CFD ANALYSIS OF VORTEX GENERATOR EFFECTS ON AERODYNAMIC PERFORMANCE OF HATCHBACK VEHICLE MODEL

Haidar A. Amer[1], Mironeasa Costel[2]
[1]Ștefan cel Mare University of Suceava-Romania, amerhaidar85@gmail.com
[2]Ștefan cel Mare University of Suceava-Romania, costel@fim.usv.ro

Abstract: *This article tries to figure out the effect of adding a vortex generator part to one hatchback vehicle on the aerodynamic performance and car's reliability by using CFD method and FEA technique in order to simulate the airflow streamlines and drag forces facing the body. In addition this paper suggests a new level of design optimization of the hatchback vehicle by controlling the dimensions and position of the Vortex Generator as a future work.*

Keywords: *Finite Element Analysis, CFD, Numerical simulations, aerodynamic performance, Vortex Generator (V.G)*

1. Introduction

New generation of industry is quality and environment oriented and the engineers consider the design optimization as a corner stone in this industry. Design optimization leads the designers and engineers to improve the concept and saving time, costs and quality as much as possible.

This paper discusses the importance of the aerodynamic performance in design optimization and the effect of vortex generator on this performance using the numerical simulation techniques. Many numerical tools have been used in this paper to visualize the airflow streamlines around the vehicle's body and pressure distribution on it, in order to provide a clear image about analyzing the numerical results.

After analyzing the effects of vortex generator on aerodynamic performance this paper concludes the advantages/disadvantages of this part and establishes for a future work related to the shape, dimensions and position of the vortex generator and how designer can continuously use it to improve the aerodynamic efficiency of the car.

Basic techniques have been used in this paper are finite element analysis (FEA) and computational fluid dynamics (CFD).

2. Computational Fluid dynamics (CFD)

2.1 CFD definition

Computational fluid dynamics (CFD) is the science of predicting fluid (liquids and gases) flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process [1].

We use CFD method in this research to simulate the air flow around the Vehicle's body and inside the engine compartments in order to validate the efficiency of the aerodynamic design and the effectively of cooling system which outfitted in the vehicle.

We can define the basic steps for a CFD simulation process as following in Figure 1, so we start by create the CAD geometry and surrounding environment then we move to analyze the physics of the problem in order to define the governing equations which are controlling our problem. Defining the physical model of the simulation is an important task to understand all the boundary conditions, initial conditions, and parameters enter the simulation, type of materials, type of flow and other key points which lead the engineer to accurate correct CFD simulation [1].
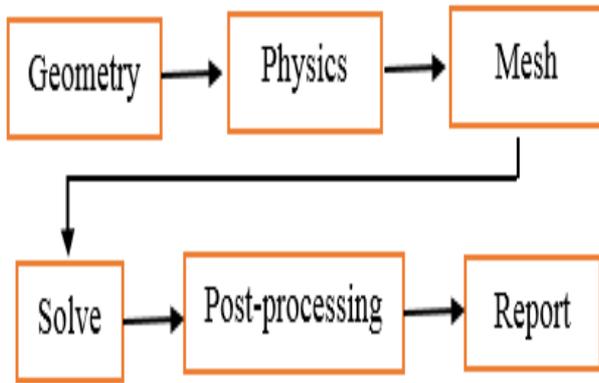
Figure 1: *CFD basic process*

Third step is create the mesh and choose the suitable type of elements based on our case then going forward to solve the numerical equations. Solving step can be done using a personal code to read the elements and create the matrix of solution, or a commercial code can be used to do that with special attention in adding the boundary conditions numerical scheme. Post-processing and reports are the last steps to display the simulation results and identify how we can read the differences and decide which design most efficient for our case.

2.2 Lattice Boltzmann Method

In CFD simulations the governing equations can be established using the conventional method [2] (Navier-Stockes Equations) or Lattice based method [6] which are used in this paper to solve the numerical model and estimate the results of fluid particles.

Lattice Boltzmann equations have been used to discrete the formulation of kinetic theory with no further approximation and the numerical integration of nodal values for each parameters has done to solve the lattice equations and apply the kinetic boundary conditions [3].

These methods are positioned in high mathematics field and our interest is how to use it in the numerical code without going deep in the mathematical explanation.

