Numerical Investigation of Drag Reduction Techniques in a Car Model

Z M Saleh and A H Ali
Mechanical Engineering, Engineering College, University of Baghdad, Karrada, Al-Jadriya, Iraq

Abstract. Reducing the drag force has become one of the most important concerns in the automotive industry. This study concentrated on reducing drag through use of some external modifications of passive flow control, such as vortex generators, rear under body diffuser slices and a rear wing spoiler. The study was performed at inlet velocity (V=10,20,30,40 m/s) which correspond to an incompressible car model length Reynolds numbers (Re=2.62×10^5, 5.23×10^5, 7.85×10^5 and 10.46×10^5), respectively and we studied their effect on the drag force. We also present a theoretical study finite volume method (FVM) of solving Reynolds-averaged Navier-Stokes equations (RANS) using a realizable k-epsilon (k-ε) turbulence model, conducted on a car, model KIA Pride, which is popular in Iraq and Iran. All computational analysis and modifications were carried out using the ANSYS Fluent 19 computational fluid dynamics (CFD) software and SOLIDWORKS 2018 modeller. The drag coefficient of the analysed car was found to be 0.34 and the results show that the drag can be reduced up to 1.73% using vortex generators, up to 3.05% using a rear wing spoiler and up to 2.47% using rear underbody diffuser slices modifications, whereas it may be reduced up to 3.8% using all previous modifications together.

1. Introduction
Undesirable aerodynamic characteristics are among the key obstacles facing a body that accelerates when it moves in the air. When a road car or a racing car moves, it consumes fuel when accelerating. The force of drag obstructs the movement of the vehicle and pulls it backwards, which adversely affects the speed of the vehicle and thus reduces the vehicle’s efficiency and balance. Approximately 50% to 60% of total fuel energy is lost solely in overcoming these undesirable aerodynamic characteristics.

In recent decades, car manufacturers and designers have been focusing on the reduction of fuel consumption and pollutant emissions by minimization of drag pressure, which is a predominant undesirable aerodynamic characteristic. The techniques of drag reduction are divided into two main categories: passive and active. Passive techniques require no energy input, while active techniques require energy disbursement and usually an advanced control-feedback mechanism. Factors that determine the use of either of these two technologies are cost, energy consumption, maintenance and car usage.

Many authors and studies have described different forms of drag and possible reasons behind them, along with some methods used to reduce it. For example, Ahmed and Chacko[1]. In their study of old fashioned car model, the assistance of CFD simulation was done by ANSYS Fluent 14 and modelling using SOLIDWORKS 2010. The airflow speed used was 20 ms⁻¹. That study aimed to reduce the
separation area of the flow and thus reduce the drag coefficient by reducing the air stagnation regions. They set up many devices for this purpose, such as vortex generators, front bonnet duct, ground effect, diffuser and rear wing. The results showed that drag reduction using vortex generators was 2%, ground effect and diffuser design yielded 3.5%, front bonnet duct 3.33%, and wing installation 5%.

Marklund [2] compared a sedan car and wagon-type car. The first model has a greater ability to reduce drag by use of a smooth floor and diffuser than was the case for the second model. The diffuser angles for enhancing drag reduction were 5º and 8º, respectively. The results showed reduction of the drag coefficient for the sedan car at 13%.

Moussa and colleagues [3] investigated drag reduction theoretically by using rear suction slits on a generic sports utility vehicle (SUV). The physical model was used with scale of 1/10th SUV without side mirrors. Geometric modelling and mathematical programming, provided by Fluent analysis system in ANSYS Workbench control volume, used a new methodology introduced for maximum reduction of aerodynamic drag to identify the size, location and magnitude of the suction velocity through the slit. The technique mixed CFD, automatic modelling of the suction slit, and a global search method optimized using orthogonal arrays (Taguchi method), which can be used to study multiple variables effects in the same time. It was found that a designed suction mechanism can reduce the drag by up to 9%.

Hassan et al. [4] worked a numerical study using under-body diffuser and exhaust gas redirection (EGR) to reduce drag on a race car. The results showed that drag coefficient decreased by 22.13% and was almost linear with the increase in slicing angle, in case of the under-body diffuser and when using exhaust redirection with 3000C, 2atm, and 45º, temperature, pressure, and angle of emission of exhaust gas, respectively. The results showed as the speed of exhaust gas decreases the value of drag coefficient decreases by 9.5%. Sharma and Bansal [5] obtained a CFD simulation for drag reduction using tail plates on a passenger car, made a numerical model to get the flow around the car and made comparison with and without tail plates placed at the end back of the car roof and at the tail bumper of the car. ANSYS-14.0 Fluent with the k-ε steady model was used for aerodynamic simulations. It was found that the tail plate could make a 3.87% reduction in coefficient of drag and also it enhanced the stability of a passenger car, in addition to fuel economy.

