University of Baghdad College of Engineering

JE JOURNAL OF ENGINEERING

Mechanical and Energy Engineering

Drag Reduction Using Passive Methods on KIA PRIDE Car Model

Zahraa M. Saleh *
M.Sc. Student
College of Engineering-University
of Baghdad
Baghdad, Iraq
Zahraamahdi89@gmail.com

Anmar H. Ali
Dr.
College of Engineering -
University of Baghdad
Baghdad, Iraq
Aha_has@yahoo.com

Mustafa S. Abood
Assistant Lecturer
Alkindy College of Medicine-
University of Baghdad
Baghdad, Iraq
mustafasabeeh@kmc.uobaghdad.edu.iq

ABSTRACT

An experimental study on a KIA pride (SAIPA 131) car model with scale of 1:14 in the wind tunnel was made beside the real car tests. Some of the modifications to passive flow control which are (vortex generator, spoiler and slice diffuser) were added to the car to reduce the drag force which its undesirable characteristic that increase fuel consumption and exhaust toxic gases. Two types of calculations were used to determine the drag force acting on the car body. Firstly, is by the integrating the values of pressure recorded along the pressure taps (for the wind tunnel and the real car testing), secondly, is by using one component balance device (wind tunnel testing) to measure the force. The results show that, the average drag estimated on the baseline car for different Reynolds numbers was (0.381) and the drag force was reduced by adding a spoiler and a slice diffuser to (4.45%, 1.5%) respectively, whereas the amount of drag reduction was (5.46%) when all drag reduction modifications were added together on the base car. No effect was noticed as vortex generators when added separately. The deviation in the drag coefficient from the real car testing was about (6.2%) and shows a very good agreements between the real car test and that of the wind tunnel test.

Keywords: Real Track Test, Kia Pride, Passive Flow Drag Reduction Methods.

تقليل قوة الممانعة باستخدام طرق سلبية على سيارة نوع كيا برايد

زهراء مهدي صالح
طالبة ماجستير
كلية الهندسة -جامعة بغداد

انمار حامد علي
دكتور
كلية الهندسة -جامعة بغداد

مصطفى صبيح عبود
مدرس مساعد
كلية طب الكندي -جامعة بغداد

الخلاصة

تم إجراء دراسة عملية على نموذج سيارة كيا برايد (سايبا 131) بمقياس حجم 14:1 في نفق هوائي، أضافة الى اجراء اختبار على سيارة حقيقية. تمت إضافة بعض التعديلات على السيارة لغرض التحكم بمسار ندفق الهواء وتشمل (مولد دوامات والمفسد

وشرائح تضاف للناشر) وبالتالي لتقليل قوة الممانعة. تم استخدام نوعين من العمليات الحسابية لتحديد قوة الممانعة التي تؤثر على جسم السيارة. العملية الاولى يتم فيها حساب قيم الضغط المسجلة والمحسوبة من صنابير الضغط بعد اجراء عملية التكامل عليها (بالنسبة لنفق الهواء واختبار السيارة الحقيقية)، والثانية باستخدام جهاز ميزان احادي البعد (في اختبار نفق الهواء) لقياس الممانعة. أظهرت النتائج أن معدل الممانعة المقدّر على سيارة بدون أضافات لأرقام رينولدز مختلفة كان (0,381) بينما تم تقليل قوة السحب بإضافة المفسد وشرائح الناشر إلى (4,45%، 1,5 %) على التوالي، في حين كانت كمية تقليل الممانعة (5,46%) عند إضافة جميع تعديلات تقليل الممانعة معًا على السيارة الأساسية. لم يلاحظ أي تأثير ملموس من مولدات دوامات عند إضافتها وحدها. كان التغير في معامل السحب عن اختبار السيارة الحقيقي حوالي (6.2 %) ويظهر توافق جيد للغاية بين اختبار السيارة الحقيقي واختبار نفق الهواء.

الكلمات الرئيسية: اختبار المسار الحقيقي، سيارة كيا برايد، أساليب الحد من قوة الممانعة.

1. INTRODUCTION

With the increase in fuel prices and economic crisis in most of countries in addition to the demand of environmental protection organizations to reduce the emission of toxic gases from the cars, so reducing fuel exchange is very important now a days. This reduction is related to reducing the pressure drag force on the car, which is dominated of about 80% of the total drag on the car body to investigate the ability of reducing drag force experimentally, so that, many researches were made in this investigation. (**Islam, et al., 2013**) studied the ability to reduce the drag force on a sedan car model by using a delta type vortex generator experimentally. The experiments were conducted on subsonic wind tunnel in different Reynolds numbers ranged from 1.2 to 6.4×10^4. It is found that, the drag was reduced by 2.15% due to delay in the flow separation at the downstream end of the car roof, which minimizing the pressure difference between the downstream and upstream sides of the car. (**Raman and Hari, 2016**) adopted three passive techniques to modify the flow over saloon vehicle, two types of vortex generator (bump shaped with a slope angle of 25° to 30° and delta wings), rear screen and rear fearing were used to reduce the drag force on the vehicle. A comparative study was made between them, the results showed that the delta wings were more effective than bump shape. The results also showed that 6.5% gotten it, in reducing the drag force. An experimental study was presented by (**Bello-Millán, et al., 2016**) to describe the effect of different yaw angles on drag force on the Ahmed body with angle of slant 25°. The body manufactured by CNC mill with lower than 0.1mm tolerance, and it was made with expanded polystyrene (EPS). Reynolds number was ranged from 3×105 to 30×105, which corresponding to free stream velocity ranged from 6.9 m/s to 24.5 m/s. The results showed that the Ahmed body drag coefficient was decreased slightly as the Reynolds number increased in a null yaw angle. Also, it found that, at $0° \lesssim$ yaw angle $\lesssim 60°$ the drag coefficient was increased and at $60° \lesssim$ yaw angle $\lesssim 75°$ remained constant. Finally, at yaw angle = 90° it was increased again. (**Sawyer, 2015**) presented an experimental work to reduce drag force by adding tabs to the end edge on square bluff body in stationary ground effect to reduce vortex shedding. The drag force with and without adding tabs had been measured. Three tests were performed using different ground plane lengths with respect to three different heights of model and at Reynolds numbers corresponding to each clearance was (1.98×10^5, 3.95×10^5, 5.92×10^5) in open suction type wind tunnel, the results showed that the addition of tabs to the trailing edge of a bluff body was ineffective in reducing the overall pressure drag because of the vortex shedding over the trailing edge, instead it was increased the drag coefficient by the amount of 8.6%. The adding ground effect when H/h = 0.50 test cases, were reduced the drag coefficients by about 3%. (**Frystak and Popela, 2016**) conducted an experimental test to study the aerodynamic characteristic of a Formula SAE car to discover the zones from which enhancement of these characteristics may be extracted. The test was made in a