LBM is based on microscopic models and equations of mesoscopic kinetic. It considers that particles have only a finite number of discrete velocity values, and we can control the velocity of every particles on the lattice by two equations: one equation for velocity and one equation for distribution function which describes the probability of streaming in one particular direction, [3].

2.3. An Overview of Finite Element Analysis

Numerical Analysis is the method to use numerical approximation for equations in a mathematical analysis. The main idea of numerical analysis is to study the physical model of our problem and obtain a suitable mathematical model that describes the problem and can control the variables and push forward for solution. Mathematical model means use approximate functions to change differential equations to integral algebraic ones, so we can use the power of computers to solve the new matrix of equations [4].

FEM is a computational method to divide the geometry (CAD model) into small finite-sized elements in simplest possible shape, the name of this network of elements is Mesh and we call the contacts points between elements Nodes as Figure 2 displays.
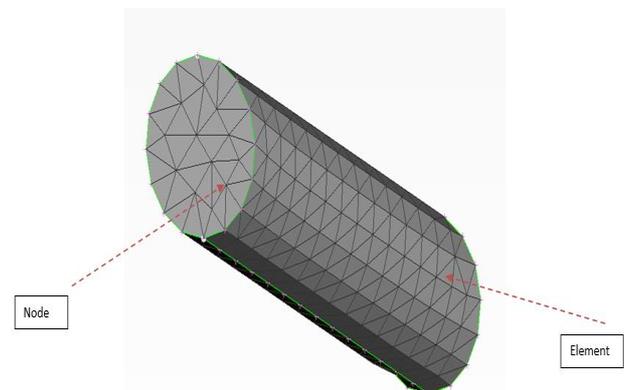

Figure 2: *Mesh elements and nods for one CAD geometry using ANSA pre-processor*

This method is the most difficult to apply however it has the best accuracy especially in complex CAD models, because of that we will

concentrate on this method and its basics and applications.

For applying FEM for engineering model we need first to specific the physical governing equations which control the model, then to obtain mathematical model we need to change PDE to integral equations using approximate functions (linear, nonlinear, quadratic polynomial, etc.); [See 3]. Using this functions and creating the Mesh which contains elements and nodes with specific number of Degree of Freedom (DOF[s]) lead us to end up with a large sparse matrix equations system that can be solved by power calculation of computers [4].

3. Aerodynamic performance and factors
3.1. Aerodynamic Importance

Aerodynamic performance means the fluency of the vehicle during the motion in the airflow, in other meaning moving the body in less drag force and less required energy. Aerodynamic study in vehicle industry is one of the main fields which must be analyzed during the design and production because it has effects on almost all other sides of vehicle's performance and quality [5] like we note in Figure 3.
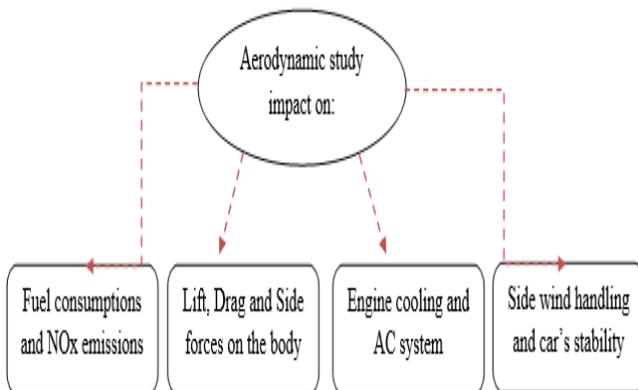

Figure 3: *Aerodynamics impact in auto-vehicle design*

Based on this big impact of aerodynamic the engineers always concentrate on the body shape and surface combination to have the most efficient model for the vehicle. In addition aerodynamics helps the designer to rethink about new concepts to reduce NOx and CO emissions and save fuel respecting to environment international standards [6, 7].

