Numerical study on aerodynamic drag reduction and energy harvest for electric vehicle: a concept to extend driving range

A W Huluka\textsuperscript{1} and C H Kim\textsuperscript{2,*}

\textsuperscript{1} Department of Automotive Engineering, Graduate School, Seoul National University of Science & Technology, 232 Gongneung-ro, Nowon-gu, Seoul 01811, Korea
\textsuperscript{2} Department of Mechanical & Automotive Engineering, Seoul National University of Science & Technology, 232 Gongneung-ro, Nowon-gu, Seoul 01811, Korea

\textsuperscript{*} Corresponding author: profchkim@seoultech.ac.kr

Abstract. Energy efficiency of road vehicles is highly attributed to its aerodynamic performance. Over the last decades, significant reduction of aerodynamic drag has been made. But, beyond the current achievement, further improvement become a challenge for car industries and professionals working in the area. This paper focuses on new approach to further find aerodynamic performance improvement particularly for electric vehicles. Therefore, a preliminary numerical study on ducting and concept of energy harvesting from aerodynamic resistance is introduced as an alternative to improve energy efficiency. Ducted and slightly modified Ahmed model is used to study aerodynamic characteristics of ducted models and how ducting would contribute to the reduction of energy consumption due to air resistance. Three-dimensional, incompressible, and steady flow governing equations are solved by CFD code (PHOENICS ver.2018) with extended turbulent model proposed by Chen-Kim (1987). From the study, it is found that the total drag reduction of over 15% on the ducted models compared to the base model. The kinetic energy preserved due to ducting is also significant. Harvesting this energy and utilizing it to extend driving range is believed to be a potential area to be dealt in detail. Therefore, the idea of duct application and energy harvesting introduced in this paper, either to reduce or use aerodynamic resistance is expected to have potential contribution to the development of future energy efficient road vehicles.

1. Introduction

Energy efficient vehicles are required to meet future emission and fuel consumption requirements. Environmental issues and public health security concerns have forced car industries and the public to look for alternatives to petroleum based means of transportation. In recent decades, the research and development activities related to transportation have emphasized the development of high-efficiency, clean and safe transportation. Electric vehicles, hybrid vehicles and fuel cell vehicles have been typically proposed to replace conventional vehicles in the future [1], [2].

Currently attention given to electric vehicle development is motivating and it is deemed to be a future car as it is energy efficient, no direct carbon emission, and low noise pollution. On the other hand, the limited driving range per charge is the challenge of current electric vehicles due to low energy density of the battery. To improve the driving range, one alternative is reducing the energy consumption of the moving vehicle. Therefore, identifying the energy consumption attributes of the moving vehicle and conducting a research on how to reduce consumption of energy would enhance the driving range of the
vehicle. The largest portion of power driving the vehicle is consumed to overcome the air resistance \[2\], \[3\], \[4\], \[5\]. Aerodynamic drag force typically accounts for 65% of the total resistance for medium sized car at 100 km/h \[2\], \[4\], \[5\], \[6\]. Hence, reducing drag force contributes significantly to the energy efficiency of battery electric vehicles \[7\], \[8\].

The idea under consideration is harvesting energy from the air resistance without affecting the base drag. As the vehicle is moving in a sea of air, the kinetic energy of the fluid relative to the vehicle is significant particularly at higher vehicle speed. As a first step for a feasibility study, ducting and its aerodynamic effect on a moving vehicle is discussed using simple body. A numerical simulation was made as a preliminary study to either reduce or use aerodynamic resistance of road vehicle

2. Analytical theory

Theoretical background on road-load power of a moving vehicle, ducting and energy harvesting concept from the aerodynamic resistance is described here in short.

