V. European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
Cavit Çınar and M. Özgür Arslan
Lisbon, Portugal, 14–17 June 2010

AERODYNAMIC OPTIMIZATION STUDY FOR ROOF SPOILER AND SIDE EXTENDER PARTS OF FORD HEAVY DUTY TRUCK USING CFD TOOLS

MSc. Cavit Çınar*, MSc. M. Özgür Arslan†

*FORD-OTOSAN Gebze Eng. Vehicle Eng. Dept.
Tubitak MAM. Technopark Gebze 41470 Kocaeli/TURKEY
e-mail: ccinar1@ford.com

† FORD-OTOSAN Gebze Eng. Vehicle Eng. Dept.
Tubitak MAM. Technopark Gebze 41470 Kocaeli/TURKEY
e-mail: oarslan1@ford.com

Key words: Exterior Aerodynamics, Design Optimization, CFD, Heavy-Duty Truck Spoiler, Heavy-Duty Truck Side Extender, Drag Coefficient (C_D).

Abstract. *Aerodynamics of trucks and other high sided vehicles is of significant interest in reducing road side accidents due to wind loading and in improving fuel economy. Due to this main reasons Aerodynamics has become the latest battlefield between heavy-duty truck manufacturers. The CFD tools in aerodynamic design process have been commonly used in otomotive industry in two last decades.* CFD, *Shorter development period offers good advantage for the companies in a highly competitive market. Thus; using CAE tools in early phase of design which one can do design iterations with a given CAD data (without having prototype vehicle) ; leads to find the best design & reduce the numbers of prototypes required for tests. In this paper presents an aerodynamic simulation process have been used to optimize Ford Heavy Duty Truck Roof Spoiler & Side Extender designs. Firstly w/o Aero-Kit Ford Heavy-Duty Truck has been analyzed by TASE group after than current spoiler& side extender have been optimized.*

1 INTRODUCTION

Minimizing fluid-dynamic drag through careful shaping has been practiced by boat and ship designers for many hundreds of years. Proposals for means to reduce the aerodynamic drag of road vehicles have been made since approximately 1914, when the speed of horsedrawn vehicles began to be exceeded [1]. Because fuel supplies were plentiful and highway speeds were still generally low, serious attempts to reduce aerodynamic drag were sporadic and not often adopted until the oil crisis of the 1970's. The oil crisis stimulated the development of add-on devices that could be affixed to trucks that were already in use, and workshops such as those represented by reference 1 were organized to disseminate ideas and information. However, the rate of acceptance and use of add-on devices was modest. A private, anecdotal survey was made by this author during the summer of 1975 coincident with a 3600-mile vacation trip (California to Iowa, round trip)[2]. This survey, which involved a sample field of 965 tractorsemitrailer vehicles, revealed that 11 percent of the tractor-semitrailer combinations having van-type trailers were using cab-mounted add-on deflector shields[2].

The purpose of this study is to improve an aerodynamically efficient spoiler & side extenders for Ford Heavy-Duty Truck vehicle. Current study is to design a new spoiler & side extenders by using Computational Fluid Dynamics (CFD) tools to have cost reduction benefit, provide sunroof functionality and further drag reduction on the truck.

Instead of simulating a real truck model, a simplified truck model was used to lower the grid generation complexity and findings were verified with the full truck&trailer model. Even with a simplfied body model, however, many details of a real truck-trailer configuration remained.

Model geometry, discretization of the physical domain, and choice of a suitable numerical computing scheme are significant factors that can determine the accuracy of the process. The method uses three dimensional Reynolds-averaged Navier Stokes equations that are solved using a finite volume method. Those k-ε and models are used for the convergence of the turbulent quantities.

2 APPLIED COMPUTATIONAL FLUID DYNAMICS (CFD) PROCESS

The beginning steps of a CFD computation are essentially the same irrespective of the method[1]. A typical aerodynamic CFD simulation of a ground vehicle starts with a computer model. The model could be generated with a CAD package or surface scanner software. Since most of the CFD tools are compatible with solid modeling packages, the CAD model of the ground vehicle is usually transferred to a solid modeling or mesh generation environment. Unnecessary geometry parts for aerodynamic simulation are removed and geometry is cleaned at this stage. Then, outer boundaries of the computational domain and CFD mesh are generated for the external aerodynamic simulation. The outer boundaries of the computational domain can also be called as the walls of a virtual wind tunnel. CFD mesh of the CAD geometry could be consist of tetrahedral, hexahedral or mixed type of elements. The type of the mesh is usually determined by the complexity of the problem and the mesh generation software. The next step in the aerodynamic simulation procedure is the solution step that generates flow field variables in the computational domain. Reynolds averaged Navier- Stokes solvers are the most common type of solvers for ground vehicle applications. After the flow field variables are obtained, the velocity, pressure and force data are analyzed, and if there is an optimization procedure involved, the force variables are plugged into the optimization algorithm[3].

