

A Review on The Effect of Aerodynamics on Car Overtaking Truck Using CFD Analysis Method

Bodhisagar J. Tayade¹, P.S. Bajaj²

¹PG Student (M.Tech. Design Eng.) Mechanical Engineering Department, SSGB College of Engineering and Technology, Bhusawal, Maharashtra

²Professor, Dept. of Mechanical Engineering, SSGB College of Engineering and Technology, Bhusawal, Maharashtra

Abstract - This report explains the rationale why mechanics play such a crucial role within the style of business trucks. It's established through the study of previous papers supported the sphere of auto mechanics however the pure mathematics of trucks affects mechanics and fuel consumption. Raising the mechanics of business tractor-trailer units involves the installation of drag reduction devices in sure regions of the truck subject to the foremost drag force. In preparation for the CFD (Computational Fluid Dynamics) study on model trucks a test suit was studied initial.

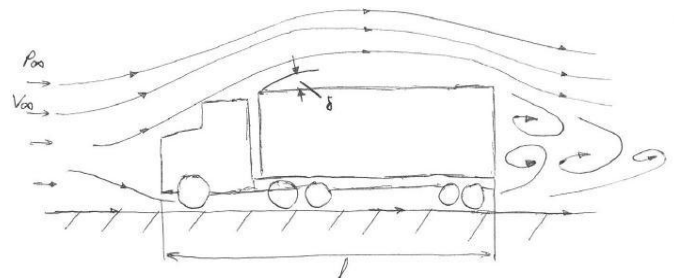


Fig. 1. The flow field around a commercial truck

Key Words: CFD Basics, Aerodynamics

1. INTRODUCTION

The external mechanics of road vehicles have a robust influence on friction resistance, fuel consumption and stability. This project can study the mechanics of semi-trailer business trucks, wherever there's sturdy mechanics interference between the tractor and trailer units, and far scope remains for improvement.

1.1 Project Objectives

The objectives of this project square measure clearly declared as follows;

- To summarize the explanations why mechanics and drag reduction may be a crucial facet of business vehicle style.
- To do a review of the present assortment of drag reduction devices utilized by the industry.
- To simulate with CFD the influence drags reduction devices wear the general coefficient of a specific model truck.

1.2 An Introduction to Mechanics

In terms of road vehicles one amongst the foremost vital physics at play is mechanics. mechanics is a huge, well matured field of science thus solely the foremost applicable aspects are going to be mentioned during this report id est, the basics. the foremost applicable aspects of mechanics relative to vehicle motion include; the properties of incompressible fluids, external flow phenomena and consistency effects.

In general, there square measure 2 flow phenomena in fluids, internal and external. Internal flow worries with motion within a confined area sort of a pipe for example. this kind of flow field doesn't relate to vehicle motion because the passing air over a vehicle isn't confined. For this reason, solely external flow phenomena are going to be mentioned.

2. LITERATURE REVIEW

In an overtaking or crossing maneuver on a highway involving two vehicles, the flow fields around the two vehicles interact generating transient aerodynamic forces that can affect car handling and stability [1]. When the relative size difference between the two vehicles is large (e.g., between a car and a truck), these forces increase on the smaller vehicle, and they increase even further under the influence of crosswinds, especially when the overtaking car is on the leeward side of the truck [2]. A vehicle is more stable when its geometric center, center of gravity, and stagnation point are all in line. Under crosswind, the air flow around the vehicle becomes asymmetric, and the stagnation point shifts towards the direction of the crosswind, affecting the stability of the vehicle [3]. As the computational power of commercially available computers has doubled approximately every two years (famously observed and predicted by Gordon Moore [4]), high quality transient computational fluid dynamics (CFD) simulations of the complex interactions between moving vehicles have not been feasible until the late 2000s. This literature review is divided into three sections that are based on the physics of the study (single vehicles, vehicle interaction, and vehicle interaction in the presence of crosswinds).