Selvaraju and colleagues [6] implemented a vortex generator as a passive device to minimize the aerodynamic forces on a sedan car body (Hyundai Elantra). It was added onto the rear part of the vehicle, then different locations and yaw angles were analysed to identify the optimal one to reduce the drag. A comparison was made between a base line and the vortex using CFD, and the researchers made a similar model using UNI-GRAPHICS software modelling and meshing the model forwarded to the ICEM-CFD software. The results clearly showed that the pressure was higher in the rear area without using vortex generators, and the drag reduced by 6% as compared to model without vortex generators.

Wahba [7] researched horizontal guide vanes as a device to reduce the drag in ground vehicles, by numerically investigation using CFD. The SUV and bus models were investigated with the assistance of CFD software, with various turbulence model such as k-ω and k-ε getting results at various guide vanes with angles 5º, 10º and 15º together. The total results found that with the lateral guide vanes at the rear end the reduction in drag coefficient increased to 18% for the two models. Furthermore, the results showed that the guide vanes with symmetric airfoil cross-sections gave better results than asymmetric airfoil in terms of reducing drag coefficient. It was estimated that, when vanes were used in the frontal area of the model, the drag coefficient was affected by up to 10%. So, the flap was important for reducing drag in a bus or SUV ground vehicle model, for driving or controlling safety and then improving fuel efficiency.

Sudin et al [8] made a review study of passive and active methods to reduce the drag on different types of vehicle. Add-on devices were studied numerically, active flow technique was effective in a wide
range of applications but the research concluded that both methods were able to reduce drag, and that a
degree of base pressure recovery can be achieved through careful shape optimization. They concluded
that the choice of mathematic model for flow, car model, velocity, design parameters and forcing
parameters needed further study, as it influences the results of drag force. Another review research
study, made by Sarkar and colleagues [9] on aerodynamic drag reduction and CFD analysis of vehicles
and various external design features added to the vehicles like (added vortex generators, rear screen,
fenders, etc.) and their results show these additions can be proved fruitful in high-end passenger
vehicles where cost is not a very significant factor.

Mukut et al[10] made a review of numerical papers studying vehicle drag reduction by active, passive
and combined methods. It was estimated that the drag reduction had been enhanced by as much as
20%, 21.2%, and 30% by use of the active, passive and combined control systems, respectively.

In the current investigation, the car model is computed using RANS simulation. The KIA Pride model
from 2003 is modelled using SOLIDWORKS 2018 software. The free stream and moving road
velocities are 10, 20, 30, 40 ms\(^{-1}\) which correspond to an incompressible car model length Reynolds
numbers 2.62×10\(^{5}\), 5.23×10\(^{5}\), 7.85×10\(^{5}\) and 10.46×10\(^{5}\), respectively. The commercial software
ANSYS FLUENT 19 solver was used for solving RANS equations using a finite volume method
(FVM). The realizable k-epsilon two-equation turbulence module was used.

The objective of this investigation was to enhance the aerodynamic characteristics of the car by
reducing the drag force as much as possible using external car modifications on the back of the car.
Reducing the drag will enhance the car’s speed while decreasing the fuel consumption and increasing
the car’s stability.

2. Geometry

In this part, both base car model and car with modifications model are described in detail with no
rotational motion of the car tires.

2.1. Base car geometry

The KIA pride, is a small passenger car. The car was modelled using SOLIDWORKS 2018 as seen in
Figure 1. The geometry was presented in cartesian coordinates, the full-scale car dimensions are
3935mm length, 1455mm height and 1605mm width. No mirrors are presented, no grilles and no solid
contents.

![Figure 1. Base car model.](image-url)
Delta type Vortex generators (VGs) were selected as the best of shapes of VGs as shown in [16]. These VGs were modelled by SolidWorks 2018 with 9 pieces, and the distance between each one is 100mm. The position and number of VGs are shown in Figure 2.