wind tunnel with model scale 1:4 used in TU Brno Racing's 2016 car /Dragon 6. Four different figures were used with the usage of inverted wings and a floor with a diffuser. Reduction in drag force was 12% when a floor was used with the diffuser while the inverted wing produced high drag in spite of increasing the down force. Also, the modification under body angle was used instead of diffuser to make the down force on the car's underbody which reduced the drag force. **(Skrucany, et al., 2016)** explained the experimental wind tunnel study of the effect of aerodynamic trailer devices to reduce the drag force, the truck model was in scale of 1/24. Two devices were sealed, the tail and wheels which had been chosen to compare between them. These devices made from soft wood (spruce) and stacked on the two sides of the model. The free stream velocities were 15.48 m/s, 17.28 m/s and 18.08 m/s. The results showed that the sealed wheels were decreased the drag coefficient higher than that when sealed on the tail**. (Mcarthur. et al., 2018)** made a passive treatment in the experimental study on heavy vehicle in a wind tunnel with a scale of 1/3, they measured base pressure, drag coefficient and wake total pressure, at 0^0 to 15^0 yaw angle for six configurations of semi-trailer cabin extenders, side-skirts sharp, flat fairing, closed and boat-tail. The results had been shown a good reduction in drag coefficient, while the closed side skirts at yaw angle = 0° was shown the best certain modifications by reducing the drag force of about 15% .**(Shankar and Devaradjane, 2018)** focused in their work on the study of aerodynamic characteristics of a general sedan car model had three numbers of delta shaped vortex generators experimentally. The middle one was left stationary and the other two vortex generators orientation had been changed using a controller and the results had been studied. The drag coefficient of the car used with four different yaw angles been quantitatively tested by sub-sonic wind tunnel. The results showed the peak coefficient of drag reduced rates of 4.53% in the case of car model with vortex generators having leading edges facing the rear end and the mid plane of the car respectively when compared with the car model without vortex generators. **(Cheng. et al., 2019)** studied the effect of yaw angle on the aerodynamic performance of an SUV car model fitted with a roof spoiler experimentally. They were studied two common types of spoilers, namely, strip and wing types. The SUV model was based on the Ahmed body at 35° slant angle. The yaw angle affects the aerodynamic performance of the SUV model negatively regardless of the presence of a spoiler. They observed that the role of a spoiler prevents the bi-stability behavior of flow, which occurred in the model without the spoiler. Furthermore, the strip spoiler was reduced the drag coefficient. The upshots of the surface pressures indicated that the modifications in the flow around the rear slanted body are the primary elements affecting the aerodynamic performance of the SUV model using the spoiler.

In the current investigation, KIA Pride (SAIPA 131) car with two experimental models are tested using low speed wind tunnel in addition to a real car was tested using real road track experiments. The free stream velocities are 10, 20, 30 and 40 m/s, corresponding to Reynolds numbers 2.51×10^5, 5.02×10^5, 7.54×10^5, 10.05×10^5 and 12.56×10^5, respectively, depending on the velocity and the characteristic length of the wind tunnel tests, while the real car speeds are varied from 10, 20, and 30 m/s for the real track tests. The pressure values were calculated using pressure taps and the drag force was measured by one-component balance methods outside the wind tunnel test section, while the pressure values of real track experiments are calculated using static pressure sensors distributed along the center line of the real car. The aim of this research is to decrease the drag of the car as possible using some external car modifications to rear back of the car model so as to improve the aerodynamic characteristics of the car. Reducing the drag will increase the car speed with decreasing the fuel consumption and also increasing the car stability.

2. EXPERIMENTAL SETUP

2.1 Wind-tunnel and Test Section

The experiments were conducted in an open circuit suction type low speed wind tunnel at the aerodynamic lab. in the University of Baghdad. The dimensions of the wind tunnel test section are (70cm ×70cm×150cm) with axial flow fan characteristics (75hp,3000rpm and 3 phase AC motor), see **Fig. 1**. The maximum calibrated velocity was tested of about 55m/s without a model, for more details see **(Hussain and Ali, 2014).**
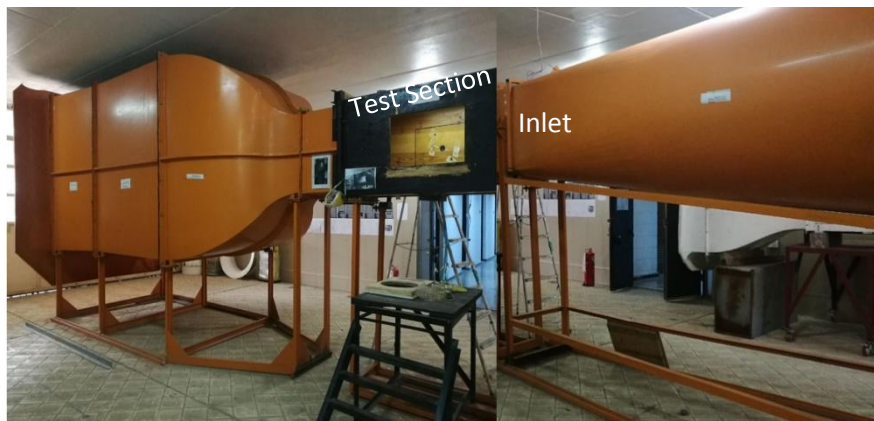

Figure 1. Low speed wind-tunnel used.

