Comparison of Linear Vortex Panel Method and Finite Volume Method for Calculation of Generated Lift in Potential Flow Over Two-Dimensional Car Bodies

Saeid Moammaei

Islamic Azad University Sari Branch

Mehran Khaki Jamei (✉ mehran.khaki@gmail.com)

Islamic Azad University Sari Branch <https://orcid.org/0000-0002-6781-1495>

Morteza Abbasi

Islamic Azad University Sari Branch

Research

Keywords: Potential theory, 2D-car bodies, Vortex panel method, Standard CFD, Pressure coefficient, Lift coefficient

Posted Date: December 2nd, 2020

DOI: <https://doi.org/10.21203/rs.3.rs-107656/v1>

License: © ⓘ This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Comparison of Linear Vortex Panel Method and Finite Volume Method for Calculation of Generated Lift in Potential Flow over Two-Dimensional Car Bodies

Saeid Moammaei¹, Mehran Khaki Jamei^{†,1} and Morteza Abbasi¹

¹Department of mechanical engineering, Islamic Azad University, sari branch, Sari, Mazandaran. Iran

[†]Corresponding author, Email: mehran.khaki@gmail.com

ABSTRACT

This paper describes one of the aspects of the panel method to analyze the aerodynamic characteristics of a sedan. The linear vortex panel method has been developed to simulate the ideal flow over a two-dimensional arbitrary car and, it also calculates the aerodynamic load on the body. By satisfying the boundary conditions on each control point, our linear algebraic equations are obtained. The results are sensitive to the distribution of the panels over the body thus the body is broken up equally into very small panels. After solving the set of equations, the vortices strength is obtained and the pressure distribution for the upper and the lower surface of the body is calculated. The impact of the angle of attack on the aerodynamic behavior of the intended car is investigated and it is found that the lift coefficient increases with the free stream angle from -4 to 4. The accuracy of the results has been determined by checking them against the standard CFD data. The pressure distribution trend is found very much in confirmation with the CFD results, however, a discrepancy at the rear end is observed. Therefore, it can be concluded that this method does not seem practical for geometries with steep slopes in the rear part of the car. Finally, both methods are applied to the other modified geometries with lower slopes at the rear section and the results compare well with the fluent.

Keywords: Potential theory, 2D-car bodies, Vortex panel method, Standard CFD, Pressure coefficient, Lift coefficient

NOMENCLATURE

a_{ij}	influence coefficient	t	tangential vector
C_p	pressure coefficient	u, v	velocity
C_L	lift coefficient	U_∞	free stream velocity
dx, dy	panel length	x, y	Cartesian coordinate
F_x, F_y	force	μ	viscosity
n	normal vector	ρ	density
P	thermodynamic pressure	ϕ	potential function
RHS	right-hand-side of matrix	ψ	stream function
		γ	vortex strength

1. INTRODUCTION

Studying flow field as the subject of air resistance over a car is still a remarkable and unknown phenomenon and predicting a precise theoretical method is not possible by quantitative evaluation of experimental data. For years, several research and attempts have been performed in order to link different flow patterns to external forces [1]. Potential theory is one of the common and important models in fluid dynamic section that simulates various external flows by considering a Non-viscous fluid as a simplifying assumption [2]. Panel method is one of the numerical models that is widely used in aerodynamics field [3,4] and even after some decades it is still applied for preliminary designing of airplanes [5] and wind tunnels [6,7]. In this procedure the surfaces of the arbitrary bodies are covered by source panels [8], doublet [9] or a combination of them. Depending on the geometry and the way it changes, the body is broken up to smaller panels and by placing potential singularities, the external flow is simulated. In general, based on Green's formula [10] and types of boundary conditions, two different formulations are obtained, namely Neumann and Dirichlet. By satisfying the boundary conditions and depending on singularities distribution being linear or nonlinear, unknown strengths are obtained and velocity distribution around the body can be calculated. Apart from cylinders, ellipsoids [11] and different airfoils [12], Studying ground vehicle aerodynamics is another important aspect of the panel method that has received less attention. In this field, different types of geometries and being 2 [13] or 3- dimensional [14] directly affect pressure and velocity distribution. Ahmet and Hucho [15] employed a low order panel method to simulate flow over a van whose sources and doublets distribution were chosen to represent the solid surface and its wake respectively. By calculating the strength of the panels, they indicated the pressure distribution along 3 types of cross-section and centerline of the body. In some cases, the centerline section of a typical car is considered as a two-dimensional car model to