3.2. Aerodynamic Coefficients

The main aerodynamic factors applied on auto-vehicle's model are related to forces which face the car during moving in the air flow, however in the real environment or virtual one. To demonstrate the effect of aerodynamic factors on vehicles we have to mention the drag and lift force affected on the body in the motion and controlling the performance of this body. The forces which faces the car's motion are the driveline friction, air resistance, acceleration change of the car and tire rolling resistance, in fact all these force are connected to the speed of the vehicle and can inverse fastly by changing this speed as we can see in Fig. 4 [8].
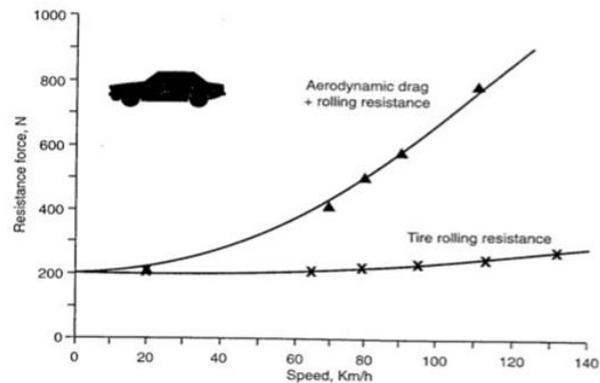

Figure 4: *Increasing the the aerodynamic drag and resistance according Vehicle's speed [9]*

Aerodynamic drag and resistance as we note from Figure 4 are increasing sharply with a small change of speed, as a result this drag force has big effect on aerodynamic performance and should be examined carefully. To express this force in mathematical method, engineers devised a non-dimensional number called the Drag Coefficient (CD).

Drag coefficient quantifies the aerodynamic sleekness of the vehicle body design and its definition:

$$C_D = \frac{D}{0.5 \cdot \rho \cdot U^2 \cdot S} \qquad (1)$$

where: D - Drag force, U - speed of the vehicle, S - Frontal area, ρ - Air density

Drag coefficient should the key factor in aerodynamic analysis because it is interfering directly in power calculation, fuel consumption, stability and performance as we note from its mathematical form.

Apparently we can say that the body shape, outer surface, curves, wings, air shutters, vortex generator and any additional surfaces [10] may be added to the vehicle's design, will change directly the frontal area (S) of our model and as result, the aerodynamic coefficient (C_D) and needed power (from the engine) will be changed in positive direction or negative one, based on the mathematical formulas above.

The frontal area (S) which presents in the aerodynamic coefficient formula differenitates from one vehicle's model to other, and maybe it considerd as a cross sectional area, lateral area or total area, that's why the designers adopted new metric coefficient to compare the aerodynamic effeciency of various automobiles which is multiplication of Drag coefficient (C_D) with frontal area (S = A). The formula for aerodynamic coefficient ($C_D.A$) is:

$$C_{D.}A = \frac{D}{0.5 \cdot \rho \cdot U^2} \qquad (2)$$

The Aerodynamic coefficient ($C_D.A$ in English or SCx in French) should the fundamental value to compare and evaluate the aero-effeciency in the researches and markets, so we calculate it our simulations and use it to compare the different designs of our model. Whenever we have less $C_D.A$ that means that we have less drag area and better aerodynamic performance of the design.

3.3 Aerodynamic Factors

Its useful also to put the light on relation between Vehicle's shape and resulting Drag and Lift coefficients. As we see in Figure 5 it appears that flow above the vehicle moves faster than below it, and if it follows the curved shape of the vehicle, we call it attached flow. However, at the backside of the vehicle, the airflow cannot follow the sharp downward turn and so this region is called "separated flow", where we have separation area of air streamlines.
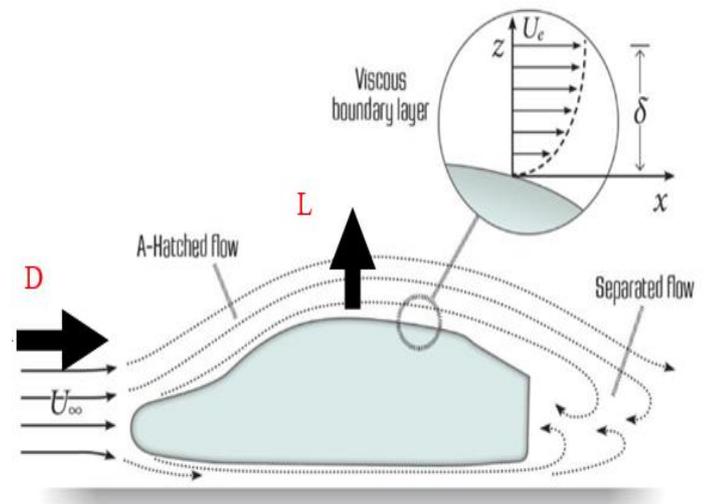

Figure 5: *Schematic description of the airflow around the vehicle's body and how creating Drag and Lift [9]*

By respecting to Bernoulli theory [11] (Daniel Bernoulli 1700-1782) which said that at higher speeds regions the pressure is low. Therefore, the pressure on the upper surface of the autmobile's body will be lower than the pressure under the body, resulting a force from high pressure are (under the body) to low pressure area (upper the body), we call it the lift (L).