Tsubokura et al. [5] conducted full-scale simulations on a single passenger car by using a Large Eddy Simulation (LES) model that could reproduce unsteady turbulence characteristics with high accuracy. The simulation used about 38 million cells, and the resulting vortices and flow structures were visualized in detail and validated with full-scale wind tunnel experiments. The yaw angle of the vehicle was also changed, to mimic sudden crosswinds, and the change in the flow structures was investigated. The study showed the advantages of using LES to provide aerodynamic data on the different eddy structures around vehicles in comparison to conventional wind tunnel tests or Reynolds-Averaged Navier-Stokes (RANS) simulations. A later study by **Tsubokura et al.** [6] investigated the aerodynamic response to transient crosswinds of a 5 percent scale model passenger car by using LES and wind tunnel experiments validation. Unsteady and gusty crosswinds were considered in the study, showing all six components of aerodynamic forces and moments in each case.

Sterken et al. [7] used the realizable $k-\epsilon$ turbulence model with standard wall functions and 90 million cells to investigate the wake-shape behind a passenger car. The moving ground and rotating wheels were simulated. Full-scale models were used for both the simulations and the experimental studies, and there was good correlation between them in terms of drag coefficient. The wake-shape showed both similarities and differences.

Using a sliding mesh method, **Wang et al.** [8] conducted a three-dimensional CFD study on the transient aerodynamic forces occurring on a motorcycle overtaken by a truck, considering three different lateral distances between the two vehicles. Results showed the variation in forces and moments on the motorcycle during the overtaking process. However, the simulation used the Reynolds-Averaged Navier-Stokes (RANS) model and only 1.7 million computational cells. This model experienced difficulties capturing unsteady flow characteristics [5]. No experimental studies were performed to validate the results.

A study by **Al Homoud et al.** [9] conducted CFD simulations on a car overtaking a truck by using full-scale geometries. It did not mention the turbulence model used in the simulations, and there was no experimental validation. The results did not show the variation of forces acting on the car with time, as the study followed a quasi-steady approach. **Zhang et al.** [10] conducted a transient CFD study on two vehicles crossing each other by using $k-\epsilon$ turbulence model; however, it is not clear whether the geometries used were two-dimensional (2-D) or three-dimensional (3-D). There was no experimental validation for the results.

Basara et al. [12], using a hybrid RANS/LES scheme (denoted as PANS, Partially Averaged Navier-Stokes [11]) to perform CFD simulations on a car overtaking a truck, showed that the drag on the car increased to its maximum

value when the car reached the front of the truck. The result was validated by experimental studies using 40 percent scaled models of both vehicles, fixed at eight relative positions between the two models, thus approximating the unsteady process with a quasi-steady process. This increase in forces on the overtaking vehicle as it approached the front of the overtaken vehicle was corroborated by similar experimental studies conducted by **Howell et al.**, performed at a 12.5 percent scale [2]. This wind tunnel study used a stepper motor and a system of cables and pulleys to simulate the overtaking maneuver between two vehicles (a truck and a car) for a range of yaw angles (-10° to +10°) and a range of lateral separating distances. It estimated the incremental loads generated in such a maneuver, showing that these loads increased in the presence of a crosswind when the car was on the leeward side of the truck.

A three-dimensional, CFD, quasi-steady study on a car overtaking a truck by **Pasala et al.** [13] assumed that the relative velocity between two vehicles has a negligible effect on the flow distribution and the forces acting on the car. This study lacked experimental validation.

Existing literature data are inconclusive regarding the effects of overtaking velocity when the size difference between the two vehicles is significantly large (e.g., a car overtaking a truck). It is suggested that the absolute velocity of the larger vehicle is dominant in determining the loads on the smaller vehicle [2]. However, according to **Shreffl et al.** [14], on the basis of an on-road experiment, the lateral loads on a car overtaking a truck tend to increase linearly with the overtaking velocity. Simulations of the dynamic passing process between generic vehicles (Ahmed bodies) were conducted by **Uystepuyst and Krajnović** [15] by using the URANS model and studying changes in the force and moment coefficients of the overtaken vehicle. However, the Ahmed bluff body was too simplified to be able to provide a basis for drawing quantitative conclusions from these simulations.

3. CFD BASICS

Upon kicking off analyzing a given flow downside in Fluent the user is round-faced with 3 initial inputs; pure mathematics, meshing and downside setup. once process every input the matter is solved then brought into a post processor to visualize the results.