2.1. Road Load Power and Aerodynamic Drag

The total power \(P_{\text{batt}}\) required to overcome resistance forces for an electric vehicle of mass \(M_v\) cruising at a speed \(V\), can be expressed as:

\[
P_{\text{batt}} = \eta_t \eta_m \left( M_v g f \cos \theta + \frac{1}{2} \rho_v C_D A_f V^2 + M_v g \sin \theta + M_v \beta V \frac{dV}{dt} \right) \tag{1}
\]

Where \(\eta_t\) - efficiency of transmission, \(\eta_m\) - traction motor efficiency, \(g\) - acceleration due to gravity, \(\delta\) - mass factor, \(\theta\) - slope or grade, \(f_r\) - rolling resistance coefficient, \(C_D\) - drag coefficient, \(\rho_v\) - air density. The terms in the bracket represent rolling resistance, aerodynamic resistance, grade resistance and inertial resistance to the vehicle motion from left to right respectively. In a high speed driving situation, the power required to overcome the aerodynamic resistance is extremely significant consuming energy to keep its speed. This drag force \(F_D\) is a function of the frontal projected area \(A_f\) of the vehicle, drag coefficient \(C_D\) and the speed of the vehicle \(V\).

\[
F_D = \text{fun}(C_D, A_f, V, \rho_v) \tag{2}
\]

Therefore, the drag force is expressed as

\[
F_D = C_D \frac{1}{2} \rho_v V^2 A_f \tag{3}
\]

Two main components of the drag force are pressure and skin friction force. Aerodynamic resistance of a ground vehicle is dominated by pressure drag whilst skin-friction drag is dominant on aircraft and marine ships \[9\], \[10\], \[11\].

2.2. Ducting

The frontal area of internal combustion engine powered vehicles is opened with the meshed grill to induct incoming air into the engine room for engine cooling. To the contrary the frontal panel of most electric vehicles is closed by design and made up of single panel Figure 1. But the closed frontal panel of electric vehicle would have considerable effect on the increment of aerodynamic resistance of the vehicle.

Figure 1. Comparison of the front panel shape of two type of sedans, source \[12\].
Ducting is not common in ground vehicle as a drag reduction technology. Here in this paper a deliberate duct for either drag reduction or energy harvesting system is made for feasibility study. This kind of approach is believed to be applicable in electric vehicles as mostly their front panel is closed by design.

2.3. Aerodynamic Energy Harvest

When a vehicle moves at higher speed, serious air resistance which is called as drag exerts on the moving vehicle due to the relative speed. If this opposing aerodynamic force acting on a vehicle body due to the momentum loss is reduced or captured by any means, two very positive effects will be expected; fuel savings and emission reduction. If the momentum energy can be recovered, energy efficiency of road vehicle will increase. The energy harvesting concept can be modelled with a radial type power turbine as shown in Figure 2.

![Figure 2. Schematic drawing of the wind energy harvesting system in an air duct.](image)

General conservation of momentum for this model can be stated as the linear momentum equation and it is equal to external forces ($F_{ext}$) acting on the control volume (CV) indicated by red broken circle in Figure 2.

$$\sum F_{ext,CV} = \frac{\partial}{\partial t} \int_{CV(t)} \rho v d\mathcal{V} + \int_{CS(t)} \rho v (\mathbf{v} \cdot \mathbf{n}) dA$$

(4)

The force exerts on the power turbine due to the change in momentum at inlet and outlet will be used to calculate the torque ($T$) of the turbine ($dT = F_t \times dr$). The torque multiplied by the angular velocity of the turbine will give the power harvested from the aerodynamic resistance.

$$Power = \frac{2\pi NT}{60}$$

(5)

Where $N$ is the rotational speed and $T$ is the torque of the power turbine.

3. Description of the test model

Ahmed model [10] is used for this fundamental study with some modifications. A (4:1) scale model is used with the dimension 4.176 m x 1.556 m x 1.152 m (L x W x H), Figure 3. Five different models are defined with different size of duct as specified in Table 1.

| Model No. | Specification of the model | Area ratio$^a$ |
|-----------|----------------------------|----------------|
| M0        | Base model with no duct    | 0%             |
| M1        | Model with 1072 mm x 160 mm (W’xH’) duct | 9.6%          |
| M2        | Model with 1072 mm x 240 mm duct | 14.4%         |
| M3        | Model with 1072 mm x 320 mm duct | 19.1%         |
| M4        | Model with 1072 mm x 400 mm duct | 23.9%         |