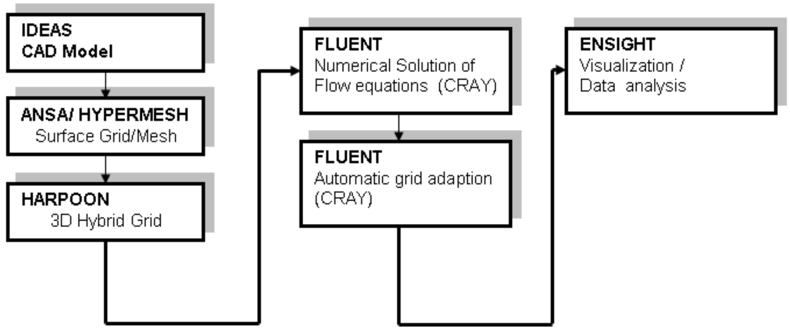


Figure 1: Ford-Otosan CFD Process

Aerodynamic calculations are conducted in a computational environment, they are very compatible with optimization.

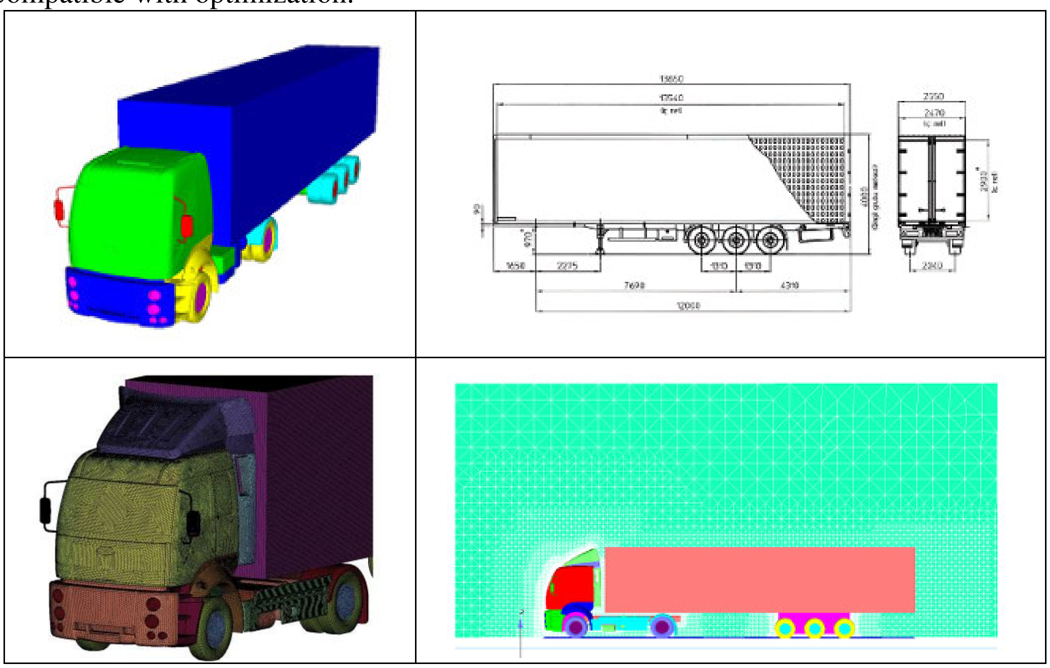


Figure 2: CFD Process from importing CAD data to Volume Mesh

Instead of simulating a real truck model has an engine and underbody components, a simplified truck model was used to lower the grid generation complexity and findings were verified with the full truck&trailer model. Even with a simplfied body model, however, many details of a real truck-trailer configuration remained.

The Volume mesh is a conformal mesh, which means that transitions between two sizes of hexahedra are maintained by pyramids. For current versions of Fluent, conformal meshes are computed much faster than non-conformal meshes. The final mesh consists of approximately 18 million cells and takes less than one hours to generate on a Dell T7400 model computer. Fluent commercial package programme has been used for all CFD applications at Ford Otosan during the past few years. It has been found to be very user friendly and versatile with a large range of usage areas. A pressure based coupled algorithm is used which gives a robust and efficient solution for steady-state flows. A second-order upwind scheme was used for the convection terms in the momentum equations while a first order upwind scheme was used for the turbulent properties. For turbulence modelling, the realizable k-ε model is used with standard wall functions.

2.1 CFD Formulations: Governing Equations & Turbulence Model

The governing equations for the turbulent incompressible flow encountered in this study are the steady-state Reynolds-averaged Navier-Stokes (RANS) equations for the conservation of mass and momentum. In the literature [4] they are presented in the following forms:

Continuity:

$$\frac{\partial}{\partial x_i}\left(\overline{\rho u_i}\right)=0 \tag{1}$$

Momentum:

$$\frac{\partial}{\partial x_j}\left(\overline{\rho u_i u_j}\right)=-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j}+\frac{\partial \overline{u}_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial \overline{u}_l}{\partial x_l}\right)\right]+\frac{\partial}{\partial x_j}\left(-\overline{\rho u_i' u_j'}\right) \tag{2}$$

In Equations (1) and (2), $\overline{\rho}$ is mean density, $\overline{p}$ is mean pressure, μ is the molecular viscosity and $-\overline{\rho u_i' u_j'}$ is the Reynolds stresses. To correctly account for turbulence, Reynolds stresses are modelled in order to achieve closure of Equation (2). The method of modelling employed utilises the Boussinesq hypothesis to relate the Reynolds stresses to the mean velocity gradients within the flow. Therefore the Reynolds stresses are given by:

$$-\overline{\rho u_i' u_j'}=\mu_t\left(\frac{\partial \overline{u}_i}{\partial x_j}+\frac{\partial \overline{u}_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial \overline{u}_i}{\partial x_i}\right)\delta_{ij} \tag{3}$$

In Equation (3), μ *t* is the turbulent (or eddy) viscosity and *k* is the turbulent kinetic energy. For two-equation turbulence models such as the *k*-ε, the turbulent viscosity is computed through the solution of two additional transport equations for the turbulent kinetic energy, and either the turbulence dissipation rate, ε.