illustrate the variation of the flow structure along the body [16]. Following that, Hucho [17] focused on the aerodynamic topics and studied the difference between the pressure distribution of the viscous and non-viscous flows specifically. To solve the flow field numerically, Takagi [18] proposed the panel method as an important procedure to simulate external flow over a ground car. In the present study, we considered a two-dimensional sedan and simulated an airflow velocity of 15 m/s by using the vortex panel method. The body was covered by the small panels and different strengths of vortices were inducers of a potential flow. Finally, by calculating a pressure distribution for upper and lower surface of the car, we studied the impact of different angles of the attack and the body geometry on the generated lift forces. Written in clear, concise and grammatically correct English so that they are intelligible to the professional reader who is not a specialist in any particular field. The number of pages of each submitted manuscript must not exceed 15 pages. Manuscripts that do not conform to these requirements and the following manuscript format may be returned to the author prior to review for correction.

2. FLOW FIELD OVER THE CAR

Generally, the only mechanisms nature has for communicating a force to a body moving through a fluid are pressure and shear stress distributions on the body surface. The force and moments which act perpendicular on the body surface and pull the car opposite to its moving direction are known as drag and rolling moment. The force and moments which act parallel to the body surface are known as lift and yawing moment (Fig. 1). Depending on the car geometry and air flow behavior, the pressure changes and pressure difference on the upper and the lower side of the car leads to a force that directly effects balance and stability of the car on the road. To this end, vortex panel method is used to calculate the lift force and study the impact of different angles of the attack on the pressure distribution over the car. In this study, the airflow over the car model is considered incompressible and the rolling resistance of vehicle tires and relative velocity are neglected.

Fig. 1. Forces and moments acting upon the vehicle

2. MATHEMATIC FORMULATION

2.1 Governing equation

To calculate aerodynamic quantities, a complete set of equations which govern the fluid flow are used that contain the continuity and momentum equations for 2D Cartesian, steady state and laminar flow. The continuity equation written for the balance of mass across a control volume is defined as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

According to Newton's second law, all the forces including body forces and surface forces acting on a control volume are given by Navier–Stokes equation:

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

$$\rho \frac{\partial v}{\partial t} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (3)$$

Navier–Stokes equation shows momentum transfer in a flow that can move in two ways, advection and diffusion. On the other hand, for the ideal flow, the viscous term can be neglected and momentum is transferred just in one way. This assumption allows us to use Euler equations as:

$$\rho \frac{\partial u}{\partial t} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} \quad (4)$$

$$\rho \frac{\partial v}{\partial t} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} \quad (5)$$

$$V^r = \frac{\mathbb{I}\varphi}{\mathbb{I}x} + \frac{\mathbb{I}\varphi}{\mathbb{I}y} \longrightarrow \frac{\mathbb{I}^2\varphi}{\mathbb{I}x^2} + \frac{\mathbb{I}^2\varphi}{\mathbb{I}y^2} = 0 \quad (6)$$

$$V^r = \frac{\mathbb{I} \psi}{\mathbb{I} v} - \frac{\mathbb{I} \psi}{\mathbb{I} x} \longrightarrow \frac{\mathbb{I}^2 \psi}{\mathbb{I} x^2} - \frac{\mathbb{I}^2 \psi}{\mathbb{I} v^2} = 0 \quad (7)$$

$$V = \frac{\mathbb{I}[\varphi]}{\mathbb{I}[y]} = 0 \quad (8)$$

$$V = \frac{\|\varphi\|}{\|\psi\|} = U_{\Psi} \sin \theta \quad \text{And} \quad U = \frac{\|\varphi\|}{\|x\|} = U_{\Psi} \cos \theta \quad (9)$$