Also the airflow will apply a high pressure on the front side of the vehicle's body, which is higher than the pressure created behind the body because of flow separation, so a force will be created from high pressure area (the front side) to low pressure area (backside) we call it drag force (D).

One must remember that in a very thin layer (called the boundary layer—shown by δ) near the vehicle surface there is a so-called "skin friction" which also adds to the drag coefficient (but its contribution in automobiles to CD is usually very small).

3.4 Points to improve the Aerodynamic

Taking in consideration the big impact of aerodynamic behavior of the car on all other sides, we selected our model in this paper to analyze this behavior by changing the design of one sensitive part in the body.

All high velocity areas on the car's body considered as sensitive parts and should have a

special care and deeper study like we see Figure 6 [10]. Because of too much details will have effect on vehicle's aerodynamic which we cannot study them in this paper, we have chosen to analyze and study the flow on Aero-angle area of the model like we see in Figure 6 and the effect of adding a vortex generator in the aero-angle area at the rear side of the vehicle.

Figure 6. *High sensitive areas of aerodynamic behavior of the vehicle*

4. Vehicle's Model and Properties

To analyze the effect of vortex generator one hatchback vehicle's model has been used. This model is very common in the global market for a wide range of clients and is produced in large numbers alomost in every vehicle industry company.

Our model is taken from a basic concept ((DACIA-Sandero-B52-2013) with making some changes in the body and later adding the V.G to the aer-angle area. More information and details about the model and dimensions can be seen [12]. In this paper we care about the aerodynamic performance by adding the V.G that's why the engine's properties or other compartments didn't discussed.

4.1 Initial Model and New V.G Model

Figure 7 shows the basic or initial model of analyzed vehicle and the aero-angle area which is our area of interest. This model will be our reference to compare and study the design optimization process. Numerical simulation has been done for the initial design at 150 Km/hr and the aerodynamic coefficients calculated to estimate the aerodynamic performance of the car. then a V.G part has been added to the aero-

angle area of the body as we can see in Figure 8. V.G part has designed with specific dimensions (70×45×25 mm).

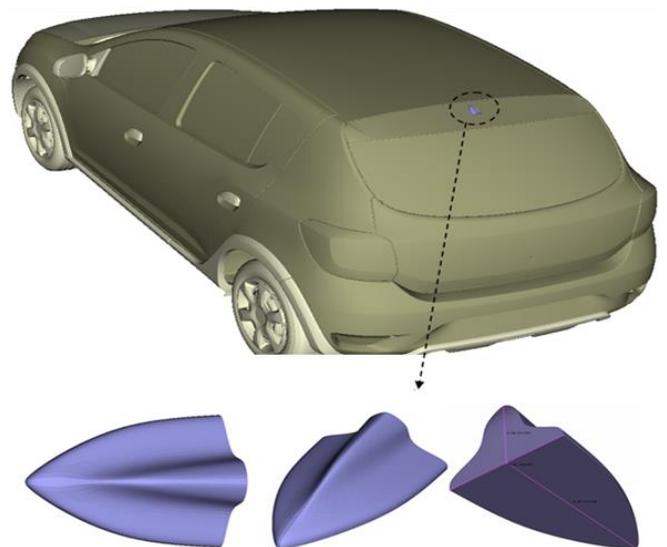**Figure 7**: *The aero-angle area for the Initial Model of the vehicle*

Figure 8: *New Model with V.G combined at the aero-angle area*

4.2 V.G Concept

Vortex generators are commonly used in aviation industry [13] to prevent the flow separation about the aircraft wing and reduce the drag. Engineers these days have started to use this part in vehicle's design in order to achieve the same goal.

Vortex Generators reduce the overall aerodynamic drag force at the backside of the car and rear-end airflow separation [14, 15], so as result they delay the separation area of air and allow faster clean air at the rear of the vehicle. As we have said before this effect helps to increase the engine performance, top speed and reduce the fuel consumption.

V.G has a sample geometry with low cost and easy installation compared with air-wings (spoilers) that's why it provides a good design solution for engineers.