Ford-Otosan TASE Team prefered to use the Realisable *k-ε* turbulence model for the all solutions. The Realisable *k-ε* turbulence model differs from the Standard *k-ε* model in two important ways. Firstly it contains a new formulation for the turbulent viscosity, and secondly, a new transport equation for *ε* has been derived from an exact equation for the transport of the mean-square vorticity function. However, In the Realisable *k*-ε turbulence model, the transport equation for *k* is identical to that of the Standard *k*-ε model[5] .

Realisible k-ε turbulence model

The Realisible *k-ε* turbulence model is presently the most widely applied turbulence model to practical engineering flows as it is robust, economical and provides reasonable accuracy for a wide range of flows. The transport equation for *k* is physically correct, however the transport equation for *ε* is heavily modelled . The modelled transport equations for *k* and ε, for steady-state and incompressible flow, are given in the literature [4] as:

$$\frac{\partial}{\partial x_i}\left(\rho k \bar{u}_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + G_k - Y_k \tag{4}$$

$$\frac{\partial}{\partial x_i}\left(\rho \varepsilon \bar{u}_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + G_\varepsilon - Y_\varepsilon \tag{5}$$

In Equations (4) and (5), σ *k* and σε are the turbulent Prandtl numbers for *k* and ε respectively. The production of turbulence kinetic energy, G_k, is approximated in a manner consistent with the Boussinesq hypothesis by:

$$G_k = \mu_t S^2 \tag{6}$$

S is the modulus of the mean rate-of-strain tensor, defined by:

$$S \equiv \sqrt{2S_{ij}S_{ij}}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j}\right) \tag{7}$$

The dissipation of this turbulence kinetic energy, Y_k, is defined by:

$$Y_k = \rho\varepsilon \tag{8}$$

$$G_\varepsilon = \rho C_1 S \varepsilon \tag{9}$$

$$Y_\varepsilon = \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} \tag{10}$$

The constants applied in the Realisable *k*-ε turbulence model are equal to:

$$\sigma_k = 1.0,\ \sigma_\varepsilon = 1.2,\ C_2 = 1.9 \tag{11}$$

3 ROOF SPOILER AND SIDE EXTENDER OPTIMIZATION STUDIES

3.1 Optimum *y-plane* Spoiler Angle Definition

Current spoiler C_D value has been determined the initially analyse as the reference value. After then possible spoiler angles have been investigated for iteration steps. Flow Velocity is 22 m/s (80 km/h)

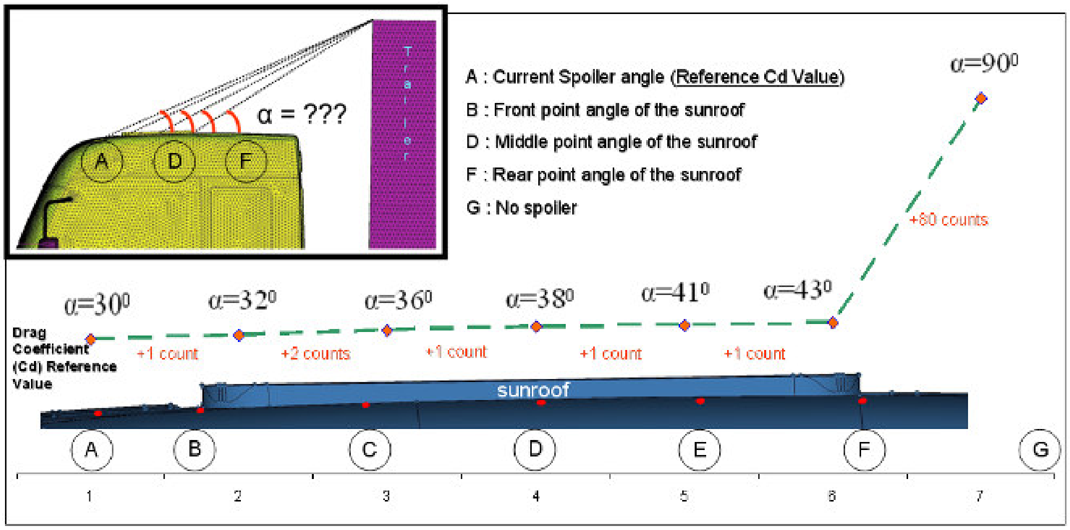


Figure 3: CD value gets worse with changing spoiler angle

As seen from Figure 3, the higher slope angle, as expected, causes the higher **total drag** coefficient. Slope angle was carried step by step from point A to point F and up to +6 counts maximum degradation was observed on CD value.

