International Conference on Mechanical, Industrial and Energy Engineering 2018
23-24 December, 2018, Khulna, BANGLADESH

ICMIEE18-125

CFD Study on Aerodynamic Effects of NACA 2412 Airfoil as Rear Wing on a Sports Car

Shamudra Dey[1,], Ranabir Saha[2]*
[1] Department of Mechanical Engineering, Shahjalal University of Science & Technology, Sylhet-3114, BANGLADESH
[2] Department of Mechanical Engineering, Bangladesh University of Engineering & Technology, Dhaka-1000,BANGLADESH

ABSTRACT

In the present research, aerodynamic effects of NACA 2412 airfoil as rear wing on a sports car has been investigated using Computational Fluid Dynamics (CFD) approach. The sports car has been modeled in the commercial software SOLIDWORKS 2016. Two different types of simulations were run: one for the flow around a simplified high speed sports car model with a rear wing, which is actually a NACA 2412 airfoil and the other for the flow without using a rear wing. The analysis has been carried out in ANSYS 15.0 FLUENT using k-epsilon model and for the velocity condition of 60 m/s. The effect of rear wing has been visualized from the aerodynamics perspective. Graphs of drag and lift coefficients and figures of velocity streamline, pressure distribution, Turbulence Kinetic Energy for both models have been discussed and compared. The details of the aerodynamic study has been presented in the paper.

Keywords: Computational Fluid Dynamics, Rear wing, Sports car, Lift, Velocity streamline

1. Introduction

After many years of research in automobile engineering, the ideal water drop body shape, very less sharp corner, smooth upper surface finish and other methods have been adopted in designing modern vehicle bodies for better aerodynamics [1]. When a sports car is driven at a very high speed especially on the highway and race circuits, it has propensity to lift over. Such incidents can take place, because as the higher pressure air is in front of the wind shield; it accelerates, causing a sudden pressure drop. This lower pressure creates lift on the car's roof as the air passes over it. In the worst case, once the air makes its way to back window, the notch created by the window dropping down to the trunk leaves a lower pressure space that the air fails to fill properly [2]. The separated flow is resulted in lower pressure, which creates lift which then acts upon the trunk surface area. An added rear wing can diffuse the airflow passing a vehicle, which minimizes the turbulence, adds downward pressure to the back end and reduces lift acted on the rear trunk to allow a vehicle to corner faster and be more stable at high speeds [1, 3].
Several literatures have been reviewed in this research work. Most of the papers available in the literature reflected upon the subject of flow analysis over passenger cars and drag reduction. As drag reduction has close relationship to fuel consumption, most of the authors have focused on drag reduction rather than vehicle safety and comfortable. Computational analysis to reduce the drag is performed by Barbut et al. [4], Rouméas et al. [5] on road vehicle and by Guilmineau [6] on the simplified car body (Ahmed body). Islam and Mamun [7] performed numerical and experimental study to measure the aerodynamic drag, but their work was concentrated on sedan car only [8]. This research paper focuses on using NACA 2412 airfoil as rear wing on a sports car to achieve significant reduction in lift force to ensure vehicle safety from aerodynamics perspective.

2. Problem formulation & CAD model description

Aerodynamic forces and moments from the air is experienced by the car when it moves through the air. The force that works in the direction opposite to the vehicle moving, is called drag, and the force perpendicular to the drag and normal to the ground is called lift. The drag and lift forces can be expressed in a non-dimensional form - the drag and lift coefficients, C_D and C_L, are defined respectively as:

$$C_D = \frac{Drag\ force}{(\rho V^2 A)/2} \tag{1}$$

$$C_L = \frac{Lift\ force}{(\rho V^2 A)/2} \tag{2}$$

An image has been provided (Fig.1) below to show the forces acting on a running car:

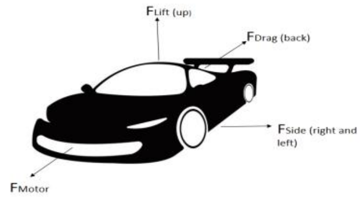

Fig. 1 Forces acting on a running sports car

The solid modelling of the present work has been done in the commercial software SOLIDWORKS 2016. The NACA 2412 airfoil with -3° angle is designed and

* Corresponding author. Tel.: +88-01879800785
E-mail addresses: shamudra.mee15@gmail.com

installed on the rear end of the sports car as rear wing. The designed car is basically a prototype which is scaled down to 1:24 (prototype to model) due to our computational limitation and desired accurate result. The solid modelling (Fig.2) of both cars (with or without rear wing) has been provided below:

