TRIDIMENSIONAL SIMULATION OF THE SPEED, PRESSURE AND TURBULENCE FIELDS OF THE AIR FLOW ON THE GEOMETRY OF A FREIGHT TRUCK INSIDE A WIND TUNEL.

LUIS DANIEL MENDOZA P
lmendoza@unet.edu.ve
RAUL ANTONIO COLMENARES P.
raulcolcan@yahoo.com

SUMMARY

This study shows the air flow behavior through the geometry of a freight truck inside a AF6109 wind tunnel with the purpose to predict the speed, pressure and turbulence fields made by the air flow, to decrease the aerodynamic resistance, to calculate the dragging coefficient, to evaluate the aerodynamics of the geometry of the prototype using the CFD technique and to compare the results of the simulation with the results obtained experimentally with the "PETER 739 HAULER" scaled freight truck model located on the floor of the test chamber. The Geometry went through a numerical simulation process using the CFX 5,7. The obtained results showed the behavior of the air flow through the test chamber, and also it showed the variations of speed and pressure at the exit of the chamber and the calculations of the coefficient and the dragging force on the geometry of the freight truck. The evaluation of the aerodynamics showed that the aerodynamic deflector is a device that helped the reduction the dragging produced in a significant way by the air. Furthermore, the dragging coefficient and force on the prototype freight truck could be estimated establishing an incomplete similarity.

Key words: Tridimensional simulation, numeric simulation, CFD, AF6109 wind tunnel, "PETERBILD 739 HAULER" freight Truck, CFX 5.7, model, prototype.

Introduction

The aerodynamics has given a turn to the competitions of automobiles and also to vehicles of common use. These changes in the lines have as benefit the diminution of the turbulence and the fuel consumption. This diminution of the coefficients of drag and sustenation has been the result of experimental studies in wind tunnels for all type of vehicles, as well as of the use and development of programs of Dynamics of Fluidos Computacional (CFD), which allows to determine and to optimize the coefficients of aerodynamic performance of the vehicles of one more a faster and economic way. In this work a study of the behavior of the air flow through the geometry of a truck within

the AF6109 wind tunnel, in order to predict the speed fields, pressure and turbulence produce by the air, to diminish the aerodynamic drag, to calculate the drag coefficient, to make an aerodynamic evaluation of the geometry of the prototype by means of the CFD technique and to collate the results of the simulation with the obtained ones experimentally, for which the scale model of a truck model "PETERBILT 379 HAULER" was used.

Flow Around Objects
For this type of flows it interests, mainly, to know what type of forces exerts the flow on the object. If the body has a form and nonsymmetrical direction, the forces and moments that the fluid exerts have components in the three coordinate directions. One gets used to choosing that one of the Y-axes is parallel to the direction of the current uniform as is observed in figure 1. The force on the body in the direction of this axis is denominated Force of Drag, and torque is denominated Rolling moment. Also, it is usual to choose that one of the perpendicular directions of the direction of the flow agrees with the direction of the gravity. The flow force that appears in this direction is denominated Lift force, and in the other perpendicular coordinate direction, the force is denominated Lateral force. Nevertheless, generally, the submerged bodies have at least a center line with respect to the flow. For these cases they solely appear Forces of Drag and Sustenation and Rolling moment. If the body has two planes of symmetry, it solely appears the Force of Drag.
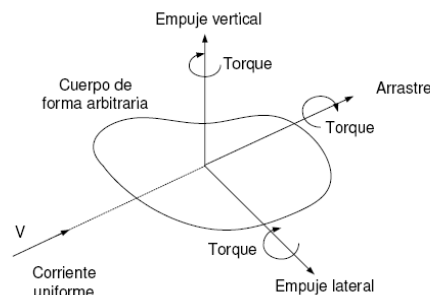

Figure 1 Flow around objects

The forces that a flow exerts on an object can be calculated integrating the normal as much as the cutting efforts, on the surface:

$$\vec{F} = \int_S \tau_w \, dS + \int_S (-P) \, dS$$

A form to find solutions for flows to high Reynolds number is the combination of the solution of the flow in two different regions: the region near the surface is solved through the boundary layer approach; and the flow far from the surface is solved using potential flow.