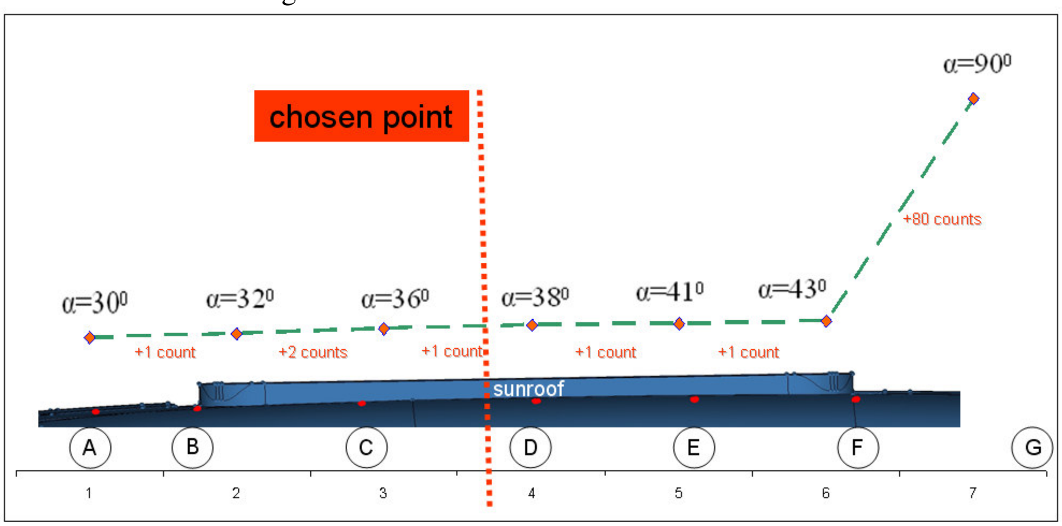


Figure 4: Leading edge starting point for analyze

Ford-Otosan TASE group chose a point between point C and point D in order to shorten the spoiler geometry and make efficient sunroof functionality.

3.2 Determination of Optimum Boundaries of Spoiler

Ford Otosan TASE Group analyzed the base truck & trailer case which do not have any spoiler and extender geometry in order to determine the flow patterns and velocity contours between truck and trailer. Optimum spoiler geometry points have been determined on flow patterns by concentrating on the several –x & –y plane velocity values comparing with inlet reference velocity.