2.2 Base Car Model

In the present work, the car model KIA Pride (SAIPA131) was selected, which is produced by the State Company for Automotive Industry in Iraq, since it is commonly used in Iraq, Syria and Iran. Therefore, based on the dimensions that obtained from the official site of KIA pride, the car was modeled by using the program SolidWorks 2018 then two car models were printed in the 3-D plastic material model. The scale was reduced to 1:14 without accessories like side mirrors. The dimensions of the full-scale car are (3935mm length, 1455mm high, and 1605mm width). For the first model, 21 holes distributed along the center line to measure the pressure values along it, and 7 holes on the right and left sides for calibration. The orifice inner diameter of the tube in the first printed model is 1 mm which connected with plastic tube of 5mm diameter by plastic joint. These plastic tubes are then connected to the Micro-Manometer to measure the static pressure along the car surface with three lines as shown in **Fig. 2**. The upper and lower parts were printed and then integrated after tubes were installed using adhesive epoxy material. The car was painted with raw material which was pasted to treat the defects and cracks and then paint it. The other car model which is tested by one component balance was printed entirely in one solid piece using 3D printer, the car was painted and drilled from the bottom of the fixation mechanism. Finally, the cars were softened with 400 grit fine sandpaper to finish surface smoothly. See **Fig. 3**. The side mirrors of the car and the under-body details are neglected in order to reduce the complexity of the problem.

2.3 Modifications of Car Model

All three modifications; vortex, diffuser slices and spoiler are designed in SOLIDWORKS "CAD" program, then they were printed in 3D printer with scale 1:14. They are made from plastic material, see **Fig. 4**. The modifications were fixed, using super adhesive on the second car model.
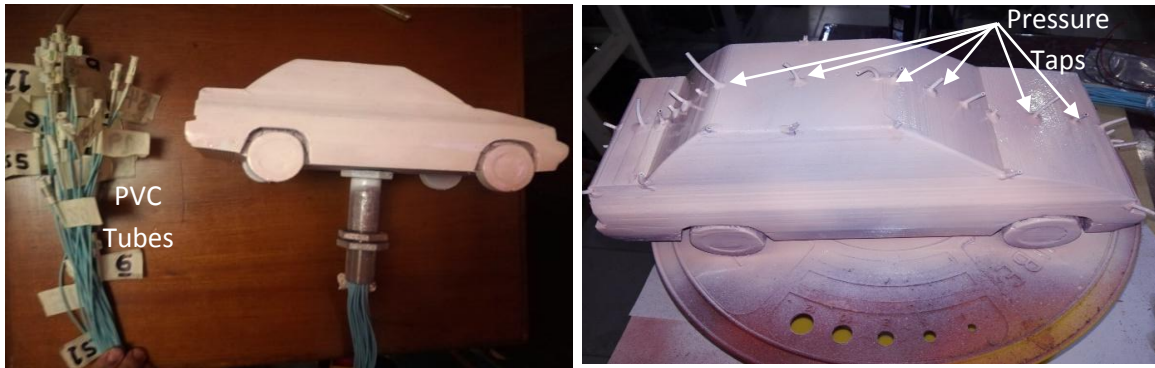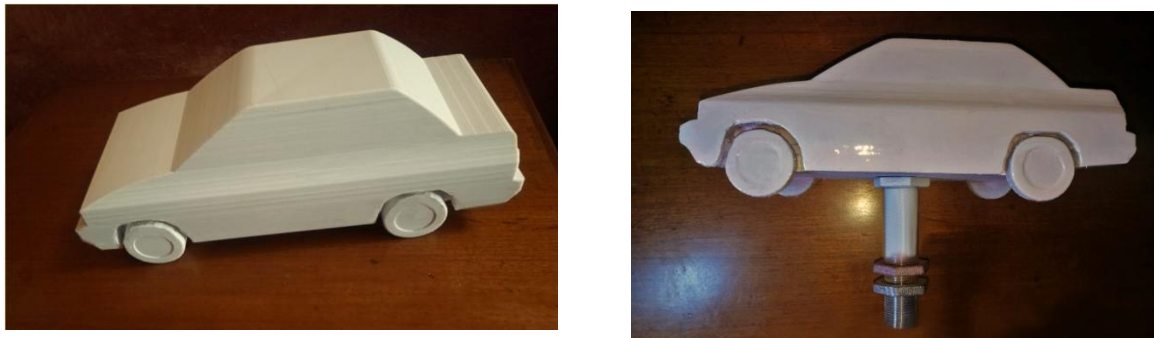
Figure 2. SAIPA 131 test model 1 manufacturing steps.

Figure 3. SAIPA 131 test model 2.

Figure 4. Modifications of SAIPA 131 test model 2.

3. EXPERIMENTAL PROCEDURE

The car model firstly fixed in the test section of the wind tunnel. The model was calibrated to fix its direction straight with the wind. The wind tunnel in other hand was operated at a specified velocity for testing run. Using the Pitot static tube to achieve the dynamic pressure of that velocity. The test run velocities are; 10, 20, 30 and 40 m/s, corresponding to Reynolds numbers 2.51×10^5, 5.02×10^5, 7.54×10^5, 10.05×10^5 and 12.56×10^5, respectively depending on the velocity and the characteristic length of the wind tunnel tests.

Two methods were used in this experiment to determine the drag force and coefficient acting on the car body. These are;

3.1 Pressure Taps Method

During tests, the values of pressure taps on the sides of the model must be equal to verify that the car is in a symmetrical position (right and left taps must be equal values). Other taps at the center line of the car model were used to record the pressure of (21 points) for upper surface pressure coefficient which is calculated using the equation;

$$Cp = \frac{(ps - p\infty)}{q\infty} \tag{1}$$

The distribution of pressure obtained over a car profile can be used to calculate the drag by the following law;

$$Drag\ Force\ (F_d) = \sum_{i=1}^{n}(picos\theta iAi) \tag{2}$$

3.2 One-component Balance Method

Fig. 5 shows the modified one component balance which is used in the present wind tunnel to find the drag force of the model. It consists of only one actuator which is used to measure the drag and acted in the horizontal direction passing through the axis of the car model support. This actuator is connected to the load cell (max weight 10 kg). The strain gage is connected to an Arduino board to measure the drag force of the model.