Fig.2 Solid Model of sports car for both conditions

The maximum length and width of the prototype car is 140.41 mm and 87.18 mm and height (from the chassis to rooftop) is 39.13 mm. It should be noted that all the dimensions are normal & from point to point. The solid models have been imported to ANSYS Design Modeler, where a virtual wind tunnel is created around the sports car. This virtual wind tunnel is basically our computational domain. An image (Fig.3) of computational domain (sports car with rear wing) has been provided below:

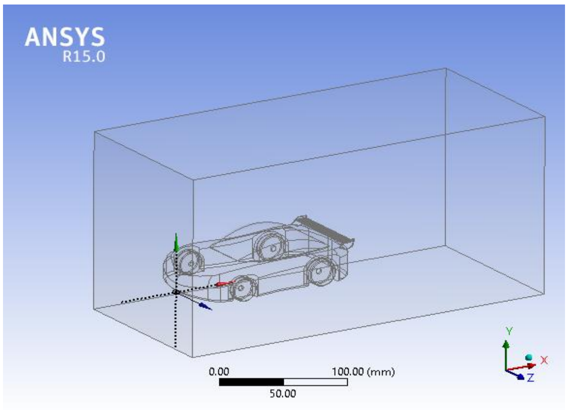

Fig.3 Computational domain

3. Mesh generation in ANSYS

The triangular shape surface mesh with lower skew ness and higher orthogonal quality has been used due to its proximity to changing curves and bends. These elements easily adjust themselves to the complex bodies used in automobile and aerospace applications [2]. With technical changes in the ANSYS ICEM CFD Meshing, the generated meshing for the sports car with rear wing can be visualized in the image (Fig. 3) provided below. The image shows a partial view of the meshing for better understanding about mesh generation. The CAD geometry of sports car without rear wing has 5, 05, 870 nodes and 27, 27, 068 elements, while the CAD geometry of the same sports car with the rear wing has 5, 59, 227 nodes and 30, 46, 978 elements.

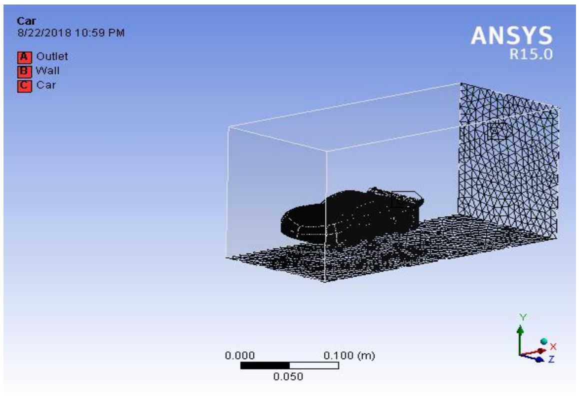

Fig.4 Mesh generation over the designed sports car with rear wing

4. Methodology of the CFD study

Aerodynamic study of air flow over a body can be executed using CFD approach or analytical method. On one hand, analytical method of solving air flow over an object can be done only for simple flows over simple geometries just like laminar flow over a flat plate. If the flow of air gets complicated as in flows over a bluff body, the flow becomes turbulent and it is impossible to solve Navier- Stokes and continuity equations analytically [2]. Favre-averaged Navier-Stokes equations are used here, where time-averaged effects of the flow turbulence are considered. Flow simulation is employed using transport equations for the turbulent kinetic energy and its dissipation rate, the well-known k-ε model [8].

4.1 Setup of Model:

Following steps have been taken for setting up the solving techniques in FLUENT solver for 3D, steady state, incompressible flow using serial processing technique:

(1) The meshed model is imported in solver and mesh is checked.

(2) The solver specifications such as pressure based, steady and absolute velocity formulation are set.

(3) Double equation based realizable k-epsilon model with non-equilibrium wall function is selected with default model constants. The k-epsilon model is a general purpose turbulence model, which is very simple to implement and converges faster. The model predicts the flows in many practical cases and is particularly good for external aerodynamics.

(4) Air is selected as the fluid and the Aluminum is selected as the solid and required properties (density, viscosity etc.) have been specified.

(5) In case of boundary conditions, the required inlet velocity flow condition of 60 m/s (216 km/h) is given as the input and the outlet is kept as pressure outlet. This particular inlet velocity condition has been selected to investigate the aerodynamic performance of the car at high speed and to compare the results with several literatures [9-14].

(6) Pressure velocity coupling based solution control is adopted for present study under the custom relaxation factors.

After these steps, the solution procedure has been initialized using hybrid initialization and the numerical solution has been done for 500 iterations with the reporting interval of 1.