The methodology for the CFD analysis conducted is outlined during this section of the report for every of the subsequent investigations;

1. The Cube
2. Interference gap dimension
3. Interference gap blocker
4. Deflector
5. Crosswind

3.1 The Cube Study

The purpose of finding out such AN elementary form during a project relating to vehicle aeromechanics is principally to induce a pity the computer code, however conjointly to determine a validation case for future truck models. in step with results documented by Frank M. White in 'Fluid Mechanics' edition five, the coefficient of drag of a cube immersed during a free stream flow field is one.07 for flows with a Sir Joshua Reynolds variety of ten,000 or larger [8]. The flow during this instance is perpendicular to a face of the cube. If the same drag coefficient is replicated it'd be an honest indication that correct CFD techniques square measure been deployed and an honest indication that results obtained for resultant truck models square measure moderately correct. This initial case, the cube, also will be noted because the action throughout the report. it's planning to be vigorously analyzed and tested. A secondary purpose of analyzing this case in such detail is to determine what influence every of the parameters inside the program wear the results. information of such influence on the action will then be applied to resultant cases.

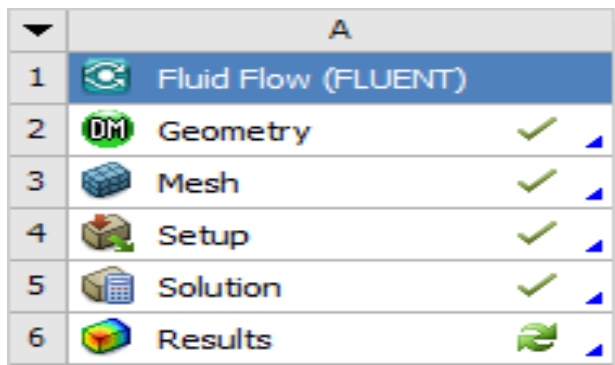


Fig. 2. Problem Specification in Fluent

3.2 Geometry Setup

For this case the flow of air over a cube is being studied. an impression volume for this air is needed to analyze the flow however it's not clear however huge it's to be to accurately analyze it. once size the management volume one doesn't wish to style it too huge as a result of this can increase the weather at intervals the domain and additionally the computing time. With this in mind Associate in Nursing initial enclosure was designed. Analysis was performed victimization it and results square measure documented in Chapter four.

The problem consists of a 1m x 1m x 1m cube in an exceedingly massive enclosure. the scale of the enclosure was chosen as 3m ahead of the cube, 10m behind it and 1m either side. This enclosure is simply Associate in Nursing initial one with many areas to the cubes rear to accommodate turbulence within the wake of the flow.

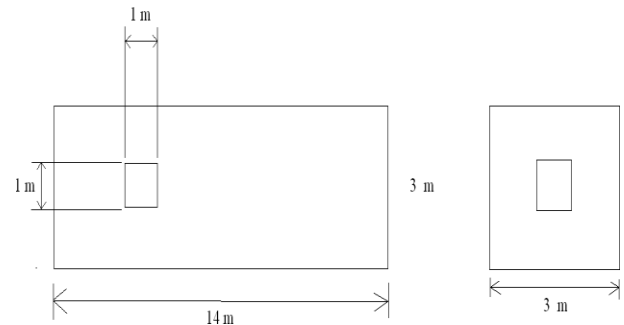


Fig. 3. Geometry Specification NOT TO SCALE

3.2 Mesh Setup

The initial mesh used a hundred and seventy,000 components with a most face size of zero.05m long. It's necessary to notice this was simply associate degree initial approximation on the quantity of components and their size needed to duplicate White's coefficient. A a lot of rigorous analysis was performed when getting a result victimization this initial mesh. Figure 0-3 below shows an image of the mesh used on only 1 plane of symmetry