+y boundaries

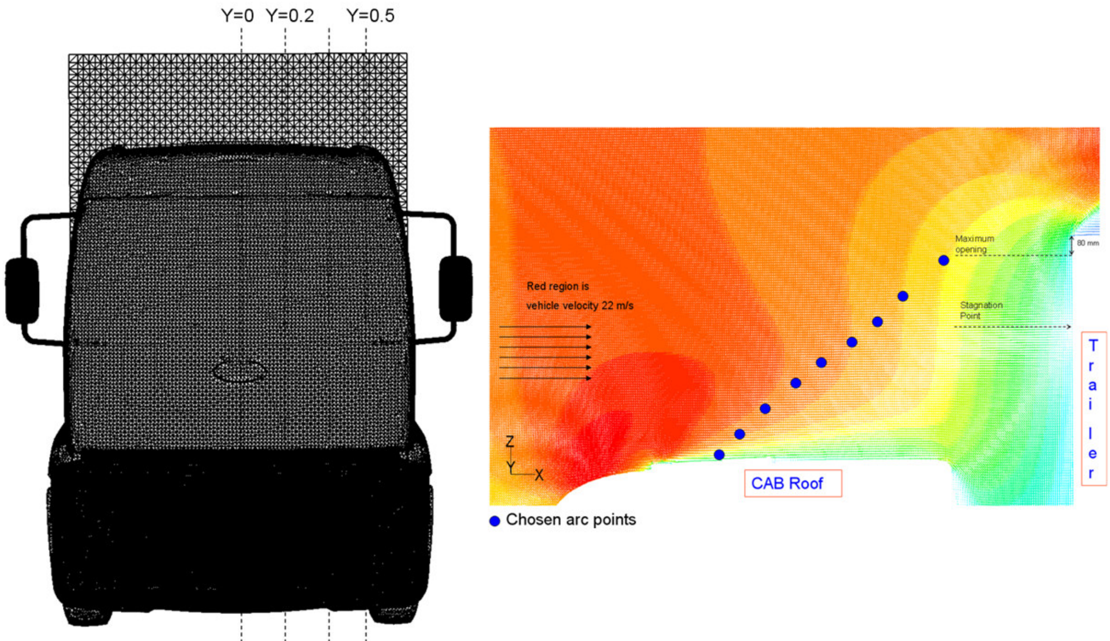


Figure 5: +y Cutting Planes with CFD result

+x boundaries

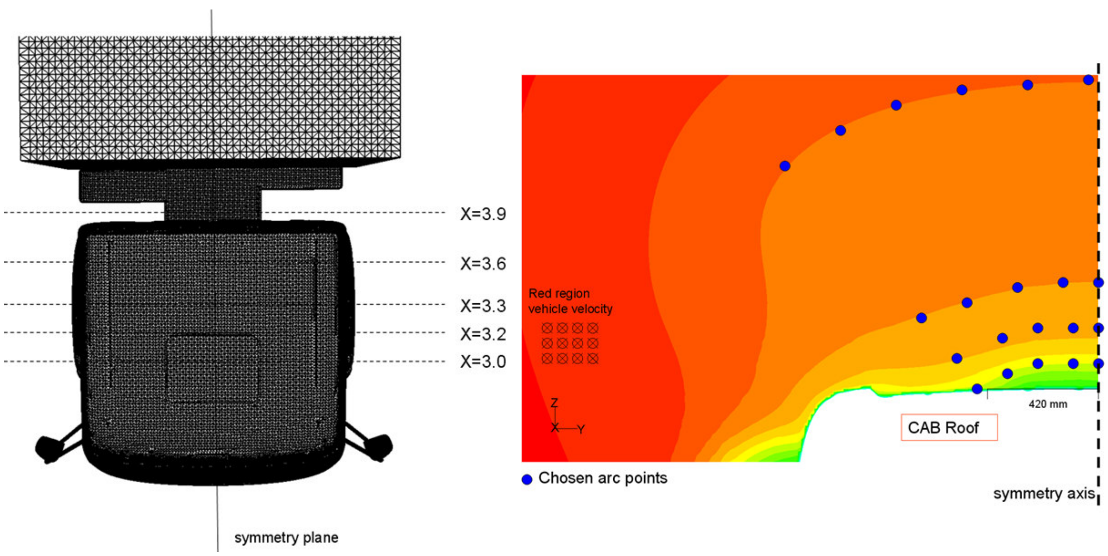


Figure 6: +y Cutting Planes with CFD result

+z plane; has been appeared spontaneously, after well chosen +x & +y plane points.

3.3 Determination of Optimum Boundaries of Extender

To determine new extender sizes Truck & Trailer geometry CFD results have been investigated in the cutting +z sections.

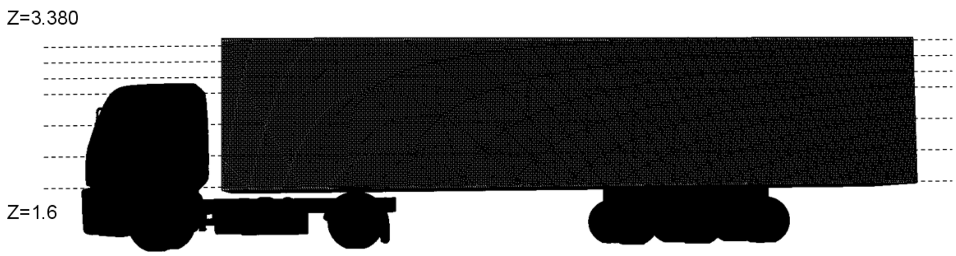


Figure 7 : Ford Heavy-Duty Truck w/o spoiler& extender

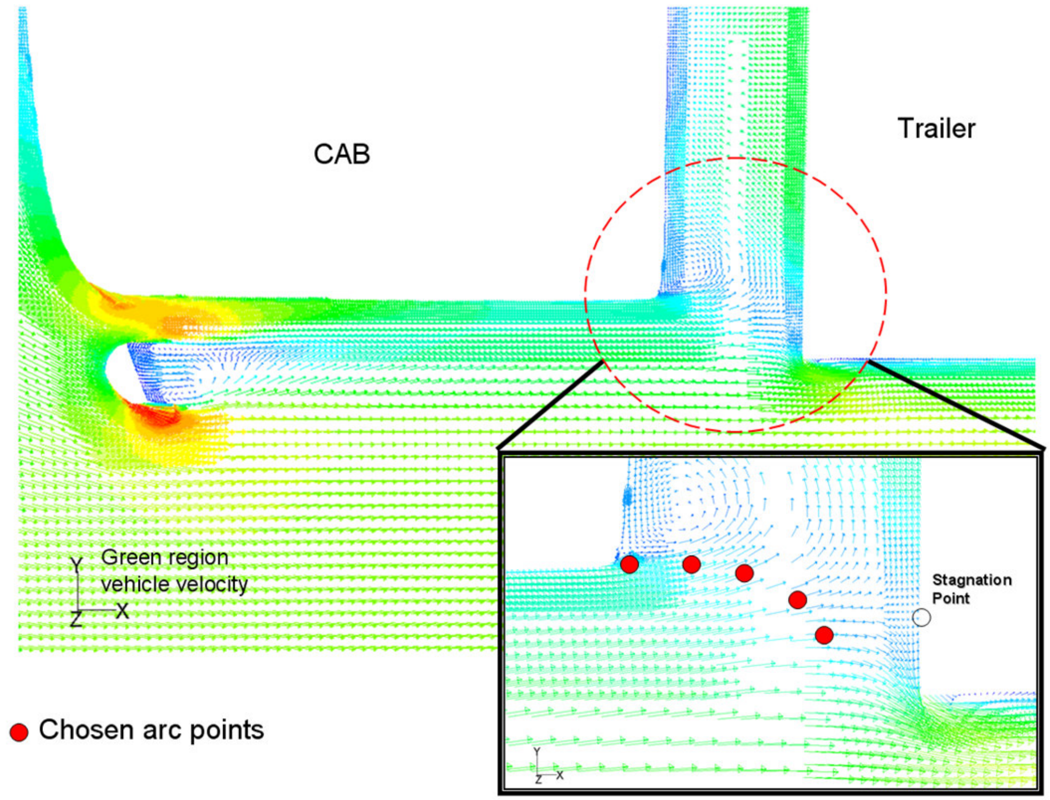


Figure 8: +z section velocity vectors for choosing extender sizes

After all these tracking point determination through the flow patterns a spoiler & extender shape proposal have been prepared by TASE& Design Studio of Ford Otosan.

