analysis simulation for the initial design of the box-type cabin the calculation was performed under 1000 iterations to obtain aerodynamic drag force and drag coefficient.

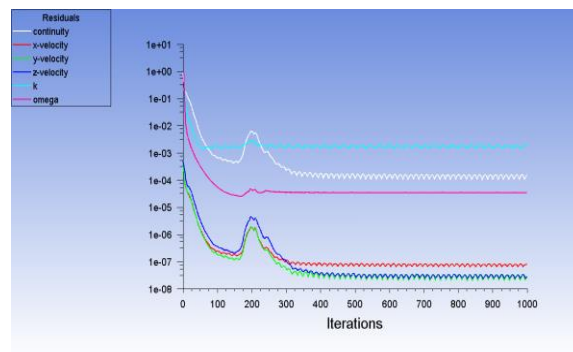
The initial design of the Box-type Cabin

The **drag force** obtained for the initial design is **1135 Newtons** and the drag coefficient is 0.41. The below (Graph 5) illustrates the fluid and the CAD model interaction also considering the movement of the road. After around 30 iterations the graph can be seen stable until 170 iterations, this means the aerodynamic drag becomes stable after fluid passes through the frontal area of the box-cabin and reaches the top flat and smooth of the cabin and shows a slight increase as it passes through the end of the cabin which creates wake causing a disturbance in aerodynamic drag.



Graph - 5: Graph of Aerodynamic Drag Force obtained by initial Cabin design

Below (Graph 6) shows that x, y, and z velocity are converged efficiently. After 1000 iterations all of the **residuals** have converged considering convergence condition of 1e-03 except turbulent kinetic energy (k). The sudden increase in the residuals after 170 iterations is because of the rear shape of the box-type cabin which doesn't have a downward flap or anything similar to control the fluid flow smoothly.



Graph - 6: Residuals' graph of initial Cabin design

Mathematical Evaluation of the results obtained for the initial design of the Box-type Cabin

Mathematically aerodynamic drag force is calculated by applying the formula given below to validate the results obtained via CFD simulation.

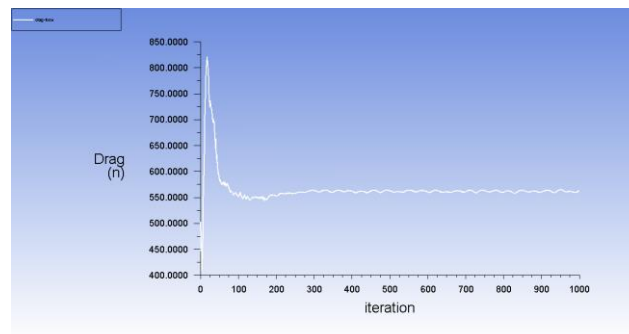
$$\text{Formula: } F_D = \left(\frac{\rho}{2}\right) \cdot v^2 \cdot C_D \cdot A$$

$$F_D = \left(\frac{1}{2}\right) \cdot 1.225 \cdot (900) \cdot 0.41 \cdot 5$$

$$\boxed{F_D = 1130 \text{ N}}$$

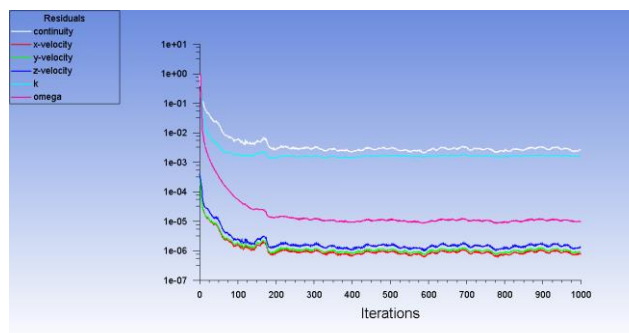
The final design of the Box-type Cabin

The **drag force** achieved obtained for the initial design is **572 Newtons** and the drag coefficient is 0.41. The below (Graph 7) illustrates the fluid and the CAD model interaction also considering the movement of the road. The unlevel nature of the graph is due to values of the scale being very less, the drag force keeps fluctuating around 572-580 Newtons.