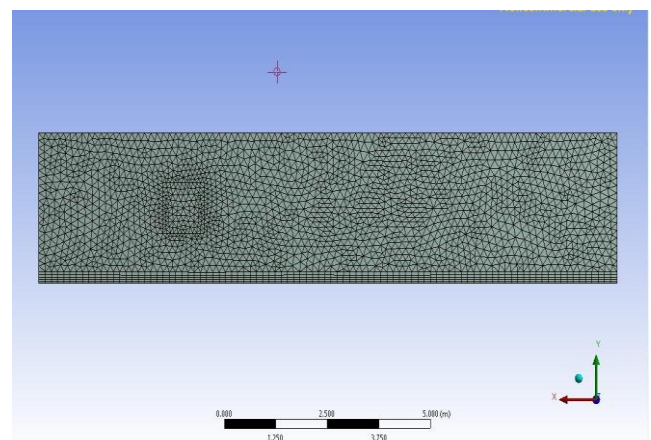


Fig. 4. Initial Mesh Specification

3.4 Problem Setup

The problem was affected to 2 vital boundary conditions; associate flow of air painter range of ten,000 or on top of and a free-stream flow condition. Free-stream flow suggests that the cube won't be stricken by physical phenomenon growth from different surfaces. The painter range is that the quantitative relation of mechanical phenomenon force to viscous force outlined.

The painter range is ten,000 or on top of, the fluid density and body area unit constant at one.225 and zero.01206 kg/m s severally and therefore the characteristic length is that of the cube (1 m). Inputting these parameters into this

equation leads to a rate of ninety-eight.45 m/s. this can be the rate needed within the simulation. but since the required painter range is ten,000 or on top of this will be rounded up to a hundred m/s for simplicity. The values employed in the CFD simulation area unit outlined in Table one.

Variable	Quantity
ρ	1.225
μ	0.01206 kg/m s
L	1 m
v	100 m/s

Table 1. Reynolds number parameters

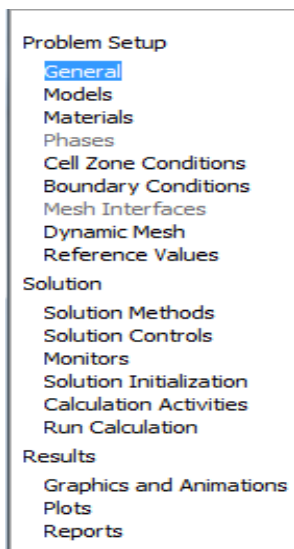


Fig. 5. Problem setup in Ansys Fluent

A pressure based mostly convergent thinker was employed in the simulation as a result of for Associate in Nursing application like this (fluid flow over Associate in Nursing immersed body) that's what's counseled by ANSYS Advantage [9]. The pressure based mostly convergent thinker in Fluent reduces the convergence time by the maximum amount as 5 times. It will this by finding momentum and pressure based mostly equations in a very coupled manner [9]. The simulation ran employing a steady-state time domain rather than a transient one. this is often as a result of the author wished a gentle state resolution with all the fluctuations time averaged out.

The flow regime chosen for this primary simulation was the fundamental inviscid model. The stipulation obligatory on the truck was a no-slip condition. this suggests the speed is zero on all the surfaces of the truck. The reference values employed in the simulation square measure outlined.

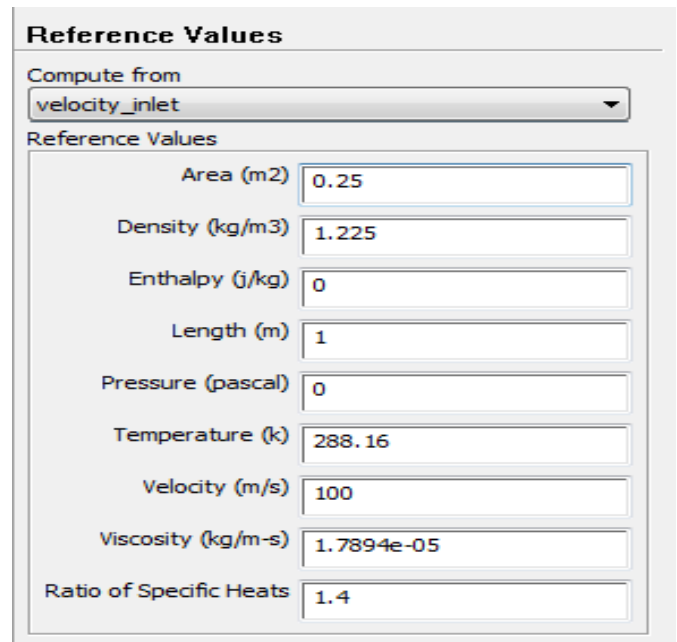


Fig. 6. Reference values used in the initial simulation

As will be seen from Figure 0-5. a reference space of was chosen. this is often as a result of the 2 symmetry planes used quarters the realm needed for the simulation. For this simulation the answer ways and controls were untouched, left because the default settings.