2.3. Rear under body slices diffuser
Slices of diffuser start from halfway between the back wheels at an angle of 12.5° and divide into three sections. The width of a slice is 27mm and the distance between each slice is 270mm [5]. It is found that 27 mm thickness gives the optimum results, as compared with other dimensions. The position and shape of the diffuser is shown in Figure 3.

2.4. Rear wing spoiler
A rear wing spoiler was used, an inverted single element wing with end plates, modelled by SOLIDWORKS 2018. The wing profile used is NACA2412 airfoil [15]. The chord length is 133 mm with 12° angle of attack and 30mm height between the minimum point on wing and the car trunk. The span length is 1290 mm. These data were selected to give more efficient results as presented in [17], and can be seen in Figure 4.
3. Mathematical and Numerical Solution

3.1. Governing equations and turbulence module
The governing equations used by the software package, a three-dimensional steady incompressible RANS for each of continuity and momentum equations:

- Continuity Equation:
  \[ \frac{\partial}{\partial x_i} u_i = 0 \]  

- Momentum Equations:
  \[ u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} - u_i u_j \right) \]  

Where \( x_i \) and \( x_j \) are the tensor forms in the ith and jth direction \( (i = j = 1, 2, 3 = x, y, z) \), \( u_i \) and \( u_j \) are the set-mean velocity tensors and \( p \) is the pressure of the air. \( \rho \) is the density of the air, assumed incompressible flow, and \( \theta \) kinematic viscosity are constant values based on International Standard American (ISA), Reynolds stresses \(-u_i u_j\), were modelled using the realizable k-epsilon two equations turbulence model. For near wall treatment, the standard wall functions are used. The values of total dissipation rate Prandtl number is 1.3. Further information may be evaluated in [14].

3.2. Assumptions
The assumptions used by the software package for the working fluid are steady flow rate, incompressible fluid, Newtonian fluid. Fluid structure interaction is neglected, as are the heat transfer effects body forces.

3.3. Boundary conditions
The significance of the boundary condition is to reproduce the real condition. In the commercial package ANSYS workbench, fluent solver, the boundary conditions presented as following the velocity of flow at the inlet boundary were used to determine the velocity of flow. In the present study the inlet velocities (10, 20, 30 and 40 ms\(^{-1}\)) are normal to the boundary. The outlet boundary was modelled using a pressure outlet. The pressure gauge was set at zero and symmetry boundary condition (no shear stress) of the far field and side walls were modelled. The bottom wall was modelled as a moving wall with a speed varying with same velocity as the inlet flow. Car wall was used to limit the solid and fluid zones. In viscous flow, no slip is required at car walls. Boundary conditions for all of the modifications are modelled, the conditions are detailed in Table 1, Figure 5 and the following equations:

- Inlet and Bottom wall Boundary Condition:
\[ \text{Inlet Air Flow Velocity} = V_x \text{ms}^{-1} \quad (3) \]

- Outlet Boundary Condition:

\[ \text{Pressure outlet } P_x = 0 \text{ Pascal} \quad (4) \]

- Far Field and Side walls:

\[ V\cdot n = 0 \quad (5) \]

- Car Body walls:

\[ V = 0 \quad (6) \]

| Table 1. Domain setup and boundary conditions. |
|-----------------------------------------------|
| Domain Setup and Boundary conditions           |
|-----------------------------------------------|
| Inlet | Velocity magnitude normal to the boundary | 10,20,30,40 ms⁻¹ |
| Outlet | Gauge Pressure magnitude normal to the boundary | 0 Pascal |
| Far field & Side walls | Symmetry (No shear) | V.n=0 |
| Car Body wall | No slip | V=0 |
| Bottom wall | Moving wall | 10,20,30,40 ms⁻¹ |
| Flow condition | Steady state | Time=0 |
| Fluid Properties | Fluid type | Air |
| Density | Kinematic viscosity | 1.225 Kgm⁻³ |
| | | 1.7894×10⁻⁵(kg(ms)⁻¹) |
| Model | Turbulence specification method | Realizable k-epsilon |
| Solver | Pressure based | |
| Convergence factor | 0.0001 | |  

3.4. Numerical Solution

3.4.1. Computational domain. The geometry consists of a rectangular domain which contains the car model. The size of the computational domain was chosen extends two car lengths (2L) in front of the car and five lengths (5L) behind the rear of the car, as seen in Figure 5. The distance behind the car should be sufficiently long to accommodate the separation of flow without affecting the flow of air in the outlet section, also to distinguish the wake region. The distance between the upper boundaries and the flyover of the car lengths is set to three car lengths (3L) and the distance between the side surface of the car and the side wall of the domain is set to one and a half car length (1.5L).