The one component balance has been accurately calibrated to obtain the precise drag force. The calibration was obtained by connecting the load cell of the Arduino board by known calibrated masses (911, 454 and 236 grams). Wooden arm was used, which is connected with the back of the car by pulley and string to the known masses. These masses were gradually placed on the holder of calibrating arm and connected to the Arduino board program to record the reading, after that the calibration factor is added to the program library and tare the rig weight. By Using this method of balancing, we get a value representing the force at each velocity to calculate the drag coefficient and according to the following law;

$$F_d = \frac{w \times g}{1000} \tag{3}$$

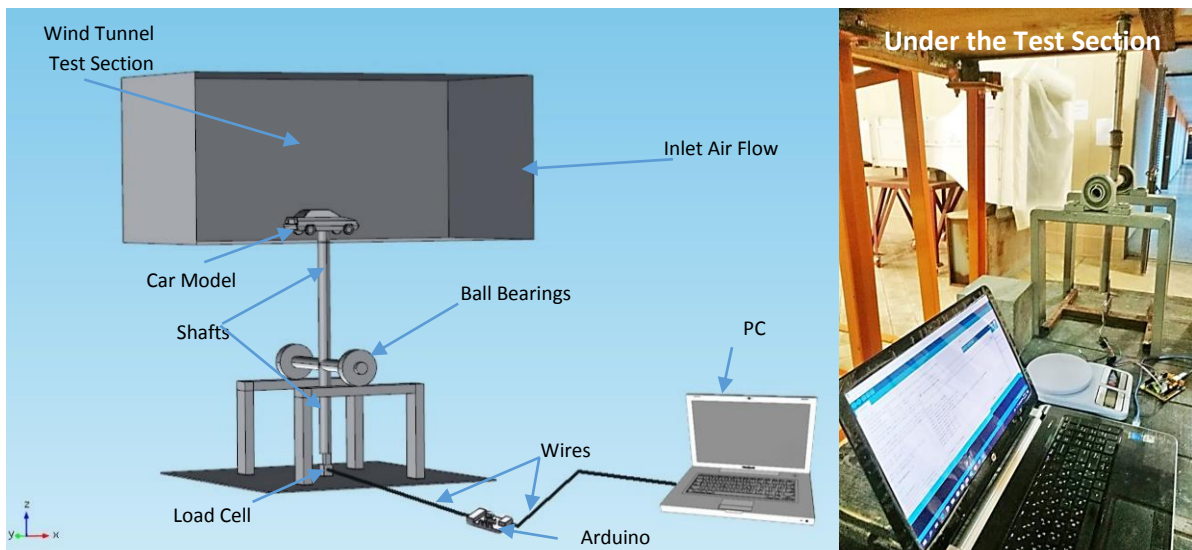$$C_d = \frac{Fd}{(0.5 \, \rho v^2 \, A)} \tag{4}$$

Figure 5. Schematic of one-component balance mechanism.

3.3 Uncertainty Analysis

3.3.1 Uncertainty of Velocity

The inlet velocity V is found by the equation:

$$V = c \sqrt{h} \tag{5}$$

Then;

$$\frac{dv}{v} = \frac{1}{2} \times \frac{dh}{h} \tag{6}$$

Where (dh) is determined from the equation below:

$$dh = \sqrt{(h - h_m)^2 + (\partial h)^2} \tag{7}$$

The first term of the R.H.S indicates the deviation from the mean (h_m), which is determined from $(h - h_m)$, where the mean is found from $\frac{\sum_{i=1}^{n} h_i}{n}$, where the number of points is (n), and the second term of R.H.S in Eq. (7) indicates the measurement error.

Such an example, when the mean velocity 20 m/s on the inlet of the test section calculated by using the measured pressure head values of points: Measured five values of (hi) at the mesh points were 23.23, 22.871, 22.669, 22.571 and 21.963 mmH2O, so the mean value hm is (22.661 mmH2O), and the deviation was (0.256302 mmH2O). The error of measurements (0.611 mmH2O). Also, from Eq. (7), the (dh) was determined as (0.66258 mmH2O), finally the Eq. (6) calculated the velocity relative error will be (±1.45%).

3.3.2 Uncertainty of Pressure

Following the same procedure mentioned above, the relative error in the pressure coefficient was found at;

$$\frac{dCp}{Cp} = \frac{d(p - p\infty)}{p - p\infty} \tag{8}$$

The mean value of (p-p∞) was calculated as (222.227), and the deviation from the mean as (2.513 Pa). The measurement error was taken as (6 Pa). The total error will be (6.51 Pa), and from Eq. (8), the relative error in Cp is (2.89%).

4. REAL CAR TESTING

Road testing is considered as a second choice which is used to study the aerodynamic characteristics of vehicles. An actual vehicle experiment testing has more reality than the other like CFD and wind tunnel model testing because the run is made on an actual car and real conditions. So, the car can move at its full speed and in the real way, where the air stream deforms due to the geometrical car under test. The disadvantages of this method firstly, the test conditions are unstable as the climate, temperature of the car and tire condition, in addition to the behavior of the driver himself, etc. In addition to the high cost required in such tests, racing cars for example, the cost of operating a car is really high. Furthermore, at the design and manufacturing stage, there is no real car that can be tested. Therefore, this method cannot be applied in the early stages of development.

Hence, the real track testing is the most realistic method to test. In this experiment real SAIPA 131 car full scale model 2009 with all details. The car profile is shown in **Fig. 6** and the dimensions are 1455mm height 3952 mm length and 1605mm width.

4.1 Test Procedure

The car was tested by driving it in real road at constant selected velocities (10,20 and 30 m/s), which corresponding to (36, 72 and 108 Km/h) at barometric conditions. The experiment was conducted in summer season with ambient temperature of 45°C.Thirteen static air pressure sensors were distributed on the center line of car at selected locations fixed in the same locations which

were distributed along the model center line that is used in the wind tunnel test to measure pressure distribution along the car profile to compare the recorded results, see **Fig. 7**.

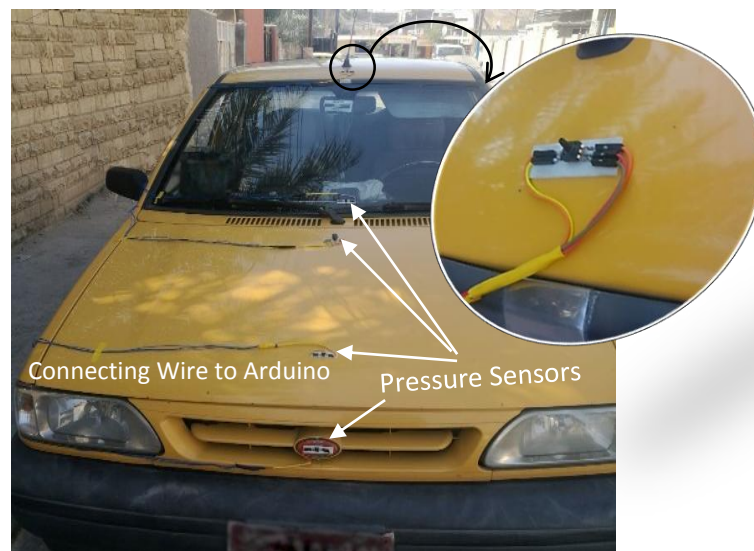**Figure 6.** SAIPA 131 Car profile with dimension **(Internet Sources)**.