After specifying the reference values a monitor for drag force was created. The drag on the cube is that the solely drag force needed therefore this was given by selecting 'cube' below the named picks. The direction was conjointly given because the negative x-direction (-1,0,0). this is often the direction opposition the front face of the cube. A hybrid format was chosen over the quality format as a result of it expeditiously initializes the answer based mostly strictly on simulation setup [9] that means the user doesn't got to offer extra inputs. Once the answer was initialized a call for participation of three hundred iterations was created and also the program calculated the answer. The results for this simulation ar documented in Chapter four.1.

3.5 Re-meshing setup

Based on the results from the initial enclosure it seems a more robust mesh is needed therefore a little mesh convergence study was undertaken. a correct mesh convergence study cannot properly be undertaken as a result of the most grid size is simply too little. therefore, this mesh convergence study is actually additional of a mesh comparison.

For the study 3 grid sizes were analyzed; a rough mesh, a medium mesh and a fine mesh, documented in Table a pair of. The course mesh used was simply the initial mesh Fluent creates around a body of air. there's no refinement.

Mesh	Element size
Coarse mesh:	21,283 elements
Medium mesh:	104,898 elements
Fine mesh:	490,480 elements

Table 2. Grid sizes used for the comparison

The medium mesh contains roughly a hundred and five,000 parts. the most distinction between this mesh and therefore the course one is that the component filler. the dimensions of associate parts face during this mesh were restricted to a most zero.6m. This accumulated the cell count from twenty-one,000 to 105,000 parts.

For the fine mesh, advanced filler functions were employed in Fluent’s meshing application. The advanced filler perform chosen was proximity and curvature. This yields a smaller component size the nearer the mesh is to the sides of the cube. A medium connectedness Centre was chosen that evoked a two hundredth rate for the weather. A slow component transition rate was per this case as a result of its additional suited to CFD analysis than a quick transition thanks to the very fact it fills the degree with parts additional swimmingly and expeditiously. The minimum component size was nominative as zero.01m even supposing the program in all probability won’t produce parts this tiny. The minimum size here isn’t that necessary. the utmost component size is but, and it absolutely was set to zero.12m. With these settings in situ the grid size accumulated to 490,480 parts. A comparison may be seen of the coarse grid and therefore the fine grid

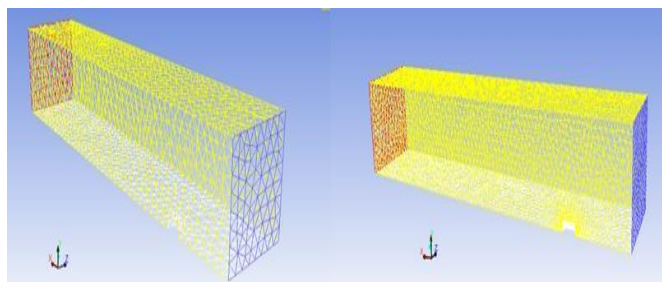


Fig. 7. Comparison of the coarse grid (left) and the fine grid (right)

Notice the dense cluster of components in real time round the cube within the fine grid. this can be due to the slow transition and proximity and curvature operate used. Once the mesh was created the Named alternatives were selected as before.

3.6 Problem setup

The problem was outlined in for the most part an equivalent method for the second enclosure because it was the primary.

There square measure 2 variations though; a flow regime study ANd an upwind theme study. A a lot of elaborated investigation went into the flow regime and upwind theme resolution strategies. For the flow regime investigation AN inviscid model, stratified model, commonplace and realizable k-epsilon models beside commonplace and non-equilibrium wall functions were all compared against every of the 3 mesh sizes. The results of that square measure documented in Figure 4-0-1. This analysis was carried so as to spot what impact every model had on the general coefficient. Near-wall treatment is vital|vital|important} in external mechanics therefore it absolutely was additionally important to look at completely different treatments offered in Fluent. the 2 closes to wall treatments thought-about were commonplace wall functions and non-equilibrium wall functions.