3.4.2. Mesh generation. Grid generation is an important step in numerical simulation, and requires a high quality of meshes to accurately capture difficult physical phenomena. Thus, in this study the number of elements depends on the grid independence test, as shown in Figure 6. The model and domain mesh are generated in ANSYS Fluent. The total number of tetrahedral grid cells about 3620547 elements are used, see Table 2. The smallest elements cells are generated near the adjacent surface of the car, and the larger cells are located near the boundary domain, see Figure 7.
Figure 5. Schematic of computational domain and boundary conditions.

| Table 2. Mesh details. |
|------------------------|
| Mesh Details           | Values          |
| Number of Element      | 3620547         |
| Number of Nodes        | 759650          |
| Number of Inflation Layer | 6             |
| Minimum cell size      | 4 mm            |
| Maximum Cell size      | 400 mm          |
| Surface Growth Rate    | 1.2%            |
Figure 6. Grid independency test at $V=30\text{m}\cdot\text{s}^{-1}$
4. Results and discussion
The results, such as surface pressure distributions and velocity of the flow field around the base car model, car with vortex generators, rear wing spoiler and rear under body diffuser slices modifications are presented in this section.

4.1. Base car model results

4.1.1. Graphical representation. The pressure distribution over the car body’s centre line (upper surface) can be seen in Figure 8. It can be seen that the nose of the car represents the maximum pressure (stagnation point). Then, the pressure decreases and increases in a wavy form which is attributed to the car surface shape. The pressure decreases to the minimum value (spike) when the airflow reaches the edge of the front bonnet due to flow acceleration. The pressure gradually increases to the beginning of the windscreen (high pressure concave region). The pressure starts to drop again when the flow reaches the flyover of the car (roof) with gradual increase in pressure along it. When the flow reaches the rear windscreen, the pressure decreases due to the convex shape between the roof and the windscreen, which causes the flow to accelerate at this region. The high deflection angle of the flow causes the flow to separate from the rear windscreen, due to high adverse pressure gradient at this region. A low-pressure region is created behind the vehicle which represents a flow separation region of vortices at the car’s back. From the figure it is clear that the change in the free stream velocity is directly affected the pressure distribution along the centre line of the car model.

Figure 7. Mesh of the car section model and the domain.
The effect of free stream velocity with the drag force is illustrated in Figure 9m where the effect of modification on pressure contours on the car body can be observed.

4.2. Car model with vortex generators
From the previous discussion of the base car model, the affected regions of separation were noticed at the rear window and the back of car must be treated to decrease the separation domain. Vortex generators (VGs) are fixed at the end stream of the car flyover when the flow separation occurs, so as to control and delay the flow separation over the car windscreen by reenergizing the boundary layer due to high energy vortices of VGs which reduces the drag force of the car.

Figures 9b,10b and 11b show the flow and surface pressures, velocity contours and 3D velocity streamlines of the car body at 40ms$^{-1}$ velocity. From these figures, an enhancement of the pressure distribution is observed at the end of the car flyover and wake region of the body. As the pressure envelope behind the car is reduced in size with use of the VGs, the low-pressure regions are concentrated behind the VGs due to the eddies that they have created. Three-dimensional surface pressure of the car model is shown in Figure 9b, which is clearly decreased, especially at the rear window of high-pressure red colour. The velocity contours and stream lines are shown in Figures 10b and 11b respectively.

4.3. Car model with rear wing (spoiler)
The spoiler is one of the major aerodynamic components used to modify the airflow around the car to reduce drag force. When fixed at the end of the car’s back surface, this can result in an enhancement in car stability by decreasing drag force, which may cause unexpected handling in a car at high velocity.

Figures from Figure 9c to 11c show the pressure and velocity distribution of the car body at 40 ms$^{-1}$ and velocity with the presence of spoiler, respectively. From these figures, the differences of pressure distribution are observed as compared with base car model at the car surface and wake region where the pressure distribution at the car surface is increasing due to the new stagnation point at the front of the spoiler, while the wake region pressure behind the car is increased. The spoiler obstructs the flow of air through it, thus preventing the air from pulling down, which would otherwise generate a low-pressure air envelope and turbulence behind the car that contributes to the pressure drag.