Since there are no theoritical rules or laws to design and position the V.G the designer engineer uses the numerical techniques to optimize the design and identify which place and dimensions are the most efficeint for aerodynamic performance. However identfying the suitable position to outfit the V.G and its good dimensions related to vehicle's dimensions will be the subject of future work followed this paper.

V.G has been outfitted at the mid-section of the car and designed with 2% ratio from the general dimensions of the vehicle.

5. Numerical Results and Discussion

Numerical calculations have been done using RTR servers and software after their permission. The geometry and mesh made using ANSA pre-processor software and the numerical soultions performed by LaBs solver. Meanwhile the post-processing level done by using Paraview open source software [15].

Total work duration was 20 working hours for designing and meshing, 55 hours for numerical solving on the server and 9 hours for extracting the results and post-processing.

5.1. Aerodynamic Coefficient Values

By calculating the aerodynamic coefficients to the initial model and V.G model we can note that the V.G has decreased the $C_D.A$ and C_D values of the vehicle as we see in Table. 1.

Model of design/Velocity	$(C_D.A)$ @ 150 Km/hr	(C_D) @ 150 Km/hr
Initial Model	0.672 m^2	0.283 m^2
V.G Model	0.656 m^2	0.276 m^2

Table. 1 Comparison the values of aerosynamic coefficients in our tow cases

Its clear that the V.G part has an impact on the aerodynamic coeffiecients, in other meaning has a noticeable effects on Drag force and energy loss created against the vehicle's

motion. V.G has reduced ($C_D.A$) about 3% and this ratio is valuable for design engineers and leaves an effect fuel consumbtion especially in high velocities driving (above 100 Km/hr).

5.1. Visual Results and comparison

In order to explain the effects of V.G on the airflow around the body we have to use many visual tools. Fig. 9 illustrates the atreamlines of the air at mis-section (Y=0) of the vehicle's body and we can notice how the V.G changes the streamlines at thr rear-side of the body and helps to create an air vortix little far away from backside glass. The streamlines are colored depend on the velocity so high velocity areas have a red color while low velocity areas have blue one.
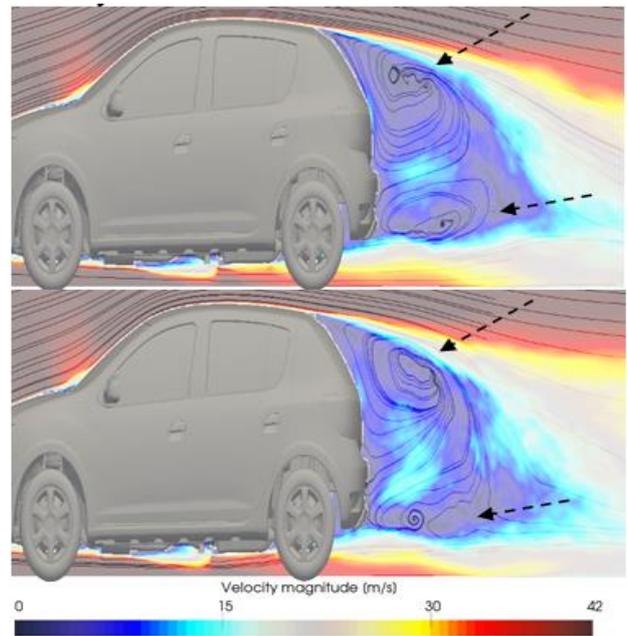

Figure 9: *Airflow streamlines at mid-section of the body for Intial and V.G models respectively*

Fig. 10 shows the pressure distribution on the body for our two cases and we can see the difference in the pressure at the rear-side area cuased by G.V.

Incresing the pressure value at the back-side of the car will reduce the pressure gradiant between the front-side and rear-side and as a result the drag force faces the vehicle will be less.