Graph - 7: Graph of Aerodynamic Drag Force obtained by final Cabin design

Below (Graph 8) shows that x, y, and z velocity are converged efficiently. After 1000 iterations all of the **residuals** have converged considering convergence condition of 1e-03 except turbulent kinetic energy (k) and continuity.



Graph - 8: Residuals' graph of final Cabin design

Mathematical Evaluation of the results obtained for the final design of the Box-type Cabin

Mathematically aerodynamic drag force is calculated by applying the formula given below to validate the results obtained via CFD simulation.

$$F_D = \left(\frac{1}{2}\right) \cdot \rho \cdot (v^2) \cdot C_D \cdot A$$

$$F_D = \left(\frac{1}{2}\right) \cdot 1.225 \cdot (900) \cdot 0.21 \cdot 5$$

$$F_D = 578 \text{ N}$$

5.2 Post-Processing

Post-processing is a standard process of displaying the results in illustrative images. In this project, results of Pressure and Kinetic Turbulent Energy in form of contour plots of both CAD models have been presented and compared.

Contour Plot 1 – Pressure on the external surface of both Cabins

The below (Figure 11) is a pressure contour plot on the external surface of the initial CAD model. The frontal area can be seen full of red contour indicating the high pressure experienced by the cabin due to the flat and vertical front exterior shape of the cabin. The maximum pressure experienced by the exterior shape of the cabin is 545.5 Pascal as illustrated in the below contour in red color.

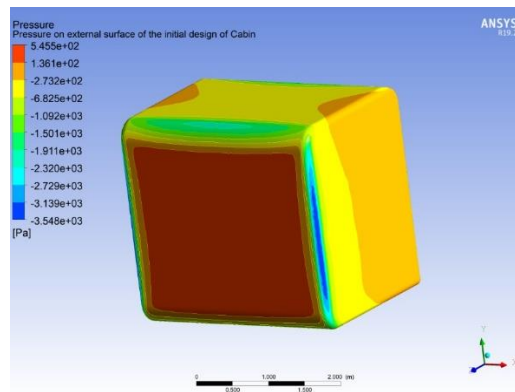


Fig - 11: Contour Plot of Pressure on the external surface of the initial Cabin design

The below (Figure 12) is a pressure contour plot on the external surface of the final CAD model. Unlike the initial CAD model, this improved CAD model has a curvature that makes the fluid flow around the CAD model smoothly. However, the below frontal portion has a flat surface hence, the pressure contour plot on that flat surface is high. Though the maximum pressure experienced by this CAD model is slightly higher than the initial CAD model but the magnitude of the area experiencing the high pressure is significantly lower than the initial design which helps in reducing aerodynamic drag.

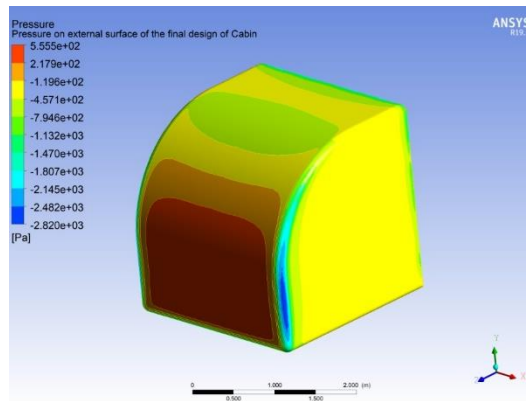


Fig - 12: Contour Plot of Pressure on the external surface of the final Cabin design

Contour Plot 2 – Pressure around both Cabins

The below (Figure 13) is a pressure contour plot projected around the initial CAD model. The frontal area around the cabin can be seen full of red contour indicating the high pressure generated around that area due to the flat and vertical front exterior shape of the cabin.