$^a$Area Ratio($R_A$); $R_A = \frac{A_d}{A_f}$ where $A_f$ is the frontal projected area of the base model and $A_d$ is the cross-sectional area of the duct ($A_d = H’ x W’$, $H’$ is duct height and $W’$ is duct width)
4. Numerical scheme and its conditions

In this study, Finite Volume Method (FVM) was used to perform the numerical calculation. The general-purpose CFD code, PHOENICS (ver. 2018) was used for a numerical simulation of the turbulent incompressible flow field. 3-dimensional Naiver-Stokes equations were solved with KECHEN turbulent model (Chen-Kim κ-ε model) which is a modified standard κ-ε turbulent model. The turbulent no slip condition near solid boundary was modelled by log law.

Chen-Kim κ-ε model believed to improve the dynamic response of the Epsilon equation by introducing an additional time scale and source term. In addition, several of the standard-model coefficients are adjusted so that the model maintains good agreement with experimental data on classical turbulent shear layers [13].

4.1. Numerical domain and its numerical grid

The numerical domain size is defined as shown in Figure 4. Especially 8L is given at the rear side of the model to have a stable convergence on the simulation result. The initial and boundary conditions are defined as given in the Table 2.

| Boundary surface   | Boundary & initial conditions                                      |
|--------------------|--------------------------------------------------------------------|
| Inlet              | Velocity inlet (60, 80, 100, 120 km/h)                             |
| Outlet             | Pressure outlet                                                    |
| Sides and top      | No slip wall condition                                             |
| Ground             | moving ground type                                                 |
| Model exterior surface | No slip wall                                                       |
| Duct surface       | No slip wall                                                       |
| Flow domain        | Quasi-3D flow, Turbulent and incompressible flow                   |

Table 2. Initial and boundary conditions.
With the aforementioned set up, numerical simulation has been done to study the duct effect on drag force, rear flow and related flow property variations. The calculation was conducted at optimized grid distribution Figure 5 which is obtained by thorough grid dependence evaluation, with different iterations and very nicely converged with an error cut off less than 0.01%. For more precise profile and boundary layer study very fine and highly dense grid distribution is also used in the study. In PHOENICS, the mesh generation does not need third party package when using Cartesian and polar coordinates and six sided rectangular brick element is used in this study.

4.2. Governing equations:
For steady incompressible turbulent flows, the flow field is characterized by the conservation laws (since the flow is assumed steady the time derivative component removed). Hence the governing equation for turbulent flow field is written:

1) Continuity equation:
\[ \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} + \frac{\partial U_k}{\partial x_k} = 0 \] (6)

2) Momentum equation:
\[ \frac{\partial}{\partial x_i} \left( \rho U_i U_j \right) = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{1}{2} \frac{\partial U_k}{\partial x_k} \right] - \rho \dot{\varepsilon}, \] (7)

where,
\[ u_{ij} = v \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}, \]
\[ \delta_{ij} = 1 \text{ if } i = j \text{ and } \delta_{ij} = 0 \text{ if } i \neq j \]

3) Extended \( \kappa-\varepsilon \) closure turbulent model (KECHEN)
- Turbulent kinetic energy equation
\[ \frac{\partial}{\partial x_i} \left( U_j k \right) = \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + G - \varepsilon \] (8)

\[ G = -u_{ij} \frac{\partial U_i}{\partial x_j} \] (production rate due to deformation) [14],[15], \[ v = C_\rho \frac{k^2}{\varepsilon}, \quad C_\mu = 0.09 \]

- Energy dissipation equation
\[ \frac{\partial}{\partial x_i} \left( U_j \varepsilon \right) = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_\rho G) - C_{\varepsilon \rho} \frac{\varepsilon^2}{k} + C_{\varepsilon \sigma} \frac{P_e^2}{\rho k} \] (9)

Where: \( C_\rho = 0.09, \quad C_{\varepsilon \rho} = 1.15, \quad C_{\varepsilon \sigma} = 1.9, \quad C_{\varepsilon \omega} = 0.25 \quad \sigma_k = 0.75, \quad \sigma_\varepsilon = 1.25 \), \( k \) is turbulent kinetic energy, \( \varepsilon \) is energy dissipation rate, \( V_i \) & \( V_j \) laminar and turbulent kinematic viscosities, \( P_e \) is the volumetric production rate of kinetic energy by shear forces, \( \sigma_k \), \( \sigma_\varepsilon \) Prandtl number connected to diffusivity of \( K \) and \( \varepsilon \) to eddy viscosity, \( f_j \) is the Lam-Bremhorst (1981) damping function which tends to unity at high turbulence Reynolds numbers [16].