4.4. Car model with rear under body diffuser slices
Ground effect produces great amount of negative lift (downforce), and increases the drag. The air leaving the under body of a car with high velocity contribution which causes the flow separation to expand. Due to the differences in pressure between upstream and downstream at the rear of the car, this may produce a circulation in a flow which increasing the drag force. A diffuser with slices is installed under the car model at the rear which is used to decrease the circulations in the wake region. Slices of the diffuser with selected angle (12.5°), essentially guide some of the flow from under body of car to the regions of low pressure. This flow decreases the impact of low pressure and vortices. The air is brought back to its first velocity because of increasing in area gradually due to diffuser when its exits from the rear under car region, thus adding slice modification decreases circulation in flow.

Figures from Figure 9d to 11d show the pressure and velocity of the car body at 40ms$^{-1}$ velocity. From these figures, the wake region has decreased because the flow moves from under body diffuser towards the low-pressure area behind the car.

4.5. Base Car and Its Modifications Comparisons

4.5.1. Pressure and velocity contour analysis. Figures from Figure 9–11 show a comparison between the base car and the car with all modifications at 40 ms$^{-1}$. The pressure contours around and on the car, velocity contours, 3D velocity streamlines around the car body, are all presented respectively. From these figures it can be seen that the effect of including all modifications at once is
not necessarily to enhance the flow, so it is important to calculate the total drag coefficient of the car, which is presented in the next section.

4.5.2. **Graphical representation.** The best decrease in the aerodynamic drag coefficient is seen when adding all three modifications, as shown in Figure 12. Figure 13 shows the pressure distribution on the upper center line surface of the car to compare the four cases of car modification. The differences in pressure are clear in the vortex generator and spoiler regions. Figure 14 shows the effect of rear diffuser by decreasing the pressure at the rear rejoin.

Table 3 shows the average drag coefficient reduction percentages, it is 1.73% by the vortex generators, 2.47% by the diffuser, 3.05% by the spoiler and 3.8% by using the three modifications together.

4.6. **Comparison with Previous Works**

There are no specific data of drag coefficient available for the KIA Pride vehicle. So, six published workers were used to validate the results [1, 4, 5, 12, 13 and 14]. Some of these works studied the computational drag coefficient, velocity contours, pressure contours and surface pressure distribution and the others offered drag coefficient with some car modifications at different boundary conditions. The drag force found is compatible with the published numerical and experimental works, in spite of the difference in car model used.

The drag reduction percentage with rear under body modification with diffuser by [4] was 4.64%, by [1] was 8.65%, by [13] was 6.146% and in the present work is 2.47%. The drag reduction percent with vortex generators by [12] was 0.4% [14] 0.6%, [1] 4.94%, [13] 2.284% and in the present work is 1.73%. The drag reduction percent with spoiler by [12] 0.6% and in the present work is 3.05%. The drag reduction percent with full modifications by [5] 4.35% and in the present work is 3.8%. The drag reduction percentage shows a good agreement with published data.

![Figure 8. Numerical pressure distribution at the car upper surface center line for 40ms⁻¹ velocity.](image-url)
Figure 9. Surface pressure contours of car body for different modifications at 40 ms$^{-1}$ velocity.
Figure 10. Velocity contours of car body for different modifications at 40 ms\(^{-1}\) velocity.
Figure 11. Velocity streamlines around car body for different modifications at 40 ms\(^{-1}\) velocity.

Figure 12. Numerical drag coefficient of the car at different Reynolds numbers with different drag reduction modifications.

Figure 13. Numerical pressure distribution at the car upper surface centre line with different drag reduction modifications at 40 ms\(^{-1}\) velocity.
Figure 14. Numerical pressure distribution at the car upper and rear lower surface centre line for base and sliced diffuser car at 40 m s$^{-1}$ velocity.

Table 3. Reduction of drag coefficient.

5. Conclusion

CFD analysis was successfully performed on a car model. After confirming the validity of the simulation of a baseline car, the next step was to study the velocity variation on the drag force and then add modifications on the base car model, which can be effective in reducing drag.

The results obtained showed that the drag coefficient slightly decreased with the increasing of Reynolds number.

The velocity effects were least significant, with changes in drag coefficient for the speed range tested.

Our study modified the KIA Pride by adding vortex generators, spoiler and diffuser slices and a comparison was made between them. The best average of drag reduction when using a spoiler was 3.05%, while using the three modifications together gave the best results of average drag reduction which was 3.8%.

Therefore, by comparing all the modifications techniques, we have concluded the best result occurs when all modifications are added together on the car.