Mathematical description of the Flow of Fluids
The derivation of the main equations of the dynamics of fluids is based on the fact that the dynamic behavior of a fluid is determined by the following Laws of Conservation:
- The Law of Mass Conservation.
- The Law of Acquired Speed Conservation (Momentum).
- The Law of Energy Conservation.

General Transport Equation:

The conservation principles, shaped in mathematical expressions, applied to control volume and written in an integral form, can be represented by a generic equation, denominated Transport Equation.

$$\frac{\partial}{\partial t}\int_V \rho\phi\, dV + \oint_A \rho\phi\, \mathbf{V}\cdot d\mathbf{A} = \oint_A \Gamma\nabla\phi\cdot d\mathbf{A} + \int_\forall S_\phi\, dV$$

In this work the model of turbulence k was used k-ϵ, although it is the most robust model to evaluate the turbulence, is not most advisable to evaluate the aerofoil profile mainly, because it is not good to consider the effects on the boundary layer, when is analyzed to very high Reynolds numbers or very high speeds. For the present investigation a speed of 22,69 was considered m/s, thus it can be used for this model. The air was considered as well like incompressible fluid and with ideal gas behavior.

The model k-ϵ in the cfx-5.6

The turbulent kinetic energy (k) is defined as the variation of the fluctuation in the speed, whereas ε it is the reason with which the fluctuations of the speed dissipate.

The basic equations are:

Continuity:

$$\frac{\partial\rho}{\partial t}+\nabla\bullet(\rho U)=0$$

Movement Quantity:

$$\frac{\partial\rho U}{\partial t}+\nabla\bullet(\rho U\otimes U)-\nabla\bullet(\mu_{eff}\nabla U)=\nabla p'+\nabla\bullet(\mu_{eff}\nabla U)^T+B$$

Where **B** is the sum of the body forces, μ_{eff} is the efective viscosity associated to the turbulence and p' it is the modified pressure and this given by

$$p'= p+\frac{2}{3}\rho k$$

$$\mu_{eff}=\mu+\mu_t$$

Where μ_t is turbulent viscosity. Model $k-\varepsilon$ assumes that turbulent viscosity is associated to the turbulent kinetic energy and the dissipation, by means of the relation

$$\mu_t=C_\mu\rho\frac{k^2}{\varepsilon}$$

Where C_μ is a constant.

Turbulent kinetic energy:

$$\frac{\partial\rho k}{\partial t}+\nabla\bullet(\rho Uk)-\nabla\bullet\left(\frac{\mu_{eff}}{\sigma_k}\nabla k\right)=P_k-\rho\varepsilon$$

Dissipation of the energy:

$$\frac{\partial\rho\varepsilon}{\partial t}+\nabla\bullet(\rho U\varepsilon)-\nabla\left(\frac{\mu_{eff}}{\sigma_\varepsilon}\nabla\varepsilon\right)=\frac{\varepsilon}{k}(C_{\varepsilon1}P_k-C_{\varepsilon2}\rho\varepsilon)$$

Where $C_{\varepsilon1}$, $C_{\varepsilon2}$, σ_k and σ_ε are constants. Being P_k the rate of production of the turbulent energy, which stops an incompressible flow is:

$$P_k=\mu_t\nabla U\bullet(\nabla U+\nabla U^T)-\frac{2}{3}\nabla\bullet U(\mu_t\nabla\bullet U+\rho k)$$

Navier-Stokes Simplified Equations

The equations are simplified considerably when they are applied to incompressible flow with constant viscosity as for the case of the air, therefore under these conditions they are reduced to:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