5. Result and discussion
The force distribution over the model is evaluated under ‘integrated forces’ indicating pressure force and friction forces. Based on this integrated force results, evaluation of the duct flow effect on each and every side is made possible. Special attention is given to the drag force being contributed from pressure force and skin friction force. The frontal pressure force and rear side drag force due to strong and counter rotating trailing vortex.

5.1. Aerodynamic flow Characteristics of the model
Aerodynamic characteristics of the flow around the model is demonstrated using the static pressure contour plot as depicted in Figure 6. The pressure distribution at front and rear of the un ducted and ducted model are different due to the duct effect. As shown in Figure 6, M1 has lower front stagnation pressure than the fully blocked body and higher rear pressure due to the flow momentum through the duct is injected to the rear wake resulting in reduced rear pressure drag. As the duct size increases, the
decreasing trend of front stagnation pressure continues. Similarly the rear pressure drag decreases with duct size.

5.2. Pressure Drag Force
Effect of the opened front panel on pressure drag force is evaluated in detail. The total pressure drag curve versus models with different duct size is shown in Figure 7. All pressure drags are nondimensionalized by the base model (M0) maximum pressure drag which is at 120 km/h. The highest pressure force is observed on the model M0. As the duct size increases the pressure force magnitude is decreasing. Quantitatively, for model M1 at a speed of 120 km/h about 20.5% reduction of total pressure drag in reference to the base model is observed. This shows that the duct has a very significant effect on a pressure drag reduction of the bluff body under consideration.

5.2. Pressure Drag Force
Effect of the opened front panel on pressure drag force is evaluated in detail. The total pressure drag curve versus models with different duct size is shown in Figure 7. All pressure drags are nondimensionalized by the base model (M0) maximum pressure drag which is at 120 km/h. The highest pressure force is observed on the model M0. As the duct size increases the pressure force magnitude is decreasing. Quantitatively, for model M1 at a speed of 120 km/h about 20.5% reduction of total pressure drag in reference to the base model is observed. This shows that the duct has a very significant effect on a pressure drag reduction of the bluff body under consideration.

![Figure 6. Pressure distribution on M0 (top) and M1 (bottom) at 100 km/h.](image1)

![Figure 7. Variation of the pressure drag with the duct size at different speed.](image2)
5.3. Skin friction drag

The friction drag generated due to the skin friction shear stress at the boundary layer near the surfaces of the model solid surfaces is relatively small. During this study, contrary to the pressure drag force, the friction drag force of the ducted models revealed an increasing trend as duct size increases. Figure 8 demonstrates the friction drag trend in each model at four different test speeds. The model with no duct (base model) experienced relatively the smallest friction drag. This could be due to increased surface area and flow boundary layer created in each inner faces of the duct. Considering one of the ducted models with smallest duct size (M1) at speed of 100km/h a net total drag reduction of about 16.3% is observed quantitatively. The pressure drag force is reduced by 19.5% while the skin friction drag is increased by 3.2% of the total drag. As the duct size increases, the friction force increases as shown in Figure 8. This can be considered as side effect of ducting as far as drag reduction is concerned while the pressure force which is the dominant drag force in ground vehicle exhibit a decreasing trend. Since the contribution of skin friction to the total drag is very low, its increase with duct size did not affect much the decreasing trend of the total drag force.