5. Result & Discussion:

5.1 Grid sensitivity analysis

A grid sensitivity analysis has been carried out for the sports car model without rear wing to validate the mode. The workstation used for this CFD (Computational Fluid Dynamics) study is a Lenovo laptop with core i5 fifth generation processor and 8 GB ram. In this grid sensitivity analysis, the main priority has been given to the values of Coefficient of drag (C_D), Coefficient of Lift (C_L) and optimal computation duration. By analyzing the result from the table of grid sensitivity analysis and considering the computational limitation, the medium type mesh with 5, 05, 870 nodes and 2, 27, 068 elements have been preferred for the present work. This medium type mesh has been used for all four CFD simulations in this research. The duration of numerical solution for this type of mesh is 540 minutes approximately.

Table 1 Grid sensitivity analysis

Mesh type	Nodes	Elements	C_D	C_L
Coarse	396652	2135381	0.0012906	0.0006944
Medium	505870	2727068	**0.0013274**	**0.0007822**
Fine	529297	2851543	0.0013394	0.0007863

After the grid sensitivity analysis is being carried out, CFD Analysis is done for the preset inlet parameters and results are obtained through it. The results are achieved for two cases: one for the flow over the sports car body without rear wing and another for the flow over the same car with the rear wing made of NACA 2412 airfoil.

5.2 CFD analysis of sports car without rear wing

The below figures shows the giving of the required boundary conditions in different panels as mentioned earlier.

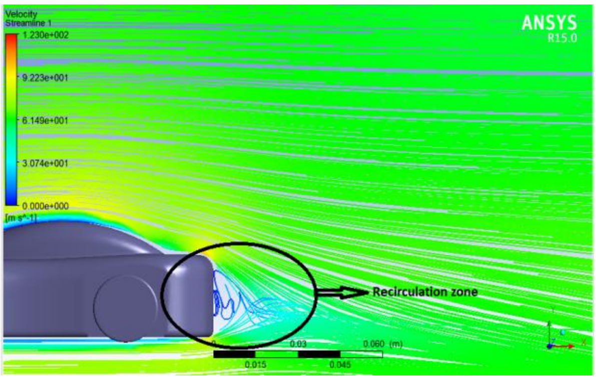

Fig.5 Velocity streamline over the sports car body without rear wing

Fig. 5 is about the velocity streamline over the car (without rear wing), which are a family of curves that are instantaneously tangent to the velocity vector of the flow. The distribution of the velocity streamline in front of the car is higher. On the rear side of the sports car, a recirculation zone can be visualized, which has occurred due to turbulent nature of the flow. Different streamlines at the same instant in a flow do not intersect, because a fluid particle cannot have two different velocities at the same point.

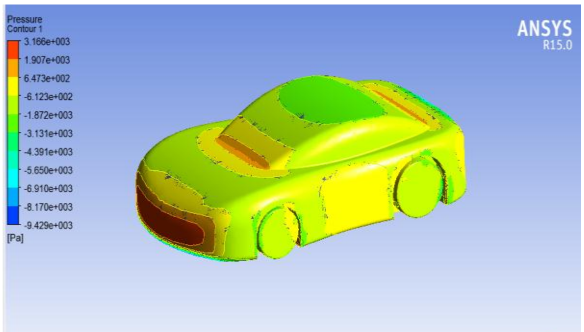

Fig.6 Pressure distribution on the body of the sports car without rear wing

Fig.6 is about the distribution of pressure on the car body. Pressure is higher in the front of the car. The pressure is also higher at the beginning of the front windshield of the car, but just after that region, pressure distribution becomes negative. This lower pressure creates lift on the car's roof as the air passes over it. In this figure, streamline has not been shown for better visualization.

Fig.7 is about the distribution of Turbulence Kinetic Energy distribution on the car body. The increment in the value of energy is observed in wheel region of the car.