4. METHODOLOGY

The proposed work results in identifying sufficient tolerance in changing the material (EN 8 steel & EN 24 steel). It is expressed in methodology as,

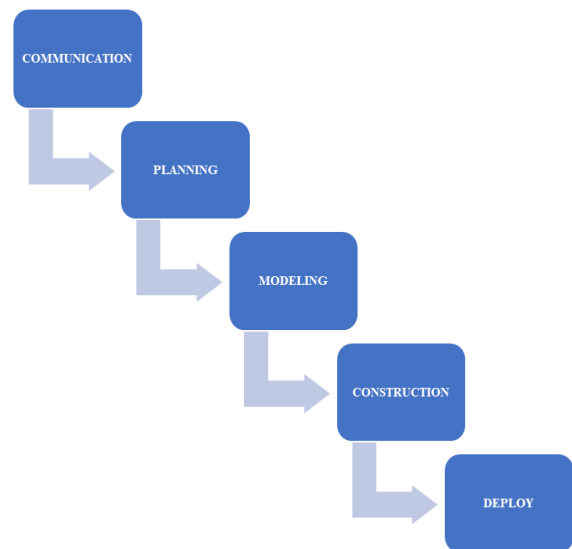


Fig. 8. Waterfall Model

We have decided to complete the project in simple waterfall model,

4.1 Communication Phase

Communication phase includes:

- Discussion of topic with guide
- Actual farm visit and understanding various farming method
- Literature survey
- Problem identification

- Analysis of problem
- Concept development
- Discussing various certainties and uncertainties

4.2 Planning Phase

Planning phase includes:

- Process planning
- Raw material planning
- Force analysis
- Process scheduling

4.3 Modeling Phase

Modeling phase includes:

- Design of various components
- CAD modeling of components
- Assembly model of component
- Prototype model making

4.4 Construction and Testing

Construction phase includes:

- Selection of proper manufacturing methods
- Working as per process scheduling and plan
- Testing of equipment on field
- Error analysis
- Repair if any

4.5 Deployment

- Comparing the project with the designed output
- Preparation of testing results
- Preparation of project report
- Final submission of project

4.6 Design procedure

Before we proceed to the process of manufacturing, it's necessary to have some knowledge about the project design essential to design the project before starting the manufacturing. Maximum cost of producing a part of product is established originally by the designer.

Design procedure:

When a new product or their elements are to be designed, a designer may proceed as follows:

- Make a detailed statement of the problems completely; it should be as clear as possible & also of the purpose
- for which the machine is to be designed.
- Make selection of the possible mechanism which will give the desire motion.
- Determine the forces acting on it and energy transmitted by each element of the machine
- Select the material best suited for each element of the machine.
- Determine the allowable or design stress considering all the factors that affect the strength of the machine part
- Identify the importance and necessary and application of the machine
- Problems with existing requirement of the machine productivity and demand.

5. CONCLUSION

From the analysis conducted during this report some vital conclusions may be drawn. The flow over AN immersed bluff body is nearly entirely pressure based mostly. The results show the contribution of skin friction to the general coefficient is negligible, but 1 Chronicles in most simulations. this means that so as to boost vehicle mechanics, the variables within the equation that govern pressure drag square measure the sole variables of importance once coming up with mechanics blunt bodies. all sorts of trucks may be thought-about blunt bodies because of their natural cuboid pure mathematics.

When analyzing the simulations of the cube it may be explicit that for the given mesh sizes used, very little distinction exists between any of the varied flow regimes used and any of the upwind answer schemes used. The coefficient calculated, mistreatment AN inviscid model and initial order upwind answer theme for Mesh two, is simply 100% off the coefficient calculated for Mesh three with a second order upwind answer theme, and realizable k-epsilon model. This little distinction poses a motivating question; however, correct do i desire a answer to be, given the additional computation value and complexness of top quality meshes, difficult turbulence models and better order upwind schemes? for several functions a ballpark result at intervals plus/minus 100% of the particular answer is adequate. This

tolerance level but wouldn't answer within the style of drag reduction devices as a result of such devices would solely improve drag by a most of 30-50%, which means each per cent is crucial.