![Figure 8. Variation of fiction drag with the duct size at different speed.](image)

5.4. Drag coefficient

The coefficient of drag of the models also showed a reduction trend even though coefficient of drag is directly proportional to drag force and inversely proportional to reference area \((A_f)\), reduction of both area and drag force by ducting does not assure a reduction in \(C_D\). The relative change rate of area and change rate of drag force is a determining factor for reduction of the \(C_D\) of ducted model. As shown in Figure 9, about 15.3\% of \(C_D\) was reduced in M4 compared to the base model (M0) while M1 \(C_D\) is reduced by about 7.8\% in same speed range. The \(C_D\) on the Figure 9 is nondimensionalized by the \(C_{D_{0}}\) of the base model. It should be noticed that an opened front panel on a moving body should have energy saving effect than the closed one and will contribute to driving range extension.
5.5. Aerodynamic energy
The velocity profile just before the entry, at the entry, inside duct, at the exit and just after the exit of the duct gives some insight about the flow dynamics and energy level at the duct. To have a big picture of the flow domain along duct, a wider range velocity profile is plotted as depicted on Figure 10. The profile is captured within a 10 m long distance in the domain passing through the middle of the duct length to see the pattern of the flow speed.

At about 3 m in front of the duct, the velocity assumes a free stream velocity (27.78 m/s) which is a test speed of 100 km/h. This velocity declines as it approaches the object (model) front side and sharply drop to about 65% of the free stream velocity near the entry edge of the duct and then raise back to about 86% at the entry. Just after entry it slightly decelerates until it attain smooth flow inside the duct where it gradually accelerate. At the exit, it drastically decelerate again as it is exposed to a chaotic tubulent flow at the rear of the object until it mix with and maintain the outside rear flow structure about 3 m behind the rear end. Without a duct, the kinetic energy of the whole bulk flow infront of the model is stagnated at the front exerting a resistance to the objec. But now it can be seen in the profile plot that some amount of the flow kinetics is maintained and channeled to the rear through duct reducing the

![Figure 9. Variation of drag coefficient with the duct size at different speed.](image)

![Figure 10. Longitudinal velocity profile through model duct center of M1.](image)
aerodynamic resistance. This kinetic energy of the air can be harvested using the principle discussed in Subsection 2.3 of this article. Considering the typical scenario power to be harvested is roughly estimated for model M1.

The calculation is performed with the following assumption. Air density is uniform throughout the duct. As the flow is guided to the turbine by duct, the turbine converts major portion of the kinetic energy in the air to mechanical energy turning the generator. Because it is a guided flow not in an open air turbine. Turbine is directly connected to the alternator mechanically. Therefore, the potential aerodynamic kinetic energy by the air flowing through a duct is estimated by:

\[
\text{Kinetic Energy} = \frac{1}{2} \rho_a A_d \bar{U}^3
\]  

The average air velocity \( \bar{U} \) inside the duct: 
\[
\bar{U} = \frac{\int V_d dA_d}{A_d}
\]

Available power harvested:
\[
P_{\text{available}} = \frac{1}{2} \rho_a A_d \bar{U}^3 \eta_T \eta_G
\]  

Where \( \eta_T \) & \( \eta_G \) are efficiency of turbine and generator respectively. Table 3 shows the electric energy harvested by the kinetic energy of air passing through the duct of the model (M4) under reasonable assumptions.

| Vehicle speed (km/h) | Averaged air velocity in the duct (m/s) | Turbine efficiency (%) | Generator efficiency (%) | Usable power (watt) | Contribution to total power in % |
|----------------------|---------------------------------------|------------------------|-------------------------|---------------------|----------------------------------|
| 60                   | 13.60                                 | 85%                    | 88%                     | 192                 | 7.32%                            |
| 80                   | 18.18                                 | 85%                    | 88%                     | 458                 | 7.39%                            |
| 100                  | 22.79                                 | 85%                    | 88%                     | 903                 | 7.47%                            |
| 120                  | 27.39                                 | 85%                    | 88%                     | 1,567               | 7.51%                            |

Power turbine and generator efficiency (assumed)

As shown in Table 3, the harvested electric energy (calculated) proportionally increases with the model speed. As the speed reaches 120km/h, the power harvested is up to 1.567kW. The ratio of usable harvested power to the total power consumed to overcome the aerodynamic resistance of the base model is over 7% as indicated in the last column of table 3. For the estimation of total driving range extension with this air duct installed at the front side of the model vehicle, both the aerodynamic drag reduction and the electric energy harvested should be considered. Experimentally supported detail analysis of the aerodynamic energy harvesting is a continuation of this work as a future plan.