Also we can note a low pressure area on the surfaces of the V.G (blue color) which means a

high velocity area for the airflow around this part, (Bernoulli's Principle) [11], so the air will have enough kinetic energy to stream far away of the body before starts the separation phenom.
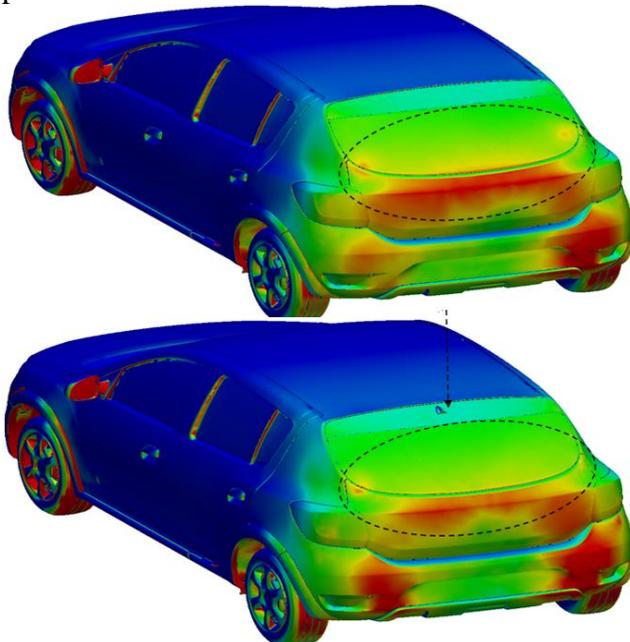

Figure 10: *Pressure distribution on the body for Intial and V.G models respectively*

Another important parameter to analyze is the sahpe of the vortex created behind the vehicle, however the vorticity region is taking energy from the car's energy.

To have an open vortex behind the car will have less seperation area and less energy loss, meanwhile the closed shape vortex causes more seperation and losses.

Figire 11 displays the shape and intensity of the vortex created behind the vehicle and we can note that the vortex in V.G model case is waker and has an open streamlines of airflow, so causes less drag force against the body.

Vortex generated in this case will cause a rotation of the airflow from high pressure area to low pressure one and this rotation decreases the drag force agoiant the body. We can note a 3D air streamlines created around the V.G at 150 Km/Hr in Figure 12.

Figure 12 shows how V.G keeps the fluency of airflow at rear-side of the vehicle as much as possible and delays the separation flow to appear because of friction drag and viscouse boundary layer [8].
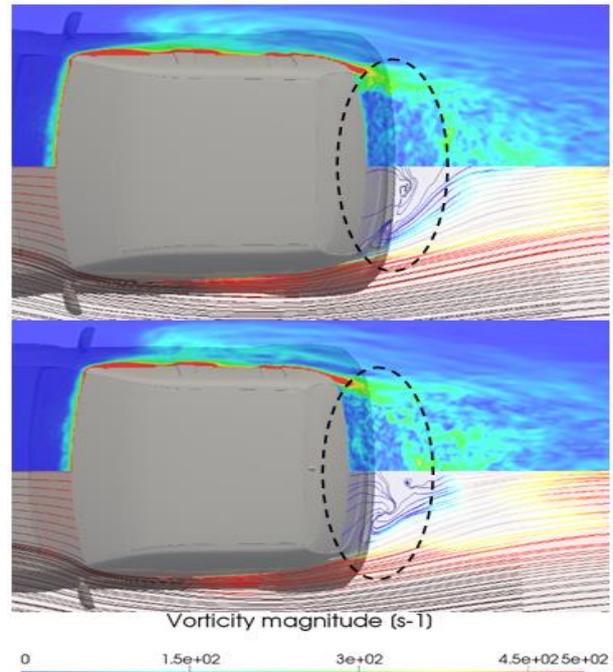

Figure 11: *Vorticity magnitude and shape at back-side of the car in Initial and V.G models respectively*
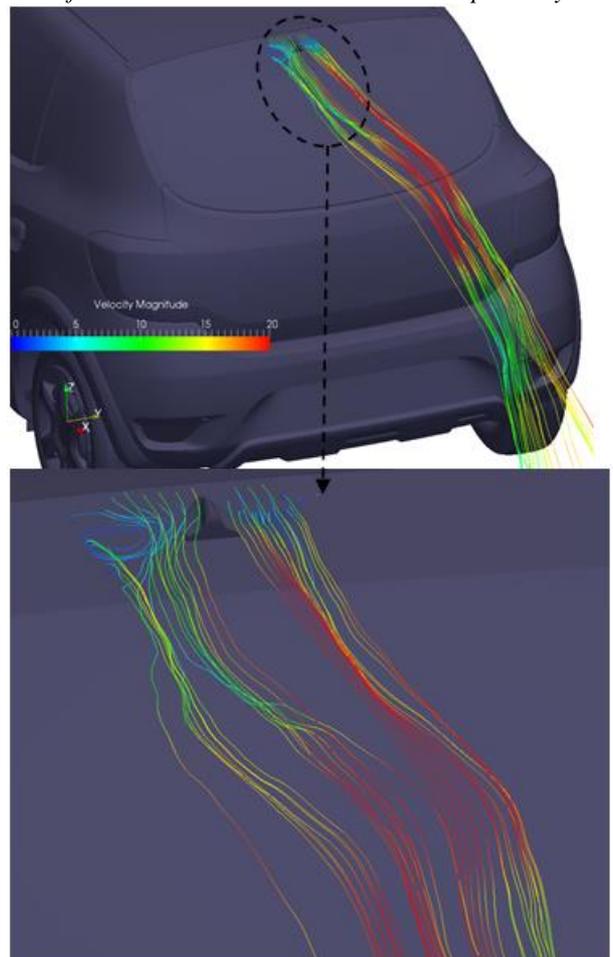