With relevancy enclosure style for cubes, it absolutely was found that a little size leads to a blockage impact made by the cube leading to a better coefficient because of the pressurized force exerted by the air. Enlarging the enclosure eliminated this impact and an identical answer to Whites was obtained. Analysis conducted on the pressure contours of the cube showed an outsized pressure gradient between the front face subject to the flow and also the rear face within the wake of the flow; the explanation for this pressure gradient is explained in Chapter one.

Results from the interference gap investigation indicate a linear relationship between the coefficient and also the gap breadth; this relationship wants additional confirmation but as none of the literature reviewed as a part of this project reported any relationship between drag and gap width. Upon analyzing the pressure distribution on every of the truck surfaces, it absolutely was discovered that any explicit face of the truck was subject to either pressure drag or friction drag however not each. The faces perpendicular to the flow were accountable for the pressure drag and also the faces tangential to the flow were accountable for the skin friction. this is often what was expected to happen as explained in Chapter one. The device style applied as a part of this project was self-made in this drag reduction of up to half-hour was recorded with the models used. This study concerned AN repetitive style method however it ne'er reached AN improvement purpose for the device geometry; this might be studied in additional detail during a later Final Year Project.

The cross-wind study was conjointly self-made. Incoming wind occurrence on the truck at 5° angle intervals was studied. Monitors for the drag force within the axial direction of the truck were created and solutions were examined and compared to ones recorded by Hucho in [3]. Similar trends within the relationship between coefficient and wind angle were discovered between results recorded during this project and people revealed by Hucho. finally, the rise in wind Angle corresponds to a rise in drag force up to an angle of roughly 20° once that purpose it reduces by a similar rate. The drag increase because of the wind angle was combated via a tangle reduction device placed within the interference gap. At the 30° wind angle the gap blocker reduces the drag by pure gold in step with the simulated results.

Conclude, there square measure huge advantages to be gained from putting in drag reduction devices to category eight trucks. As explicit in Chapter one fuel consumption relies on mechanics drag, thus by reducing the drag force by means that of add- on devices, transportation firms with fleets of fuel thirsty trucks will save substantial cash on fuel.

REFERENCES

- [1] Li Song, Zhang Jicheng, Liu Yongxue, Hu Tieyu, "Aerodynamic Drag Reduction Design of Van Body Truck by Numerical Simulation Method," in Second International Conference on Digital Manufacturing & Automation, Changchun, 2011.
- [2] U.S. Department of Energy, Transportation Energy Data Book, Oak Ridge, Tennessee, 2012.
- [3] W.-H. Hucho, The Aerodynamics of Road Vehicles, 1988.E.
- [4] Mu Wang¹, Qiang Li, Dengfeng Wang, Changhai Yang, Jinlong Zhao, Guijin Wen, "The Air-deflector and the drag: A Case Study of Low Drag Cab Styling for a Heavy Truck," Jinan, 2009.
- [5] R. Drollinger, Heavy duty truck aerodynamics, Society of Automotive Engineers, 1987.
- [6] R. Wood, "A Discussion of a Heavy Truck Advanced Aerodynamic Trailer System," 2010.
- [7] J. Eaton, "Session 26: Computational Fluid Dynamics," Galway, 2013.
- [8] F. White, Fluid Mechanics, WCB McGraw-Hill, 2005.
- [9] Ansys, "Ansys Advantage Volume 5 issue 1," 2011.
- [10] M. Lanfrit, "Best practice Guidelines for Handling Automotive External Vehicle Aerodynamics with FLUENT," Darmstadt, 2005.
- [11] F. Browand, "Reducing Aerodynamic Drag and Fuel Consumption," 2011.
- [12] F.-H. Hsu, "Drag Reduction of Tractor-Trailers Using Optimized Add-On Devices," American Society of Mechanical Engineers, Los Angeles, 2010.
- [13] Subrahmanya Veluri, Christopher Roy, "Joint Computational/Experimental Aerodynamic Study of a Simplified Tractor/Trailer Geometry," American Society of Mechanical Engineers, Blacksburg, 2009.
- [14] T. Favre, "Aerodynamic Simulations of ground vehicles in Unsteady Crosswind," 2011.
- [15] F.-H. Hsu, "Design of Tractor-Trailer Add-On Drag Reduction Devices Using CFD," Los Angeles, 2009.