Space Discretización and Methods of Solution

Throughout the history of the numerical methods applied to the mechanics of the fluids many they have been the approaches or forms to pass of the model Physic-Mathematical to the Numeric-Discreet model. The modern technologies are constructed with an important additional requirement, consists of the condition of which the final result of the discretización is easily integralable in a certain architecture of calculation. The application of the basic laws of the physics allows to obtain the relations between the different variables through a set from differentials equations into partial derivatives: they are the equations of constitution of the mechanics of fluids (continuity, angular momentum, energy). Due to its great complexity, it is necessary to look for alternative methods that provide a good prediction, that is, methods that provide an approximate numerical solution. Thus, the approximated numerical solution will be obtained from the resolution of a series of algebraic relations obtained by means of techniques of discretización of the differentials equations into partial derivatives. Several methods for the accomplishment of a space discretización, among which it is possible to mention the Method of Finite Differences, the Method of Finite Elements and the Method of Finite Volumes. This latter is one of the most used and in what the CFX-5.7 is focused.

Geometry of the Truck

In order to create the geometry of the truck software Solid Edge V.15. was used This solid was constructed taking the necessary measures from a model on scale 1:75 of a PETERBILT 379 HAULER truck, that allowed to make the experimental measurements in the AF6109 wind tunnel.
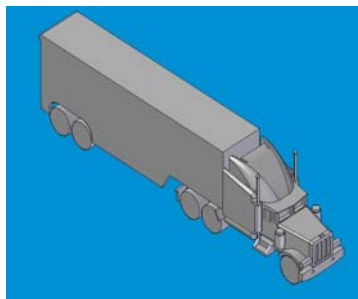

Figure 2 Geometry of the Truck

Main Characteristics of the AF6109 Wind Tunnel

- It is of open circuit.
- The terminal velocity of the air in the section of test is 28 m/s (100 km/h).
- The Mach number is 0.08.
- The test section type is closed, it has rectangular form (it diminishes the effect wall in the bidimensional profiles) and presents/displays the following dimensions:
- Cross-sectional area: 210 mm x 360 mm

- Length: 500 mm
- Total dimensions: 3400 mm x 800 mm x 1800 mm

In order to make this experimentation it is remembered that the air is a stationary flow, incompressible, that far from the profile presents/displays a speed uniforms but when it goes through the geometry, fields of speed, pressure and turbulence are generated which are object of this study. In addition it is considered that the model of the truck is not submitted under thermal effects. Will assume that viscosity μ and the density ρ of the air they are constant at any moment. Within the camera of test the model of the truck will not have any rake with respect to the horizontal since the main forces of study is the drag force that exerts the air on the profile which this related to the aerodynamic form of the model.

In the study the model of a **"PETERBILT 379 HAULER"** truck was used which is on scale 1:75 with respect to the prototype, this one has an aerodynamic form designed by the manufacturer on which experimental measurements were made of the fields of speed and pressure (figure 3).
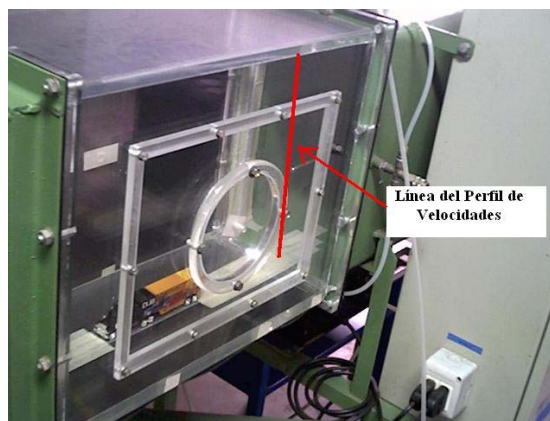