Figure 7. SAIPA 131 Car with sensors.

5. RESULTS AND DISSCUSIONS

In wind tunnel experiments, four test runs were performed on two test car models. The first car model test, the pressure distribution along the model surface were measured from the car pressure taps. The second model, one component balance was used on the base car at first and then adding the vortex generators once then adding the spoiler. Adding that after the diffuser slices and finally adding the all modifications together, each step the drag force was recorded at four different velocities. Both models were used to find the drag coefficients. The required velocity in each test run was found by measuring the dynamic pressure at the test section inlet. The pressure data and drag forces were recorded; then the results were analyzed and the relationship between the studied variables was studied. The data in this part of investigation recorded for different velocities (10, 20, 30 and 40m/s).

Fig.8 shows the experimental pressure distributions along the upper surface of the model for different speeds. From the figure, it can be seen that the pressure behavior along the car profile are a little affected by the velocity change due to the constant car surface profile, while the surface

pressure magnitudes range increased with the inlet flow velocity increase, but that's not conflicting with the Bernoulli's equation ''when pressure decreases in the nozzle, the velocity will increase because they both are inversely proportional''.
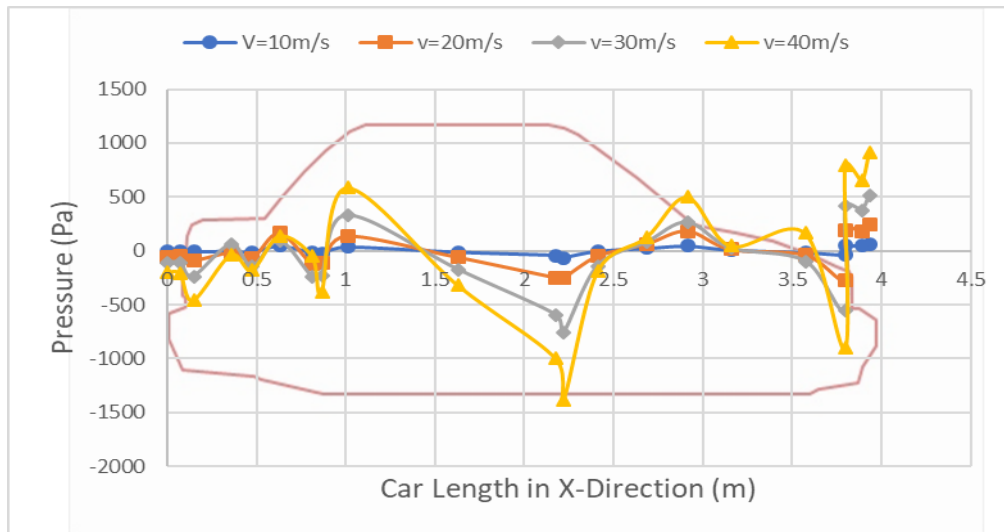

Figure 8. Experimental pressure distribution over car model at different velocities.

The experimental results of the drag force and its coefficients which is computed using the two methods; the pressure integration of the surface pressure distribution and the one-component balance methods of the base car model as shown in **Fig.9** and **Fig.10**. It is obvious that, the drag force increases as the velocity of flow is increased due to increasing the axial force of the car model while the drag coefficient decreases because it is inversely proportional to square velocity which cause the coefficient to decrease. The variation is very clear in **Fig.10** between the two methods, because that the drag coefficient in balance method indicates all the types of drag on the car whereas the pressure tap method indicate the pressure drag only.
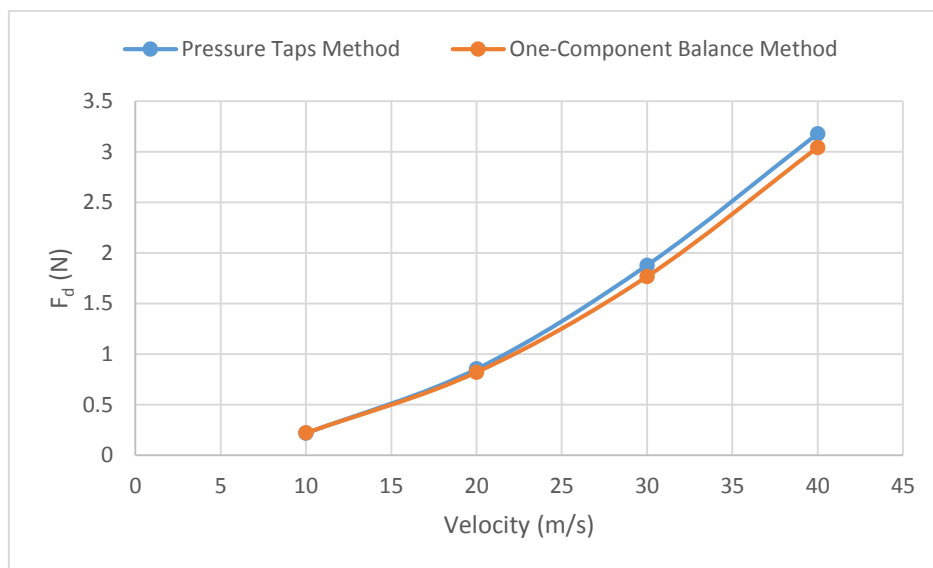