Fig - 13: Contour Plot of Pressure around the initial Cabin design

The below (Figure 14) is a pressure contour plot around the final CAD model. Unlike the initial CAD model, this improved CAD model has a curvature that makes the fluid flow around the CAD model smoothly. However, the below frontal portion has a flat surface hence, the pressure contour plot around that flat surface is high.



Fig - 14: Contour Plot of Pressure around the final Cabin design

Contour Plot 3 – Turbulence Kinetic Energy around both Cabins

The below (Figure 15) is a contour plot of Turbulence Kinetic Energy around the initial CAD model. Varying contours can be seen in the rear area indicating the high turbulence generated by the cabin. The maximum Turbulence Kinetic Energy experienced by the initial CAD model of the cabin is 64.71 square meter per second squared.

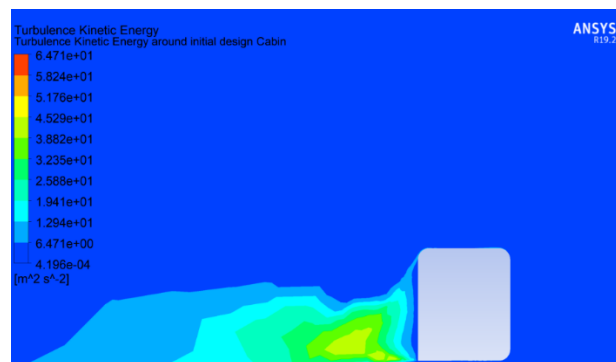


Fig - 15: Contour Plot of Turbulence Kinetic Energy around the initial Cabin design

The below (Figure 16) is a contour plot of Turbulence Kinetic Energy around the final CAD model. Varying contours can be seen in the rear area indicating the high turbulence generated by the cabin. The maximum Turbulence Kinetic Energy experienced by the final CAD model of the cabin is 48.25 square meter per second squared.

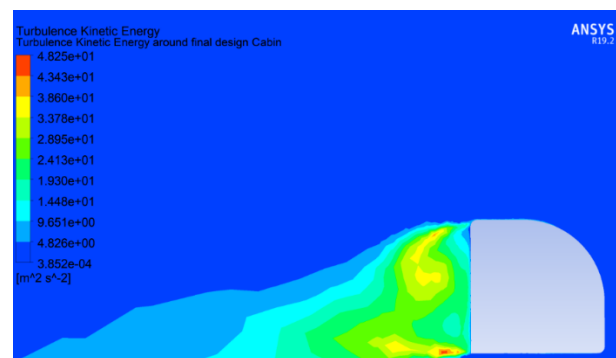


Fig - 16: Contour Plot of Turbulence Kinetic Energy around the final Cabin design

Comparing the results achieved for both cabin designs.

Table- 2: Tabulated results of both cabin designs

Results	Initial Cabin Design	Final Cabin Design
Drag Force (Newtons)	1135	572
Drag Coefficient	0.41	0.21
Minimum Pressure (Pascal)	- 3548	- 2820
Maximum Pressure (Pascal)	545.5	555.5
Minimum Turbulence Kinetic Energy (square meter per second squared)	0.0004196	0.0003852
Maximum Turbulence Kinetic Energy (square meter per second squared)	64.71	48.25

6. CONCLUSION

The automotive market's recent demands of the need for commercial vehicles' transportation to be quicker and rising fuel prices makes it essential for the commercial vehicles to design to be improvised for higher speed and stability. By achieving drag reduction is one of the most common ways to improve a vehicle's design for higher speed and stability. In this project, the aerodynamic drag force got reduced about 50%, from 1135 Newtons to 572 Newtons. From the drag reduction achieved, it can be concluded that the box-type cabin of a semi-truck or forward control semi-truck cabin can be optimized aerodynamically to achieve the required drag reduction. However, the results are independent of the contribution made by the trailer, tyres, tyre areas and side mirrors of a semi-truck, towards the aerodynamics of the vehicle. Therefore, when considering a truck itself, the results may slightly differ. Since the project focuses on the influence of the exterior shape of a truck's cabin on the aerodynamics of the vehicle it can be concluded that by changing the curvature of the frontal area aerodynamic drag can be significantly reduced. Since the project was completed using personal resources, the results obtained could be more accurate if the simulation is performed under an advanced computer.