Figure 3 Chamber test of tunnel of Vento AF6109

Analysis of Sensitivity of the Mesh

This study consisted of an iterative process where successively it was managed to improve the quality of the mesh comparing the executions made in the program of simulation with the data collected experimentally in the wind AF6109 tunnel beginning with a mesh made by defect soon to refine the same one and to obtain most suitable according to the convergence parameters. Between the parameters of convergence used to obtain the most suitable mesh there are the residual average (RMS) obtained during the simulation, the speed and the pressure taking at the tunnel exit. Initially the suitable length of the virtual wind tunnel was analyzed, for what the condition of developed turbulent flow was evaluated, to fulfill the condition of the trailing edge, which consisted of a differential pressure equal to zero. Later an analysis was made of the mesh sensitivity, with ANSYS ICEM CFD 10 obtaining therefore a better control of the mesh in those zones of special care, creating a size of the uniform element on the study dominion, as it is in figure 3. The inflation layers allow to obtain in the surface of the geometry of the virtual truck the condition of speed zero in the surface and formation of the boundary layer in each one of the borders in contact with the fluid. On the other hand the size of the elements is small in those zones where the mesh requires it to adjust to geometry. The size of the elements is bigger in those regions of the mesh far from the geometry.
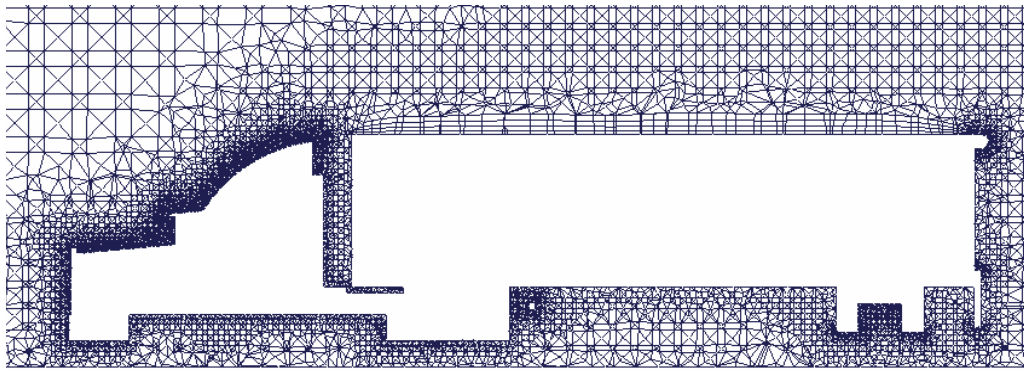
Figure 3 Mesh of the calculation dominion

The speed average to the entrance that was taken for the simulation process was the seizure by means of the experimental wind tunnel that threw a value of 22,69 m/seg.

In figure 4 a graph is observed that presents/displays the speed average at the exit against the number of elements for each joint of meshes to evaluate the behavior of the simulation process. In the graph of figure 4 the process of refinement of the mesh can be observed where from 3505945 elements no longer it is necessary to continue increasing the amount of such since the speed average stays in a value near the established one for the simulation, therefore, he is not justifiable to continue increasing the calculation exigencies.
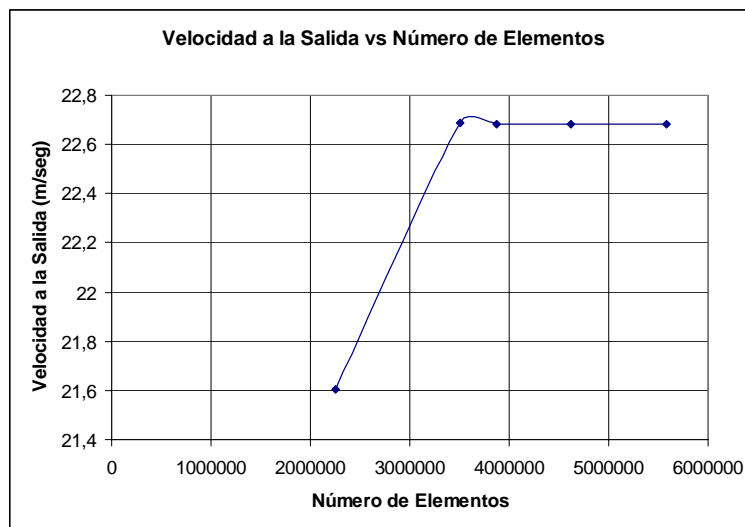