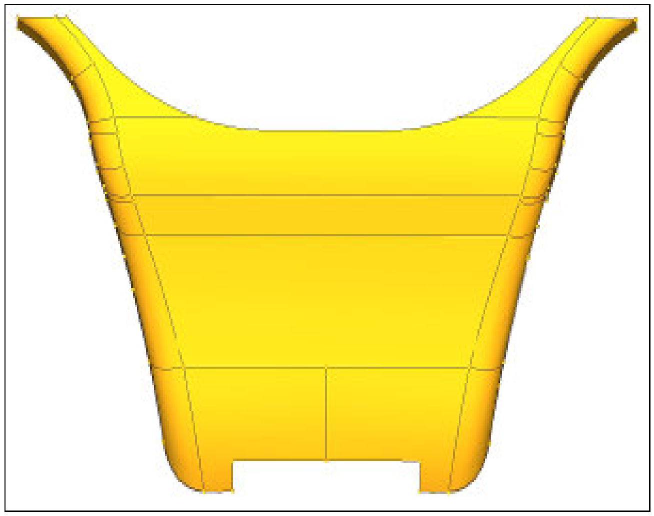


Figure 9 : New Designed Spoiler & Extender Shape

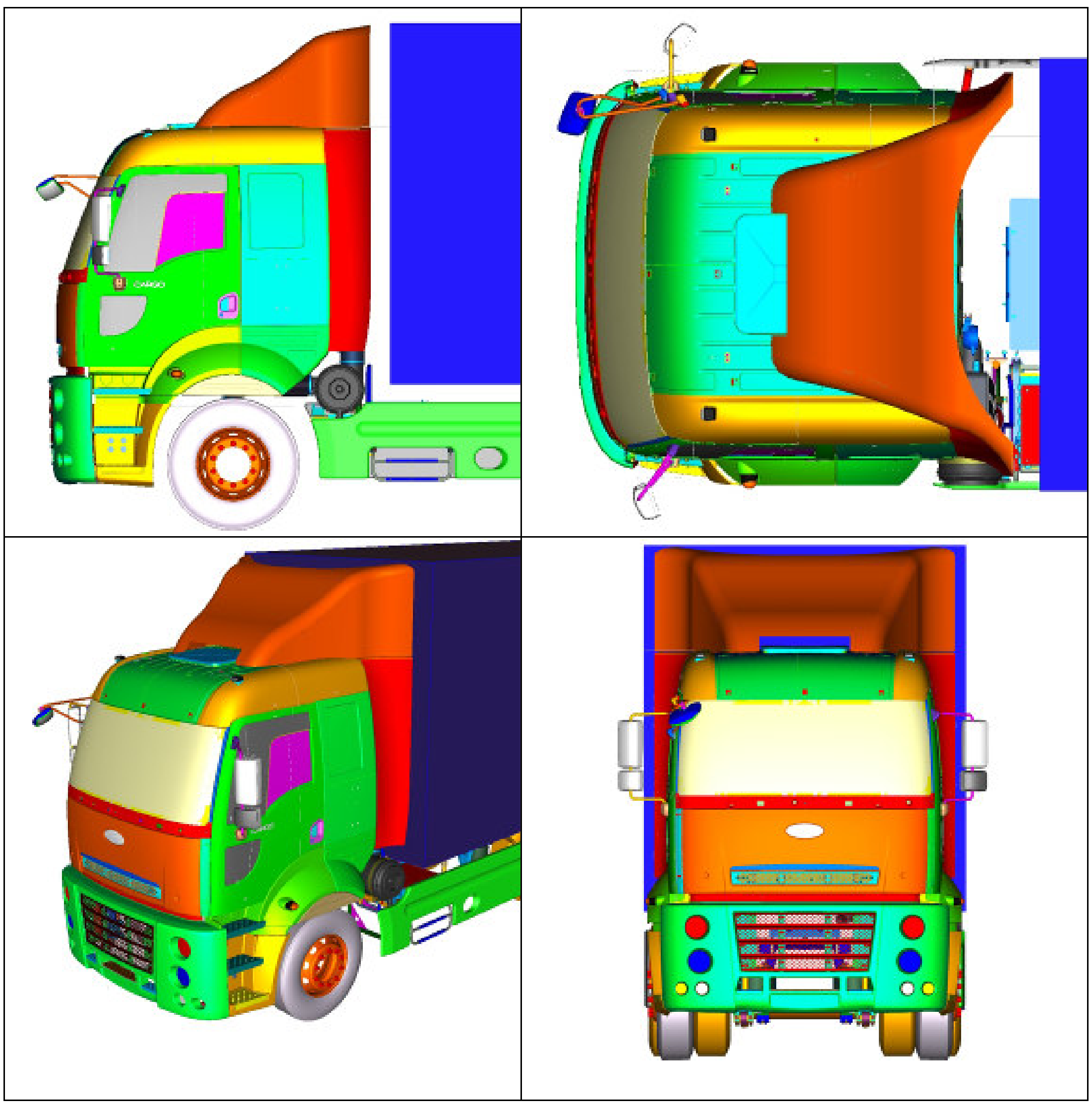


Figure 10: New Designed Spoiler & Extender Shape

4 CFD RESULTS OF NEW DESIGNED SPOILER & EXTENDER

Ford Otosan TASE Group analyzed new design proposal & current spoiler shape aerodynamically. Its CFD results have been compared with Current Spoiler & Extender.