In the present study, the purpose of using a low order vortex panel method is to calculate the pressure distribution over the car. This method is based on the vortex panels substituting and satisfying the boundary conditions. Each panel as a representative of the car body, induces a velocity that impresses the other panels. The induced velocity component in the Cartesian coordinate are defined as follows [20]:

$$u_i = \frac{\gamma}{2\pi} [\tan^{-1} \frac{y_i - y_{j+1}}{x_i - x_{j+1}} - \tan^{-1} \frac{y_i - y_j}{x_i - x_j}] \quad (10)$$

$$v_i = -\frac{\gamma}{4\pi} [\ln \frac{(x_i - x_j)^2 + (y_i - y_j)^2}{(x_i - x_{i,j})^2 + (y_i - y_{i,j})^2}] \quad (11)$$

$$n_i = (\sin \alpha_i, \cos \alpha_i) \quad (12)$$

$$t_i = (\cos\alpha_i, -\sin\alpha_i) \quad (13)$$

$$u = u_i \cos \alpha_i + v_i \sin \alpha_i \quad (14)$$

$$v = -u_i \sin \alpha_i + v_i \cos \alpha_i \quad (15)$$

According to the boundary condition, summation of the normal component of induced velocity and free stream velocity must be zero and by considering the former on each panel, the influence coefficient is obtained:

$$a_{ij} = (u, v)_{ij}, n_i = [-u_{ij} \sin(\alpha_i - \alpha_j) + v_{ij} \cos(\alpha_i - \alpha_j)] + [-v_{ij} \sin(\alpha_i - \alpha_j) + u_{ij} \cos(\alpha_i - \alpha_j)] \quad (16)$$

3

$$u_{i=j} = \pm \frac{\gamma_{i=j}}{2} \quad \text{and} \quad v_{i=j} = 0 \quad (17)$$

$$a_{i=j} = a_{i=i} = \frac{1}{2} \quad (18)$$

3.2 Right Hand Side Vector

Right-hand side of the equations are related to free stream velocity and it is clear that the normal component of the free stream velocity on the body must be zero. This term of equation is known, so it is transformed to the right-hand side as:

$$RHS_i = -U_{\infty} \sin \alpha_i \quad (19)$$

3.3 Kutta Condition

This condition pointed that the closest streamlines to the body for the upper and the lower surface should reach at the trunk of the car in the same time and leave the car smoothly. For satisfying a Kutta condition, the vortex strength of the upper and lower surfaces should be equal at the rear end:

$$\gamma_N + \gamma_1 = 0 \quad \text{or} \quad \gamma_i = -\gamma_{i-1} \quad (20)$$

Now by specifying the Kutta condition and influence coefficient, we have N+1 equation with N unknown vortices strengths. So one of the equations must be deleted and Kutta condition as one extra equation is added and our algebraic set of equations are obtained as follow:

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} & \dots & a_{1N} \\ a_{21} & a_{22} & a_{23} & a_{24} & \dots & a_{2N} \\ a_{31} & a_{32} & a_{33} & a_{34} & \dots & a_{3N} \\ \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots \\ a_{N-1,1} & a_{N-1,2} & a_{N-1,3} & a_{N-1,4} & \dots & a_{N-1,N} \\ 1 & 0 & 0 & 0 & \dots & 1 \end{bmatrix} \begin{bmatrix} \gamma_1 \\ \gamma_2 \\ \gamma_3 \\ \dots \\ \dots \\ \gamma_{N-1} \\ \gamma_N \end{bmatrix} = \begin{bmatrix} RHS_1 \\ RHS_2 \\ RHS_3 \\ \dots \\ \dots \\ RHS_{N-1} \\ 0 \end{bmatrix} \quad (21)$$

4. Discretization of Body

One of the important features of designing the geometry in panel method is limitation in length and height of the intended body. In this method, the main flow components are independent of unknown parameters and the only effective factor for obtaining the vortices strengths is geometric coordinates. So, the car is placed between 0 and 1 on the X axis, -0.1 and 0.2 on the Y axis as indicated in Fig. 3. First, the car was designed in AutoCAD and due to the body shape and the way it changes, geometry is divided to the small panels as shown in Fig. 3.