Figure 9. Comparison between pressure taps and balance methods for the drag force varies with velocity changes.
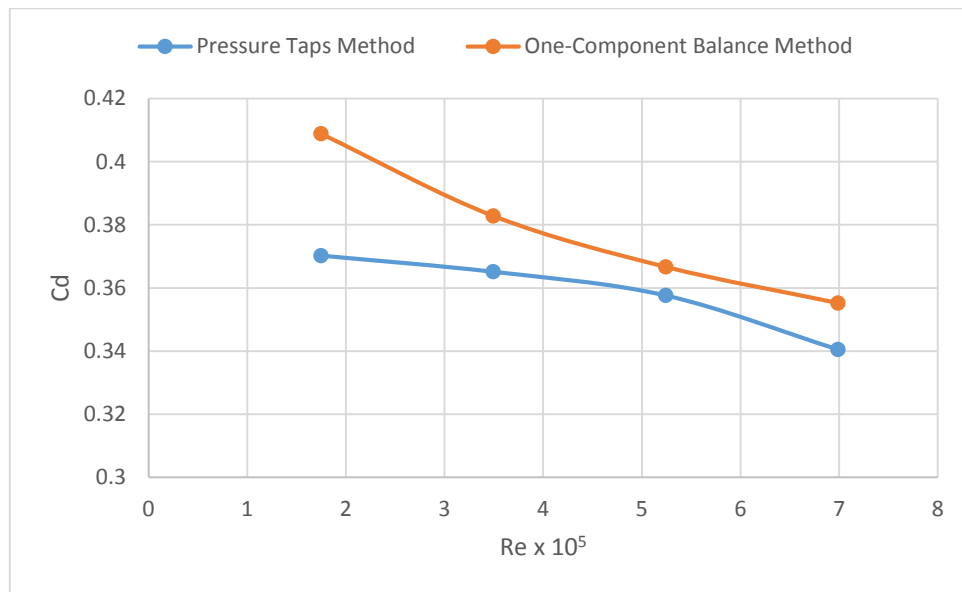
Figure 10. Comparison between pressure taps and balance methods for the drag coefficient varies with Reynolds no. Changes.

Table 1. shows the experimental results of average drag coefficient reduction when adding the modifications which were computed from one component balance method. From the table it can be shown that there is a good improvement in aerodynamic characteristics by reducing the drag coefficient, starting with little effect using VGs to the highest effect using the spoiler whereas the all modifications used together made the maximum rate in drag reduction.

Table 1. Shows the experimental results of average drag coefficients reduction when adding the modifications computed from one component balance method.

Velocity of flow (m/s)	Experimental drag coefficient (C_d)								
	C_d (base car)	C_d (Car with VGs)	$\Delta Cd\%$	C_d (Car with Spoiler)	ΔC_d %	C_d (Car with diffuser)	ΔC_d %	C_d (car for Three Modifications)	$\Delta Cd\%$
10	0.409	0.410	/	0.384	6.11%	0.4	2.20%	0.38	6.85%
20	0.383	0.386	/	0.367	4.18%	0.377	1.57%	0.363	5.20%
30	0.367	0.370	/	0.355	3.27%	0.363	1.09%	0.3505	4.50%
40	0.355	0.359	/	0.34	4.23%	0.351	1.13%	0.3362	5.30%
Average			/		4.45%		1.50%		5.46%

Fig. 11 shows the experimental results of drag coefficients computed from one component balance (by load cell connected to Arduino board). The difference in not seen clearly because the variations in the forces are very small when adding the modifications. From the figure, it can be noted that the amount of variation is very small and cannot be observed unless the range of y-axis (Cd) of the graph has been limited within the studied values only. All the modifications minimized the drag, especially when they are together, but the vortex made no clear change in drag reduction due to manufacturing problems.

In real track experiments, three tests were performed on real car and real road. The pressure distribution along the car surface were measured from the static pressure sensors. The pressure distribution results were integrated to find the drag coefficients. The test was done within three car velocities; 36,72 and 102 km/h. The pressure data were recorded using an Arduino board controller; then the results were analyzed. The data in this part of investigation recorded for different velocities (10, 20 and 30 m/s).

Fig. 12 shows the real track pressure distributions along 12 selected points on the upper surface of the real car at three different speeds. From the figure, it can be seen that the pressure behavior over the car surface profile are a little affected by the velocity change due to the constant car surface profile, while also the surface pressure values increased with the flow velocity increasing. **Fig. 13** shows a comparison between wind tunnel and real track test at 30 m/s velocities.
Fig. 14 shows a comparison between the published papers; (**Desai, et al., 2008**), (**Zala, et al., 2015**) and (**Hussain and Faris, 2017**) with the present experimental data for the drag force of the base car model at different velocities. The drag force is in a good agreement with the published experimental works and the small variation in results is due to the reduced scale used in each work and the type of model used.

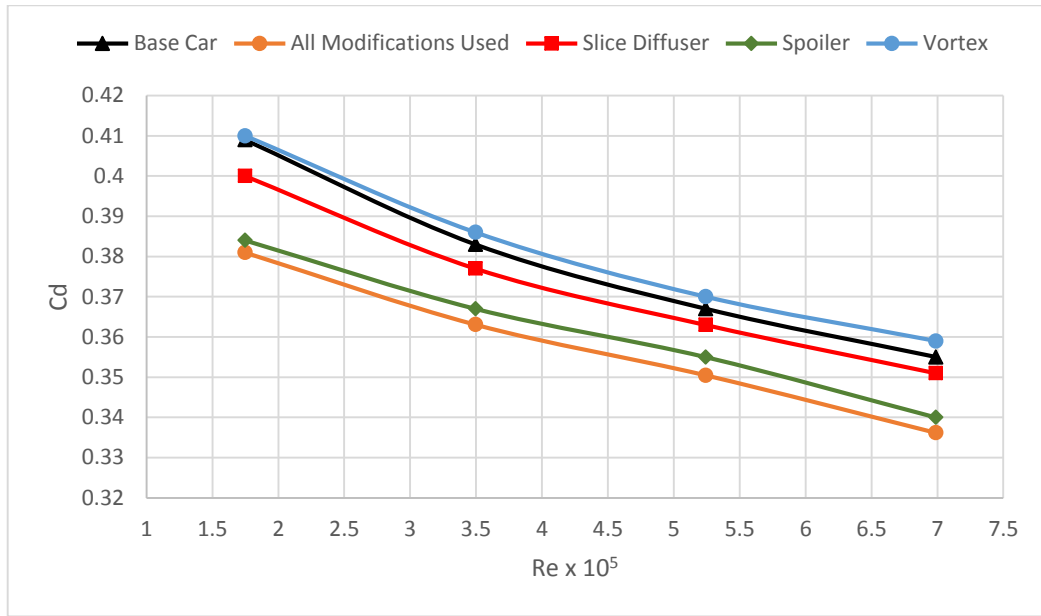**Table 2.** shows the difference in drag coefficient between the two methods of tests (wind tunnel and real track).