7. FUTURE SCOPE

As the results achieved by this project demonstrates the importance of improvising a semi-truck cabin's exterior design for reducing aerodynamic drag significantly, it opens more research opportunities concerning the exterior design of a vehicle. Since the markets are demanding vehicles with high speed and stability the scope of this project is to identify a research area of development that can be essential in creating vehicles with high speed and stability. The research area highlighted by the project is less complicated compared to other research areas as it strictly depends on the exterior shape of the vehicle. The exterior shape of the Box-type Cabin was manually optimized by applying aerodynamics theoretical knowledge because personal resources had to be utilised for completion of the project, hence, the project can be more efficient if the optimization performed is automatic or optimization is performed via a Computer-Aided Engineering software by using an advanced computer.

REFERENCES

- [1] A. 4Wolf, "Cab-Over-Engine Trucks — Their Status and Advance in Design," SAE Transactions, vol. 32, pp. 571-580, 1937.
- [2] H. Choi, J. Lee and H. Park, "Aerodynamics of heavy vehicles," Annual Review of Fluid Mechanics, vol. 46, pp. 441-468, 2014.
- [3] A. Gilhaus, "The influence of cab shape on air drag of trucks," Journal of Wind Engineering and Industrial Aerodynamics,

vol. 9, no. 1-2, pp. 77-87, 1981.

- [4] H. Chowdhury, H. Moria, A. Ali, I. Khan, F. Alam and S. Watkins, "A study on aerodynamic drag of a semi-trailer truck," *Procedia Engineering*, vol. 56, pp. 201-205, 2013.
- [5] L. Hjelm and B. Bergqvist, "European truck aerodynamics - A comparison between conventional and coe truck aerodynamics and a look into future trends and possibilities," *Lecture Notes in Applied and Computational Mechanics*, vol. 41, pp. 469-477, 2009.
- [6] B. G. Hwang, S. Lee, E. J. Lee, J. J. Kim, M. Kim, D. You and S. J. Lee, "Reduction of drag in heavy vehicles with two different types of advanced side skirts," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 155, pp. 36-46, 2016.
- [7] J. J. Kim, S. Lee, M. Kim, D. You and S. J. Lee, "Salient drag reduction of a heavy vehicle using modified cab-roof fairings," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 164, pp. 138-151, 2017.
- [8] S. Hamidreza, "Aerodynamic Analysis of Drag Reduction Devices on the Simplified Body for Tractor and Trailer by Using CFD," Eastern Mediterranean University, Gazimağusa, 2016.
- [9] G. v. Raemdonck and M. v. Tooren, "Numerical and Wind Tunnel Analysis Together with Road Test of Aerodynamic Add-Ons for Trailers," *Lecture Notes in Applied and Computational Mechanics*, vol. 79, pp. 237-252, 2016.
- [10] R. Miralbes, "Analysis of some aerodynamic improvements for semi-Trailer tankers," *Proceedings of the World Congress on Engineering*, vol. 3, no. 4-6, 2012.
- [11] A. Savkoor, S. Manders and P. Riva, "Design of actively controlled aerodynamic devices for reducing pitch and heave of truck cabins," *JSAE Review*, vol. 22, no. 4, pp. 421-434, 2001.
- [12] X. Yang and Z. Ma, "Drag reduction of a truck using append devices and optimization," *Communications in Computer and Information Science*, vol. 405, pp. 332-343, 2014.