Fig. 3. Discretization of car surface

Second, by writing a simple loop, centers of all panels are defined as a control point and the vortices with an unknown strength are placed there. The distance between all control points are calculated and finally their impact on each other are studied. As an example, by specifying the boundary condition for the 24th collection point (Fig. 4), the set of equations are as follows:

Fig. 4. Inducing the potential velocity with 24th control point and its impact on the other panels

$$\begin{aligned}
 & \frac{1}{2}\gamma_1 + a_{12}\gamma_2 + a_{13}\gamma_3 \dots + a_{1208}\gamma_{208} + a_{1209}\gamma_{209} - U_\infty \sin \alpha_1 = 0 \\
 & a_{21}\gamma_1 + \frac{1}{2}\gamma_2 + a_{23}\gamma_3 \dots + a_{2208}\gamma_{208} + a_{2209}\gamma_{209} - U_\infty \sin \alpha_2 = 0 \\
 & a_{31}\gamma_1 + a_{32}\gamma_2 + \frac{1}{2}\gamma_3 \dots + a_{3208}\gamma_{208} + a_{3209}\gamma_{209} - U_\infty \sin \alpha_3 = 0 \\
 & \vdots \\
 & a_{141}\gamma_1 + \dots + \frac{1}{2}\gamma_{24} + \dots + a_{14208}\gamma_{208} + a_{14209}\gamma_{209} - U_\infty \sin \alpha_{24} = 0 \\
 & \vdots \\
 & a_{2081}\gamma_1 + a_{2082}\gamma_2 + a_{2083}\gamma_3 \dots + \frac{1}{2}\gamma_{208} + a_{208209}\gamma_{209} - U_\infty \sin \alpha_{208} = 0 \\
 & a_{2091}\gamma_1 + a_{2092}\gamma_2 + a_{2093}\gamma_3 \dots + a_{209208}\gamma_{208} + \frac{1}{2}\gamma_{209} - U_\infty \sin \alpha_{209} = 0
 \end{aligned} \tag{22}$$

5. Pressure coefficient

When the strengths of the vortices are known, the perturbation tangential velocity component at each collection point can be calculated by summing the induced velocities of all the panels. In addition, the pressure coefficient for each panel is obtained as:

$$C_p(i) = 1 - \left[\frac{\gamma_i}{U_\infty \cos(-\alpha + \alpha_i)} \right]^2 \tag{23}$$

The length and the height of each panel are defined as:

$$dx_i = x_{i+1} - x_i \tag{24}$$

$$dy_i = y_{i+1} - y_i \tag{25}$$

With the pressure coefficient for each panel, normal and tangential forces are found and the total value is calculated as:

$$F_x = \sum_{i=1}^N C_p(i) dy_i \tag{26}$$

$$F_y = \sum_{i=1}^N C_p(i) dx_i \tag{27}$$

Finally Lift coefficient is defined as:

$$C_L = F_y \cos(\alpha) + F_x \sin(\alpha) \tag{28}$$

6. VALIDATION

In this study, the pressure distribution and the way it varies for the upper and lower surface for different geometries are calculated. For validation, NACA 4412 airfoil as a common case has been considered which is cambered one with sharp trailing edge. At the first step, the solver type, discretization method and basic setting are defined in the fluent as it shown in Table 1. For potential flow the inviscid model is specified, because of that, the only effective fluid characteristics is density. In the next step, the computational domain and its regions are considered. The left, top and the bottom boundaries are assumed inlet with prescribed horizontal velocity of $v=50$ m/s and the right boundary is modeled as an outlet condition (Fig. 5).