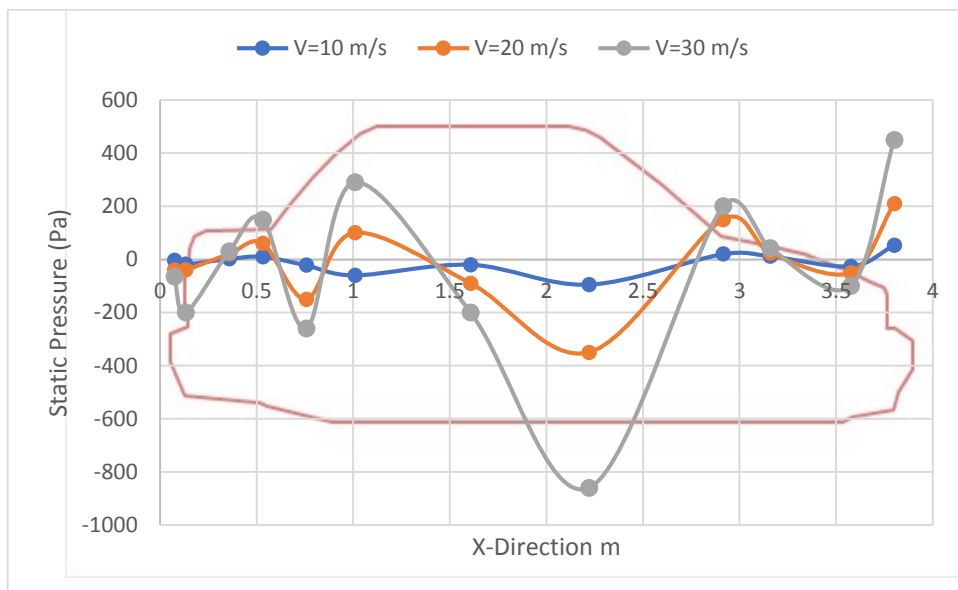**Figure 11**. Drag coefficient variation with Reynolds no. changes for different model modification cases.

Figure 12. Real track pressure distribution on the real car upper surface center line at different velocities.
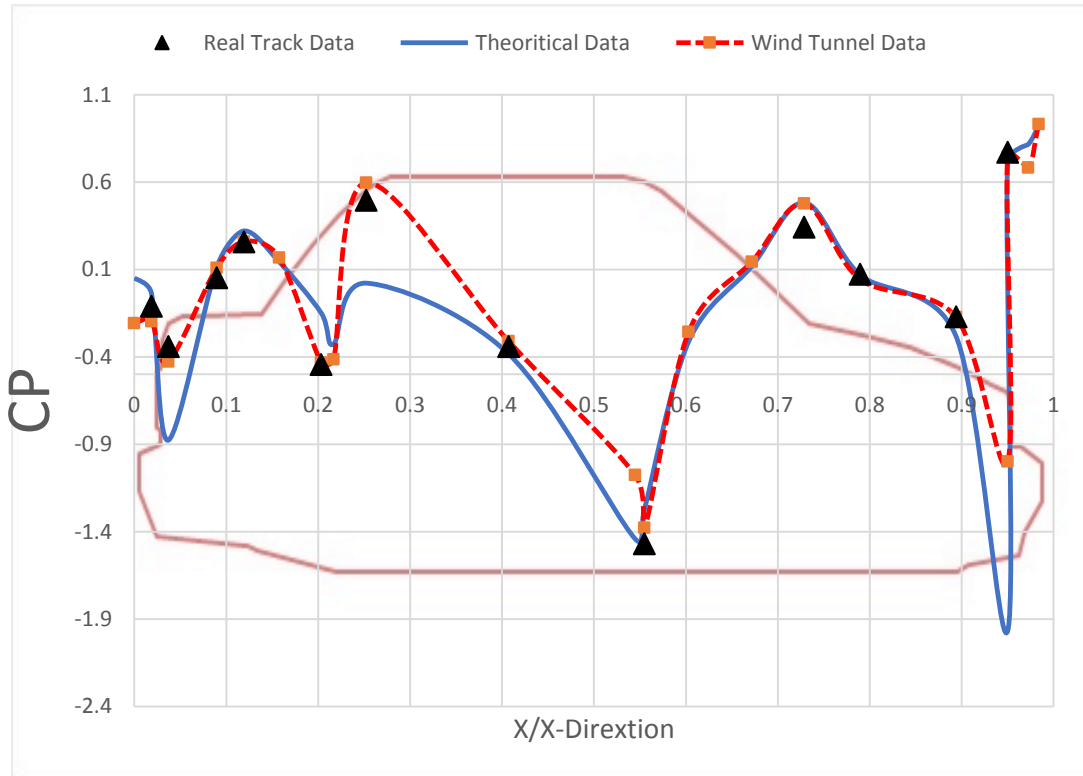
Figure 13. Real track & experimental & theoretical verification of pressure distribution for base car model at V=30m/s.
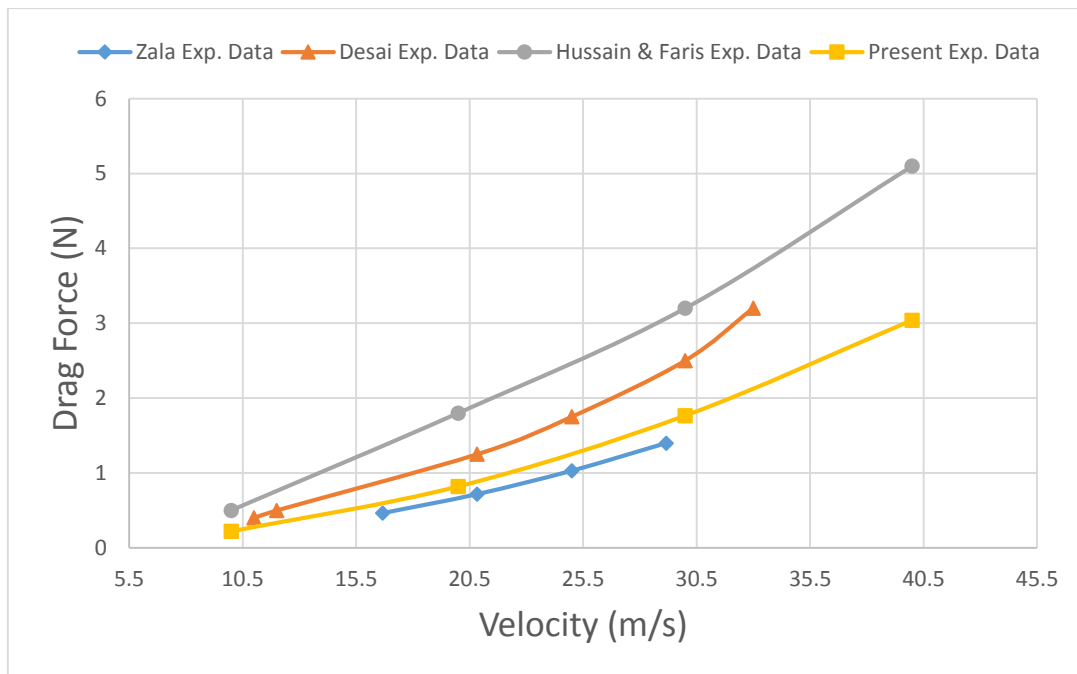