Acknowledgments

The authors would like to thank Lec. Assit. Mustafa S. Abood and Dr. Ahmed Alsaadi for their support.

| V (m s$^{-1}$) | C$_d$ (base car) | C$_d$ (Car with VGs) | C$_d$ (Car with Spoiler) | C$_d$ (Car with diffuser) | C$_d$ (car for Three Modifications) | $\Delta$C$_d$% |
|----------------|------------------|----------------------|--------------------------|--------------------------|-----------------------------------|--------------|
| 10             | 0.352            | 0.34                 | 3.35%                    | 0.339                    | 3.63%                             | 0.335        | 4.77% |
| 20             | 0.340            | 0.335                | 1.49%                    | 0.329                    | 2.67%                             | 0.326        | 4.14% |
| 30             | 0.335            | 0.331                | 1.08%                    | 0.327                    | 1.67%                             | 0.325        | 2.87% |
| 40             | 0.333            | 0.33                 | 1.02%                    | 0.327                    | 1.92%                             | 0.322        | 3.42% |
| Average        | 0.34             | 1.73%                | 3.05%                    | 2.47%                    | 3.80%                             |              |      |
References

[1] Ahmed h and Chacko 2012 Computational optimization of vehicle aerodynamics 23(1) p 313–318.

[2] Marklund J 2013 Under-body and diffuser flows of passenger vehicles [M.Sc. thesis] Department of Applied Mechanics, Chalmers University of Technology, Sweden.

[3] Moussa A A, Yadav R and Fischer J, August 2014 Aerodynamic drag reduction for a generic sport utility vehicle using rear suction Journal of Engineering Research and Applications Vol 4 Issue 8 p 101-107.

[4] Hassan S M R, Islam T, Ali M and Islam M D Q 2014 Numerical study on aerodynamic drag reduction of racing cars Procedia Engineering vol 90 p 308 – 313.

[5] Sharma R B and Bansal R Jul. – Aug. 2013 CFD simulation for flow over passenger car using tail plates for aerodynamic drag reduction Journal of Mechanical and Civil Engineering IOSR-JMCE Vol 7 Issue 5 P 28-35

[6] Selvaraju P N, Parammasivam KM, Shankar, Devaradjane G 2015 Analysis of drag and lift performance in sedan car model using CFD Journal of Chemical and Pharmaceutical Sciences JCHPS Issue 7(7)

[7] Wahba E M 2012 Aerodynamic drag reduction for ground vehicles using lateral guide vanes CFD Letters ISSR Journals Vol 4(2) P 68-79.

[8] Sudin M N, Abdullah M A, Shamsuddin S A, Ramli F R, Mohd M. 2016, 2014 Review of Research on Vehicles Aerodynamic Drag Reduction Methods International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol 14 No 02.

[9] Sarkar S, Thumnmar K, Shah N, Vagrecha V. 2019 A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles International Research Journal of Engineering and Technology (IRJET) vol 06 issue 02.

[10] Mukut, A. N. M. M. I. and Abedin, M. Z. (2019). Review on Aerodynamic Drag Reduction of Vehicles. International Journal of Engineering Materials and Manufacture, 4 (1), 1-14

[11] Cattafesta, Louis and Bahr C J 2010 Fundamentals of wind-tunnel design Encyclopedia of Aerospace Engineering, Florida State University,

[12] Anish A, P G Suthen and Viju M K 2017 Modelling and analysis of a car for reducing aerodynamic forces International Journal of Engineering Trends and Technology vol 47(1) p 1–16.

[13] Hussain I Y and Faris S 2017 Aerodynamic characteristics of Peugeot 405 car model [MSc. thesis], university of Baghdad, Mechanical Engineering Department.

[14] KoikeM, Nagayoshi T and HamamotoN 2004 Research on aerodynamic drag reduction by vortex generators Mitsubishi Motors, Technical Papers p 11-16.

[15] Chodagudi K and Rao T B S 2012 analysis of NACA 2412 for automobile rear spoiler using composite material International Journal of Engineering Research & Technology Vol 01 Issue 07.

[16] Govindharajan R Parammasivam K M and Narayanan S S 2013 Design of vortex generators for light transport vehicles (LTVS) using CFD Proc. of the 8th Asia-Pacific Conference on Wind Engineering (APCWE-VIII) p 978–981.

[17] Kumar D M V S 2017 Design, analysis and manufacturing of a car rear spoiler for drag reduction Iarjset vol 4 No. 6 p 89–96.