Table 1. Naca-4412 Setup-Models in fluent

Caption	Type
Solver	Pressure based
Formulation	Implicit
Space	2 Dimensional
Velocity Formulation	Absolute
Time	Steady
Model	Inviscid

Fig. 5. Specifying the boundary condition for computational domain around the Airfoil

The inlet and the outlet boundaries are placed at $12.5C$ and $20C$ from the airfoil body, respectively (C refers to the airfoil chord). In this case, quadrilateral cells are used to account different flow gradient in various direction. For monitoring the region near the body that is exposed to the complex flow features, the high-resolution mesh is needed. Fig. 6 indicates the enlargement of the computational domain with different mesh numbers around the airfoil to ensure that variation in number of elements does not affect the results. This flow field is solved at different number of elements and it is found that the variation in result from element count of 12,000 to 1,200,000 is less than 0.6 % (Table. 2). Finally, the computational domain with a structured mesh system and 12,000 nodes is picked out to monitor the pressure variation. In addition, for the panel method, the intended airfoil is simulated utilizing 120 panels. Fig. 7 illustrates the numerical predictions and experimental results [21] for the midsection of this airfoil.

Fig. 6. Enlargements of the computational domains with various grid numbers around the NACA 4412

Table 2. Impact of computational domain with various grid numbers on the lift coefficient

Number of Elements			
1	2	3	4
1.200	12.000	120.000	1.200.000
$C_l=0.5674$	$C_l=0.5937$	$C_l=0.5913$	$C_l=0.5899$

Fig. 7. Comparison of the vortex panel method and fluent results with the experimental data for Pressure coefficient distribution on an NACA 4412

The comparison of the pressure coefficients clearly proves that both panel method and CFD results agree very well with the experimental data. In general, the trend of the panel method prediction indicates that the unknown vortices strengths are calculated correctly.

In Table 3, lift coefficient of both procedures are compared. Evidently, the vortex method with less panels and limited domain can predict the potential flow over an airfoil more accurately and the result in the comparison with the CFD solution is closer to the experimental data.

Table 3. Comparison of the vortex panel method and fluent results with experimental data for calculated lift and drag coefficients

	C_L	C_d
Fluent	0.649	0.00137
Vortex Panel Method	0.588	0.0184
Experiment	0.593	0.0014

7. RESULT AND DISCUSSION

The airflow deviates when it reaches the car, depending on the car geometry and the way it changes. In the potential flow, in regard to Bernoulli's equation, variation of the velocity along the streamline is related to the variation of the pressure along the same streamline. By considering the ideal flow, the viscous effects are neglected and the fluid layers slide on each other smoothly. At the front part of the car, the closest layer to the body is divided into two parts as shown in Fig. 8. One layer as the representative of the new field passes over the top surface of the car and effects the upstream flow and the other layer passes the lower surface of the car and effects the downstream flow.

Fig. 8. Streamlines and flow patterns around the car

Therefore, the pressure distribution is presented separately for upper and lower surface of the body. Variation of the pressure for the upper surface is shown in Fig. 9. At the front bumper of the car (a), a sudden deceleration of the flow occurs, resulting in a rise of the static pressure; this point is known as the stagnation point and $C_p=1$ proves this. At the rear bumper (b), the detached streamlines reach each other. Another stagnation point is created and the flow is trapped between these two points. The pressure and velocity at the end of the car change to compensate their variation at the front section.

Fig. 9. Calculated pressure coefficient distribution on the upper and lower surfaces of the car with vortex panel method

Following that, the flow is accelerated and the static pressure gradually increases at the leading edge of the hood and along the bonnet (c and d), reaching a local maximum value at the bonnet-windshield junction (d). A sudden abatement of pressure follows and another suction peak occurs at the leading and trailing edge of the roof-windshield junction (e and f). The high velocity in these two regions is observed. Moreover, the velocity along the back screen (g) is gently reduced and due to the neglected viscous effects, the separation is not observed, and the streamlines slid on each other in any condition. Therefore, at the backscreen-trunk (g) the pressure coefficient becomes positive again and another stagnation point occurs like the front section. This feature is observed in potential flows for compensating the previous effects and finally in the mid-region of the trunk (h) the maximum velocity emerges.

For the lower surface of the car, at the front fender, similar to the upper surface, flow is accelerated and pressure remains unchanged between points of (k) and (m). Along this part of the body, the local velocity approximately equals the freestream velocity. Finally, at the rear fender, the pressure reaches the local negative peak again. In addition, the impact of the angle of attack on the flow field and the generated lift is studied. Figures 10 and 11 show variations of the pressure coefficient along the upper and lower surface of the car for different angles of attack. According to Fig. 10, it is clear that the variation of the angle of attack directly effects the pressure distribution for the front part of the car in comparison with the rear part.