6. Conclusion and future work
In this numerical study, a new approach to further reduce or use aerodynamic resistance is introduced. It is found that the air flow duct installed on the model vehicle show the positive effects and significant decrease on the total drag force. The pressure drag shows a decreasing trend as duct cross-sectional area increases although the skin friction drag increases with the duct size. It was also confirmed that skin friction drag increase due to ducting is insignificant as pressure drag force accounts over 90% of the total drag. Overall a drag reduction of over 15% is observed in a model with a minimum duct size among the test models in this study.

Velocity profile through the duct centre indicated the ducted flow characteristics particularly the fluid acceleration and deceleration before, at the duct entry, exit, and after exit respectively. The level of kinetic energy preserved through ducting is also clearly indicated on the velocity profile Figure 10. The electric energy harvested from this kinetic energy of the flow in the duct would definitely contribute to driving range extension of electric vehicle if effective and efficient system is designed to capture the energy. Effect of ducting on the stability of the model, feasibility of energy harvesting through ducting and front & rear flow control for better duct flow acceleration is part of the future work.
References

[1] Mehrdad Ehsani A E and Gao Y 2010 Modern electric, hybrid electric and fuel cell vehicles, 2nd Ed. CRC Press, 2010.

[2] Kim C 2011 A streamlined design of a high-speed coach for fuel savings and reduction of carbon dioxide, Int. J. Automot. Eng., 2, 101–7.

[3] Nalanagula S and Varadharajan G T 2016 Aerodynamics drag reductions methodology for the commercial vehicles using computational fluid dynamics, SAE Tech. Pap. Ser., 1, 2016.

[4] McCallen R C, Salari K and Ortega J M 2004 DOE’s effort to reduce truck aerodynamic drag - joint experiments and computations lead to smart design,” in 34th AIAA Fluid Dynamics Conference and Exhibit, June 28 - July 1, Portland, USA.

[5] McCallen R, Couch, et al 2010 Progress in reducing aerodynamic drag for higher efficiency of heavy duty trucks (Class 7-8),” SAE Tech. Pap. Ser., 1(724), 2010.

[6] Altinisik A, Yemenici O and Umur H 2015 Aerodynamic analysis of a passenger car at yaw angle and two-vehicle platoon, J. Fluids Eng., 137(12) 121107.

[7] Hucho W and Sovran G 1993 Aerodynamics of road vehicles, Annu. Rev. Fluid Mech. 25(1) 485 - 537.

[8] Johnson T V 2015 Review of vehicular emissions trends, SAE Int. J. Engines, 8(3), 1152 - 67.

[9] Wood R M 2004 Impact of advanced aerodynamic technology on transportation energy consumption, SAE World Congr., 113(21) 854 - 74.

[10] Ahmed S R, Ramm G and Faltin G 1984 Some Salient Features of the Time -Averaged Ground, SAE Trans. 93 473–503.

[11] Kalyan D K and Paul A R 2013 Computational study of flow around a simplified 2D Ahmed Body,” Int. J. Eng. Sci. Innov. Technol., 2(3) 148–153.

[12] www.hyundai.com, Posted on Oct. 21, 2017.

[13] Chen Y-S and Kim S-W 1987 Computation of turbulent flows using an extended K-Epsilon turbulence closure interim report, NASA-CR-179204, Alabama, USA.

[14] Tennekes H 1972 A First course in turbulence. Massachusetts: The MIT Press, 1972.

[15] Hinze J O 1975 Turbulence. McGraw-Hill, New York, 2nd Ed., 1975.

[16] Banks J 2007 Turbulence Modeling in PHOENICS [Online]. Available at: http://www.cham.co.uk/phoenix/d_polis/d_enc/turmod/enc_t342.htm.