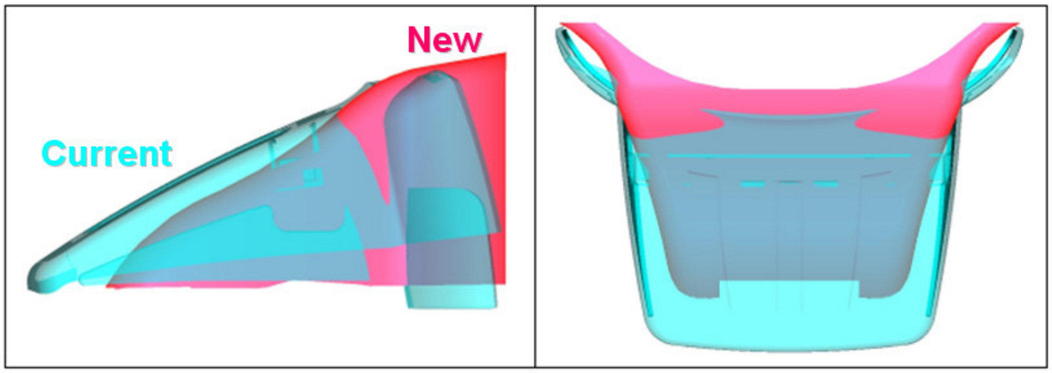


Figure 11: Comparison of Current & New Designed Spoiler &Extender Shapes

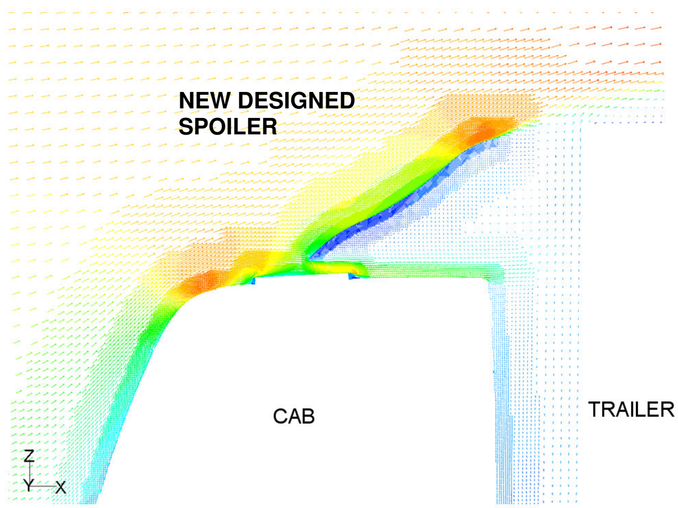


Figure 12: New Designed Spoiler & Extender CFD results

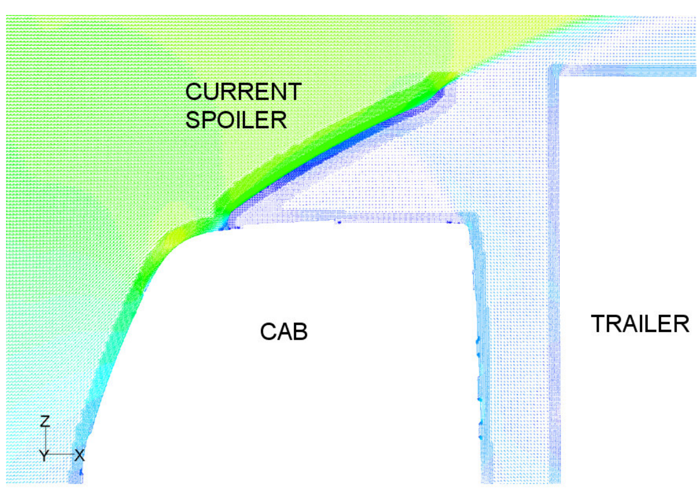


Figure 13: Current Spoiler Extender CFD results

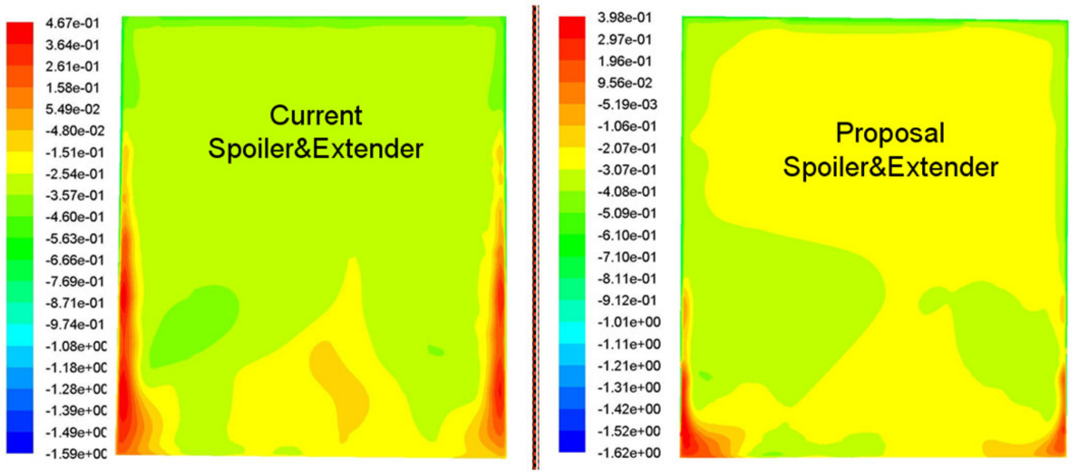


Figure 14: Comparison of Pressure Distrubution of Trailer front faces

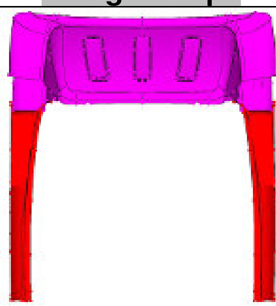
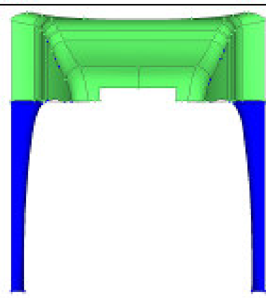
Design Shape	***Total Cd Deviation***
	current spoiler & extender (reference value)
	Same level C_d with current spoiler& extender

Figure 15: Last Cd Values

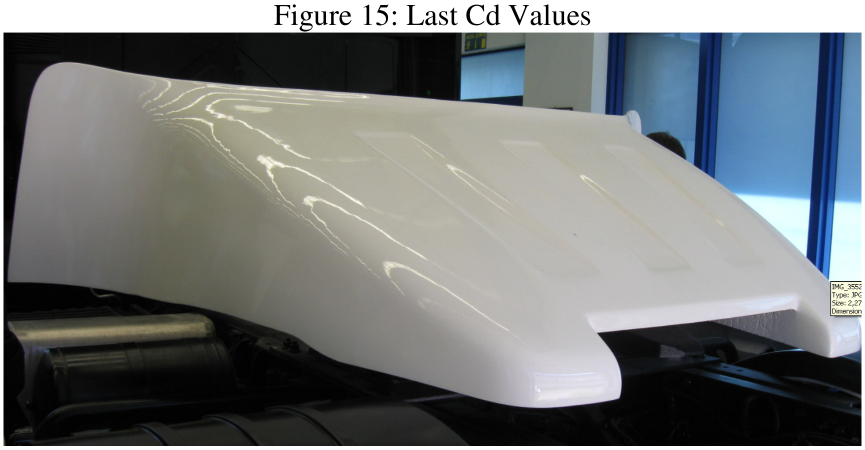


Figure 16: A Prepared Prototype Spoiler (closed potion)

5 CONCLUSIONS

As a result of this study;

- The current spoiler & extender were aerodynamically optimized with same level improvement on total drag coefficient of Ford Heavy Duty Truck.
- The Spoiler shape became smaller to gain a cost reduction.
- The spoiler beginning point has been changed a being behind of sunroof functioning for customer satisfaction.