At the end of the car, the pressure lines for different angles become closer and a similar velocity is observed. Considering the inviscid fluid as the simplifying assumption is the main reason of this event, because of that, the only way that the momentum can be transferred is an advection. Additionally, Fig. 10 expresses this reality that the airflow tends to lift the car from the front part, because of that, for race cars, most accidents occur when the lift force exceeds the car weight and unexpectedly lifts it from the front part [22].

The effect of the angle of attack upon pressure coefficient along the lower surface is illustrated in Fig. 10 and the calculated result is in accordance with the body shape truly. For the low angles, oncoming airflow comes in contact with the front part of the hood and a negative force for this special part pushes the car to the ground.

Fig. 10. Impact of the angle of attack on pressure coefficient distribution on the upper surface of the car

Fig. 11. Impact of the angle of attack on pressure coefficient distribution on the lower surface of the car

The calculated lift coefficients for different angles of attack are given in Table 4. It is found that the lift force, generated due to the asymmetric pressure distribution along the upper and lower surfaces, increases with the free stream angles from -4 to 4.

In this study, the vortex panel method is employed to calculate lift coefficient for different angles of attack. In order to validate the result, a potential flow over the intended car is simulated and solved by fluent with the same conditions and setup-models from above solution. An enlargement of the computational domain with different structured grid numbers around the car are shown in Fig. 12. For the minimum grid-spacing in the y and x-direction 0.00472L is chosen as the fine mesh at the surface of the car to monitor the curvatures in the geometry and the coarser mesh is used away from the vehicle.

Table 4. Calculated lift coefficients with vortex panel method at different angles of attack for the sedan

Alpha	C_L
-4	-0.18
-3	-0.05
-2	0.066
-1	0.19
0	0.31
1	0.44
2	0.7
3	0.69
4	0.82

Fig. 12. Enlargements of the computational domains with various grid numbers around the car

The geometry is positioned between $X/L = 0$, and $X/L = 1$, where L is the length of the car and the free stream velocity was taken as 15 m/s. Block-structured grids are employed and the classification of the computational domain into several segments is shown in Fig. 13.

Fig. 13. Specifying the boundary condition for computational domain around the car

The inflow and outflow boundaries are located twenty body-length upstream and 30L downstream and taken as the velocity-inlet and pressure-outlet respectively. The upper and lower boundaries are located 20L above and under the body. The effect of the grid refinement and the bigness of coarse and fine grids around the car on the pressure coefficient distribution are shown in Fig. 14. The results for the various grid numbers do not show significant differences except in some parts.

Fig. 14. Impact of computational domain with various grid numbers on the pressure coefficient distribution on the upper surface of the car

A comparison between the pressure distribution obtained by the panel method and CFD solution is illustrated in Fig. 15. Clearly, both methods indicate good agreement with each other but after $x=0.87$, discrepancies are observed at the end of the trunk and vortex panel method has predicted a high velocity in comparison with the CFD solution at the rear.

Fig. 15. Comparison of the vortex panel method with fluent result for Pressure coefficient distribution on the upper and lower surfaces of the car

The present disagreement for the rear section is related to the limitation of the panel method for the special geometries with the steep slope at the rear end. Generally, low order panel method is convenient for the geometry where the lower and the upper surface of the body are attached together with low slope at the end part of the car. Therefore, both methods are applied to the other car geometry with the lower slope at the rear section. Fig. 16 demonstrates the variation of the slope angles at the rear end of the new cars' geometries.

Fig. 16. Variations of the slope angle at the rear end of new car geometries

Fig. 17 shows the influence of shape modifications and slope angle on the pressure distribution along the body of the car and by comparing the calculated result, the agreement of the pressure variation is observed. As the slope angle decreases, the results agreements increase. In addition, apart from the rear section, geometry of the front part of the car, especially the bonnet and windshield-junction designing, directly affect the flow structures passed over the body. That is why body modifications such as removing the sharp segment and the lower slopes determine the flow behavior.