Figure 4 Speed Average at the exit according to the Process of Refinement

Validation of the Mesh
The mesh was validated by means of the data collected by means of experimental measurements in the AF6109 wind tunnel. Underneath a graph is shown where validity of the selected mesh is checked, one for the simulation, comparing the experimental graph which can be observed in figure 5 with the obtained one by means of the simulation software. The experimental graph is a velocity profile taken by means of the pitot tube at the exit of the test chamber, the same sample that when increasing the height increases the speed until reaching a speed near the seizure to the entrance of the test chamber.
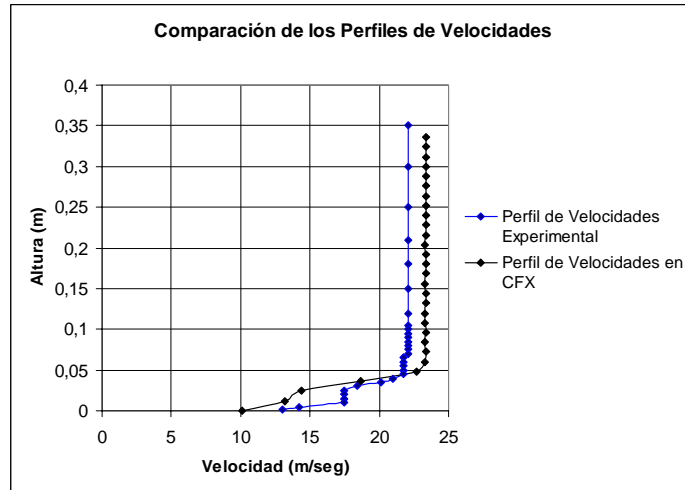
Comparación de los Perfiles de Velocidades

(Altura (m) vs Velocidad (m/seg))

Perfil de Velocidades Experimental
Perfil de Velocidades en CFX

Figure 5 Velocity profiles thrown by CFX and the obtained one experimentally

Model and the Prototype

In the execution of this comparative study two involved models were taken which are: a real model and a virtual model. The real model is conformed by the truck on scale 1:75 according to a PETERBILT 379 HAULER prototype, the real model is located in the AF6109 wind tunnel that is where the experimentation is made and the virtual model is the one created with software to put under it a process of computacional simulation. The prototype is the truck that has the natural scale that counts on the real dimensions, the prototype was used to construct the real model that is on a scale 1:75. As the real model were used to construct the virtual model if a comparison between the measures is made of both these are equal since they were done on scale 1:1.

The virtual model was validated experimentally through data taken from the real model. The speed at the entrance of the test chamber was set to 22.69 m/seg for the experimental wind tunnel.

The real model of the truck is related to a prototype through a scale 1:75, therefore, a dynamic similarity settled down incomplete to be able to consider the approximated value of the force of drag in the prototype, and through the computacional simulation the force of drag for the real model of the truck could be considered.