Figure 12: *3D airflow stream lines created by V.G*

13

Visual numerical tools can produce a clear image about V.G imapct and help us to analyze and optimize the design more and more in future work especially about V.G dimensions.

6. Conclusion

Vehicle's aerodynamic is a key concept in automobile industry and saving the energy resources and environment.

Numerical simulations provide powerful tools and techniques for engineers to analyze the complex cases in aerodynamics.

V.G has clear and noticeable effects on pressure distribution and airflow lines at the backside of the body as we have seen in this paper.

V.G has a positive impact on vehicle's aerodynamic by reducing the drag coefficient and aerodynamic coefficient (about 3%) which leads to less fuel consumption.

V.G provides a sample and low cost solution for engineers to improve the aerodynamic performance of the car and reduce the drag forces created.

More analysis is needed to optimize the design and the position of the V.G related to the vehicle's body, so we can know the impact of increasing the volume of V.G on aerodynamic performance. However the position of V.G near the aero-angle area or inside it also will be the subject of our future work in this domain.

References

[1] Zingg, D.W., 1999, Fundamentals of Computational Fluid Dynamics, University of Toronto Institute for Aerospace Studies, Canada

[2] Younsi, R., 2012, *Navier-Stokes Equations Properties, Description and Applications*, Nova Science Publisher, USA

[3] Bill Bao, Y., Meskas, J., Lattice Boltzmann Method for Fluid Simulations, Simon Fraser University, Canada, (2011)

[4] Haidar, A., Applications of Numerical Analysis in Simulation Engineering and Manufacturing Technologies, P- ISSN- 1224-029X, TEHNOMUS conference, Romania, (2017).

[5] Hucho, W.H., 1993, *Aerodynamics of Road Vehicles*, General Motor Research, Michigan University, USA

[6] Regulation (EC) No. 715/2007 of the European Parliament and of the Council, Official Journal of the European Union, L 171/1, June 29, 2007

[7] EU1-6 NOx emission standards: *https://www.rac.co.uk/drive/advice/emissions/euro-emissions-standards/* (15.05.2018)

[8] Katz, J., 2016, Automotive Aerodynamics, Wiley Publisher, UK

[9] *(https://mechanixillustrated.technicacuriosa.com/2017/03/04/an-introduction-to-automobile-aerodynamics/)*, (20.05.2018)

[10] Panchal, H., Kumar, K., *A Review on Aerodynamic Study of Vehicle Body using CFD*, Mandal Institute of Technology, India, (2014)

[11] *Bernoulli's Principle*, National Aeronautics and Space Administration, NASA, USA, 2010

[12] Dacia model and dimensions (15.05.2018): *https://www.dacia.ro/gama-dacia/modele/sandero-stepway.html*

[13] Selvakumar, K., Parammasivam, K.M., *Experimental Investigations on Optimization of Aerodynamic Characteristics in a Hatchback Model Car 3 using Vortex Generators*, APCWE journal, India, (2013)

[14] Mukkamala, Y., Shivam, S., Aerodynamic Drag Reduction of a Notchback Car Geometry by Delaying Flow Separation using Vortex Generators, IJERT journal, ISSN: 2278-0181, (2015)

[15] Srinivas, V.L., *Shape Optimization of a Car Body for Drag Reduction and to Increase Downforce*, Indian institute of Technology Kharagpur, India, (2016)

[16] ParaView Documents and Guide (26-05-2018): *https://www.paraview.org/documentation/*