Finally, Fig. 18 and Table. 5 illustrate the impact of the angle of the attack on the lift coefficient for all the car configurations considered in this study. By comparing the results obtained by using panel method and CFD solution, it is found that the car geometry modification can decrease the generated lift. In addition, reduction of the rear slope angel is playing an important role in overcoming the panel method limitation. As shown in Fig. 18, for the first body the predicted lift coefficient is in disagreement with the CFD value but for the second and third bodies, by reducing the angle of slope at the rear end, good agreement is seen in the result trend.

Fig. 17. Comparison of the vortex panel method with fluent results for Pressure coefficient distribution on the upper and lower surfaces of the new car geometries

Table 5. Calculated lift coefficients with vortex panel method at different angles of attack for 3 types of car

	Alpha	-4	-3	-2	-1	0	1	2	3	4
First Car	C_L	-0.18	-0.05	0.066	0.19	0.31	0.44	0.57	0.69	0.82
Second Car	C_L	0.009	0.107	0.18	0.23	0.26	0.32	0.34	0.4	0.56
Third Car	C_L	-0.07	-0.06	-0.04	0.01	0.05	0.16	0.25	0.33	0.45

Fig. 18. Comparison of the vortex panel method with fluent results for generated lift coefficient by 3 types of car geometries at the different angles of attack

8. CONCLUSIONS

A numerical study of the flow over a two-dimensional car model is presented. The computations are preformed using a low order vortex panel method to simulate the potential flow. A free stream velocity of 15 m/s was considered. The body was broken up to the 225 panels equally and different vortex strengths induced a velocity to each panel. The CFD analysis for calculating a pressure coefficient and generated lift is also presented to show the validity of the panel method. The results are summarized as follows:

1. For the validity and accuracy of the panel code, NACA 4412 airfoil as a common benchmark case is chosen. The grid refinement does not significantly change the structure of the computed flow field and the panel method result in the comparison with the CFD solution is closer to the experimental data.
2. Variation of the pressure coefficient obtained by using the panel method indicates the satisfactory prediction of the flow structure over the car.
3. The pressure distribution trend is found very much in confirmation with the CFD solution but discrepancies at the end of the trunk are observed leading to discord in predicted lift coefficients.
4. The disagreement for the rear section is related to limitation of the low order panel method for the special geometries. To overcome this problem, the lower and the upper surfaces of the body must be attached with a lower slope at the rear end.
5. For both modified car geometries, the results are in qualitative agreement with CFD analysis and reduction of the generated lift is obtained
6. The lift coefficients for the third modified car geometry at the different airflow angles where data rates are good to moderate are estimated to have an error of $\pm 7\%$ from the CFD results.
7. Coefficient of lift is found to increase with the free stream angel of attack from -4 to 4.
8. Oncoming airflow angle, fender design and hood slope at the front part of the car has an important role to determine flow field at the rear section. Changing the angle of attack prove that the car tends to lifted from front part and the same velocities are seen at the end.

DECLERATIONS

Availability of data and materials: No data and materials are available.

Competing interests: The authors declare that they have no competing interests.

Funding: The authors received no financial support for the research, authorship, and/or publication of this article.

Authors' Contribution: All persons who meet authorship criteria are listed as authors, and all authors certify that they have participated sufficiently in the work to take public responsibility for the content, including participation in the concept, design, analysis, writing, or revision of the manuscript.

Acknowledgments: No acknowledgment is available.