Figure 14. Verification of drag force variation for base car model

Table. 2 Comparison of Drag Coefficient for two Experimental Methods on Base Car Results

Velocity of flow (m/s)	Base Care Pressure Coefficients		
	C_d (Exp. By One component balance)	C_d (Exp. By pressure tapping)	C_d (Real Track. By pressure Sensors)
10	0.409	0.370	0.376
20	0.383	0.365	0.363
30	0.367	0.358	0.356
40	0.355	0.341	/

6. CONCLUTIONS

Two experimental approaches (wind tunnel and real track tests) were used to study the aerodynamics drag of a small model of KIA pride (SAIPA 131), and the following is the summary of results conclusion.

- The experimental results (real track tests) of (pressure distribution and drag coefficient) compared well with the experimental test (wind tunnel tests) with an average variation of 6.2%.
- The surface pressure along the upper surfaces of the car increases to freestream velocity increasing.
- The drag coefficient is gradually decreasing with the increasing speeds until the change of drag coefficient is very low at high speeds.
- The performance of the car improves as the drag coefficient decreases.
- The modifications had a reduction in drag about; 4.45% by the spoiler, 1.5% by the slice diffuser whereas the amount of the drag reduction using all the modifications together was 5.46% and recommended for another modification like front bonnet duct and double segment spoiler for the studied model.
- The velocity effects were the least significance of the drag coefficient.

NOMENCLATURE

A = area, m^2.
Ai = normal area to the pressure force for two or more pressure taps, m^2
c = constant depends on the density
Cd = drag force coefficient, dimensionless.
Cp = pressure coefficient, dimensionless.
dh = absolute error in the head of the dynamic pressure
Fd = drag force, N
g = gravitational acceleration, (m/s²)

h = relative error and it is found by logarithmic differentiation
P = pressure, Pa
p∞ = free stream static pressure, Pa
Ps = static pressure, Pa
$q∞$ = free stream dynamic pressure, Pa
v = velocity m/s
w = weight, gram
$θ$ = angle between the direction of normal pressure force and relative velocity, deg.
$ρ$ = air density, kg/m^3

REFERNCES

- Md. Rasedul Islam, Md. Amzad Hossain, Mohammad Mashud, Md. Tanvir Ibny Gias, 2013. Drag Reduction of a Car by Using Vortex Generator. *International Journal of Scientific & Engineering Research*, *Volume 4*, Issue 7.

- L. Anantha Raman, Rahul Hari H.,2016. Methods for Reducing Aerodynamic Drag in Vehicles and thus Acquiring Fuel Economy. *Journal of Advanced Engineering Research*, *Volume 3*, Issue 1, pp.26-32.

- F.J.Bello-Millán , T.Mäkelä , L.Parras b,n, C.delPino , C.Ferrera,2016. Experimental study on Ahmed's body drag coefficient for different yaw angles. *J. WindEng.Ind.Aerodyn. vol 157*, pp.140–144.

- Scott Randall Sawyer,2015.Experimental Investigation of Drag Reduction by Trailing Edge Tabs On A Square Based Bluff Body In Ground Effect . M.Sc. thesis, department of mech. Eng., California Polytechnic State University, San Luis Obispo.

- Lukáš Fryšták, Robert Popela,2016. Formula Sae Aerodynamic Optimization. Msc. Thesis, department of Mech. Eng, Brno University of Technology.

- Tomas Skrucany, Branislav Sarkan,Jozef Gnap,2016.Influence of Aerodynamic Trailer Devices on Drag Reduction Measured in a Wind Tunnel.*Niezawodnosc – Maintenance and Reliability Vol.18*,pp. 151–154.

- Damien McArthur, David Burton, Mark Thompson, John Sheridan, 2018.An Experimental Characterization of the Wake of a Detailed Heavy Vehicle in Cross-Wind. *Journal of Wind Engineering & Industrial Aerodynamics 175* pp. 364–375.

- Shankar G. and Devaradjane G.,2018. Experimental and Computational Analysis on Aerodynamic Behavior of a Car Model with Vortex Generators at Different Yaw Angles, *vol. 11*, no. 1, pp. 285–295.

- Cheng Y., Kwang-Yhee C. ,Shuhaimi M. and Abd Basid A. c,.2019, Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear

spoiler, *Journal of Wind Engineering and Industrial Aerodynamics*, *Volume 184*, Pages 305-312.

- Hussain I. Y. and Ali A. H., 2014. Calibration of Low-Speed Wind Tunnel (LSWT) Test Section. *Baghdad University Engineering Journal*, Dossier No. ME-576.

- Desai M. Channiwala S. A. and Nagarsheth H. J., 2008. Experimental and Computational Aerodynamic Investigations of a Car. *WSEAS TRANSACTIONS on FLUID MECHANICS* Issue 4, *Volume 3*.

- Zala B. H., Sorathia H. S., Dinesh L. S. and Vashant K. P., 2015. comparative assessment of drag force of sedan car model by computational fluid dynamics and experimental method. *International Journal of Advanced Engineering Technology*.

- Hussain I. y. and Saad m. F., 2017. aerodynamic characteristics of Peugeot 405 car model. MSc. thesis, department of Mech. Eng., university of Baghdad.