REFERENCES

[1] S. Wolf-Henry Hucho, Aerodynamics of Road Vehicles, p.771, SAE, 1998

[2] Edwin *J. Saltzman,* Robert *R. Meyer, Jr* “A Reassessment of Heavy-Duty Truck Aerodynamic Design Features and Priorities” NASA/TP-1999-206574

[3] I. Bayraktar, T. Bayraktar, “Guidelines for CFD Simulations of Ground Vehicle Aerodynamics” SAE Paper, 2006-01-3544

[4] R. Fluent Inc. 2006, *.FLUENT 6.0 User.s Guide*.

[5] Oktay Baysal, Ilhan Bayraktar “Computational Simulations for the External Aerodynamics of Heavy Trucks” SAE Paper, 2000-01-3501

[6] W.D. Pointer, Tanju Sofu and D.Weber “Development of Guidelines for the Use of Commercial CFD in Tractor-Trailer Aerodynamic Design” SAE Paper, 2005-01-3513

[7] Ron Schoon, Fongloon Peter Pan “ Practical Devices for Heavy Truck Aerodynamic Drag Reduction” SAE Paper, 2007-01-1781

[8] Subrata Roy, Pradeep Srinivasan “ External Flow Analysis of a Truck for Drag Reduction” SAE Paper, 2000-01-3500

[9] Steven Perzon, Johan Janson, Lennart Höglin “On Comparision Between CFD Methods and Wind Tunnel on a Bluff Body ” SAE Paper, 1999-01-0805