REFERENCES

1. Carr G (1968) The Aerodynamics of Basic Shapes for Road Vehicles. Part 2: Saloon Car Bodies .
2. Hirt C, Ramshaw J (1978) Prospects for numerical simulation of bluff-body aerodynamics. In: Aerodynamic Drag Mechanisms of Bluff Bodies and Road Vehicles .Springer, pp 313-355
3. Hess J (1990) Panel methods in computational fluid dynamics. Annual Review of Fluid Mechanics 22 (1):255-274
4. Halsey N (1979) Potential flow analysis of multielement airfoils using conformal mapping. AIAA Journal 17 (12):1281-1288
5. Erickson LL (1990) Panel methods: An introduction .
6. Browne L Application of panel methods to wind-tunnel wall interference corrections. In: 28th Aerospace Sciences Meeting, 1990. p 7
7. Lee K (1981) Numerical simulation of the wind tunnel environment by a panel method. AIAA Journal 19 (4):470-475
8. Hess JL (1962) Calculation of potential flow about bodies of revolution having axes perpendicular to the free-stream direction. Journal of the Aerospace Sciences 29 (6):726-742
9. Albano E, Rodden WP (1969) A doublet-lattice method for calculating lift distributions on oscillating surfaces in subsonic flows. AIAA journal 7 (2):279-285
10. Morino L, Kuo C-C (1974) Subsonic potential aerodynamics for complex configurations: a general theory. AIAA journal 12 (2):191-197
11. Hess JL, Smith A (1962) Calculation of non-lifting potential flow about arbitrary three-dimensional bodies. Douglas Aircraft Co Long Beach CA ,
12. Moran J, Cole K, Wahl D (1980) Analysis of two-dimensional incompressible flows by a subsurface panel method. AIAA Journal 18 (5):526-533
13. Angelis W, Drikakis D, Durst F, Khier W (1996) Numerical and experimental study of the flow over a two-dimensional car model. Journal of wind engineering and industrial aerodynamics 62 (1):57-79
14. Chometon F (1983) Calculating three-dimensional separated flow around road vehicles. International Journal of Vehicle Design, Technological Advances in Vehicle Design Series, SP3, Impact of Aerodynamics on Vehicle Design:374-386
15. Ahmed S, Hucho W-H (1977) The calculation of the flow field past a van with the aid of a panel method. SAE Transactions:1672-1692
16. Stafford L (1973) A Numerical Method for the Calculation of the Flow Around a Motor Vehicle. Advances in Road Aerodynamics BHRA Fluid Engineering:167-183
17. Hucho W (1987) Aerodynamics of Road Vehicles: from fluid mechanics of road Vehicle engineering. Buferworths 2:46-53
18. Takagi M (1990) Application of computers to automobile aerodynamics. Journal of Wind Engineering and Industrial Aerodynamics 33 (1-2):419-428
19. Anderson Jr JD (2010) Fundamentals of aerodynamics. Tata McGraw-Hill Education ,
20. Katz J, Plotkin A (2001) Low-speed aerodynamics, vol 13. Cambridge university press ,
21. Abbott IH, Von Doenhoff AE (2012) Theory of wing sections: including a summary of airfoil data. Courier Corporation ,
22. Katz J (1995) New directions in race car aerodynamics: designing for speed. Bentley Publishers ,

Figures

Figure 1

Forces and moments acting upon the vehicle

Figure 2

Boundary conditions in an ideal flow

Figure 3

Discretization of car surface

Figure 4

Inducing the potential velocity with 24th control point and its impact on the other panels

Figure 5

Specifying the boundary condition for computational domain around the Airfoil

Figure 6

Enlargements of the computational domains with various grid numbers around the NACA 4412

Figure 7

Comparison of the vortex panel method and fluent results with the experimental data for Pressure coefficient distribution on an NACA 4412

Figure 8

Streamlines and flow patterns around the car

Figure 9

Calculated pressure coefficient distribution on the upper and lower surfaces of the car with vortex panel method

Figure 10

Impact of the angle of attack on pressure coefficient distribution on the upper surface of the car

Figure 11

Impact of the angle of attack on pressure coefficient distribution on the lower surface of the car

Figure 12

Enlargements of the computational domains with various grid numbers around the car

Figure 13

Specifying the boundary condition for computational domain around the car

Figure 14

Impact of computational domain with various grid numbers on the pressure coefficient distribution on the upper surface of the car

Figure 15

Comparison of the vortex panel method with fluent result for Pressure coefficient distribution on the upper and lower surfaces of the car

Figure 16

Variations of the slope angle at the rear end of new car geometries

Figure 17

Comparison of the vortex panel method with fluent results for Pressure coefficient distribution on the upper and lower surfaces of the new car geometries

Figure 18

Comparison of the vortex panel method with fluent results for generated lift coefficient by 3 types of car geometries at the different angles of attack