Also the result of the force of drag the model with the aerodynamic baffle plate and without him was analyzed as shown in figure 6.
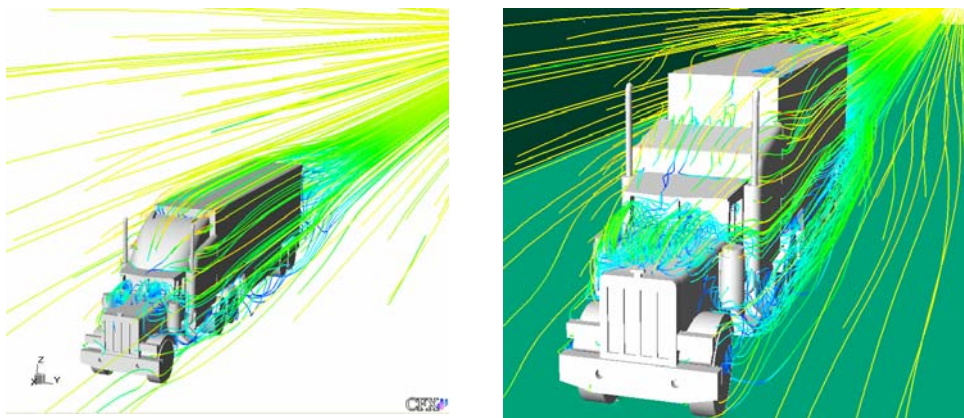

Figure 6: Trajectory lines on the Models

For the aerodynamic evaluation the air drag force was considered. By means of the simulation made with software this force was determined. In this study it was considered to clear the aerodynamic baffle plate to evaluate its influence in the aerodynamics of the truck and simultaneously to calculate by means of the simulation the drag force, based on the mesh chosen according to the sensitivity study that experimentally allowed to select the optimal mesh that adapts more to the conditions of this study.

The mesh raised for the truck without the aerodynamic baffle plate threw acceptable results for the values average of the pressure and speed at the exit, as well as a turbulent velocity profile developed at the exit but did not exist a convergence on the basis of the residual averages (RMS), that is to say, all the values were not below 1E-4.

The force of drag for the model of the virtual truck without the aerodynamic baffle plate had a value of 0,191678 N being greater than the force of drag obtained by means of the virtual truck with the aerodynamic baffle plate that had a value of 0,158458 N in the simulation process. For the two configurations the value of the lift force was almost null. The coefficient of adimensional drag for the model of the virtual truck without the aerodynamic baffle plate had a value of 0,62 being greater than the coefficient than corresponds to the virtual truck with the aerodynamic baffle plate that threw a value of 0,52, the value that is obtained for each coefficient is due to the change in geometry.

On the basis of the simulation process the force of drag calculated for the prototype the truck without the aerodynamic baffle plate had a value of 1,54 KN being greater than the force obtained by means of the truck with the aerodynamic baffle plate that threw a value of 1,29 KN.

The power necessary to overcome the aerodynamic drag in the truck with the baffle plate threw a value of 35,18 KW being smaller to the one than it calculated without the baffle plate which threw a value of 41,99 KW.

Conclusions

- It was determined by means of the simulation process that the mesh selected when having surfaces refined in the walls of geometry threw the best results since the condition of boundary layer is reached better in the air flow.

- In the aerodynamic evaluation of geometry one demonstrated quantitatively that the drag force is more significant that the lift force which stayed in a despicable value.

- It was verified that the force of drag for the truck without aerodynamic baffle plate is greater than the calculated one with the baffle plate, therefore concludes that this device is important at the time of diminishing the value of the drag that produces the air on geometry.

- By means of the calculation of the required power to overcome the aerodynamic drag it was verified that more energy in a truck without the aerodynamic baffle plate is consumed that in that it has it, therefore concludes that with the use of this device it is contributed to the saving of the fuel.

References

- Bardina, J.E., Huang, P.G. and Coakley,T.J.,"Turbulence Modeling,Validation, Testing and Development," NASA Technical Memorandum 110446, 1997. (Ver también: Bardina, J.E., Huang, P.G. and Coakley,T.,"Turbulence Modeling Validation",AIAA Paper 97-2121).

- Heberly Adam, (1999), "Velocity and Aerodynamic Drag". Estados Unidos, redwood

- John D: Anderson, Jr. (1995), "Computational Fluid Dynamics". Estados Unidos, Mc Graw Hill.