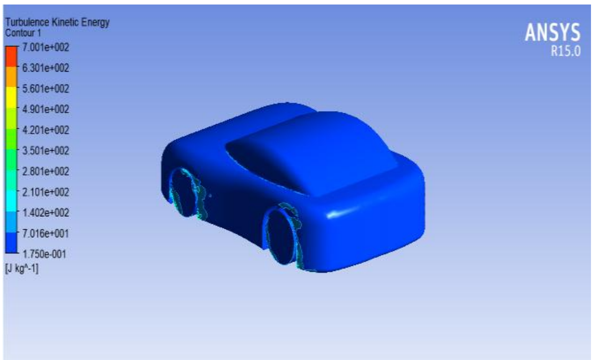

Fig.7 Turbulence Kinetic Energy distribution on the sports car without rear wing

5.2 CFD analysis of sports car with rear wing

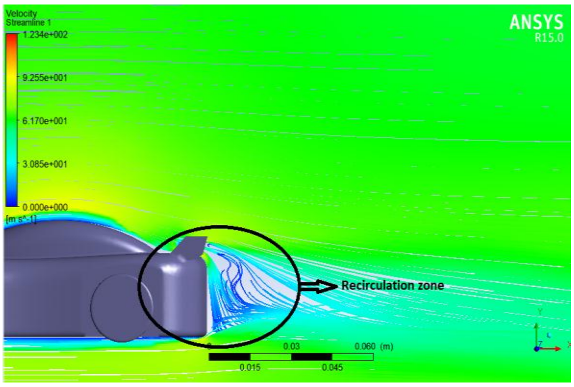

Fig.8 Velocity streamline over the sports car body with rear wing

Fig.8 is about the velocity streamline around the sports car when the rear wing is used. The velocity is higher in front of the car but, in the rear region, velocity is significantly lower as rear wing is installed. Besides, due to the effect of rear wing, the recirculation zone of air flow is almost gone.

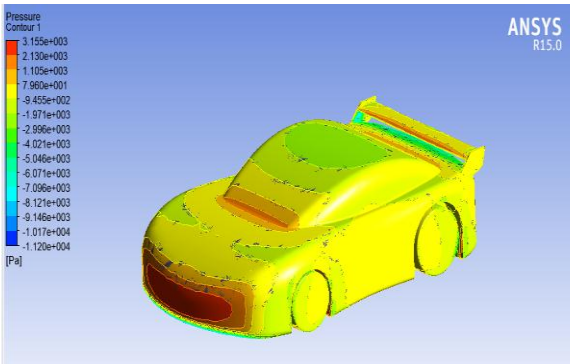

Fig.9 Pressure over the sports car body with rear wing

Fig.9 is about the pressure distribution due to the air flow over the sports car with rear wing. As the rear wing has been installed, a higher pressure region has been observed on the rear wing of the car. This higher pressure region creates down ward force which reduces lift and thus, the research objective is achieved.

The distribution of Turbulence Kinetic Energy (Fig.10) is almost same as the distribution on the sports car without rear wing. But in this case, increment of energy is visualized in few region on the rear end of the sports car.

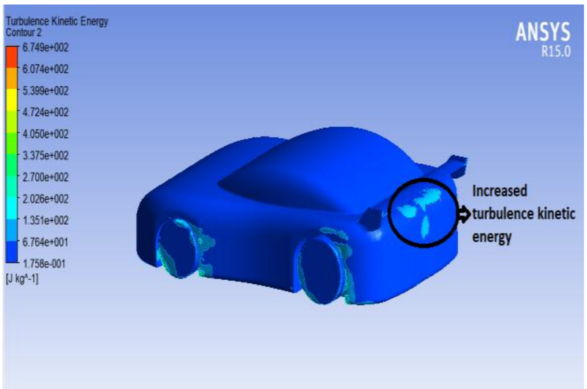

Fig.10 Turbulence Kinetic Energy distribution on the sports car with rear wing

5.4 Comparison of graphs of C_L & C_D

The graphs of C_D & C_L versus iterations have been plotted from this CFD study. A significant lift reduction has been observed in the graph of sports car with rear wing. In that numerical investigation, value of C_L is found to be negative, which ensures downward force. The obtained values are small because the solid geometry model is scaled down to a definite ratio, which has been discussed earlier. Graphs of C_L & C_D for both conditions (with or without rear wing) have been plotted below:

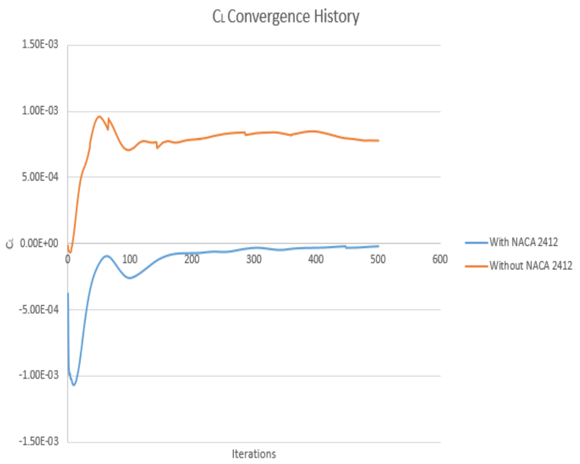

Fig.11 C_L vs. iterations for both conditions

From the graph of C_L (Fig.11), the value of C_L for the sports car solid model without rear wing and with rear wing has been found to be 0.00078223 and -0.00001688 respectively. The differences between the plotted lines in the graph can be visualized. As the lift has been reduced due to the installation of rear wing made of NACA 2412 airfoil, the downward force is achieved due to this negative C_L value.

In Fig.12, the graph of C_D has been plotted for the sports car solid model without and with rear wing. The value of C_D has been found to be 0.00132745 and 0.00152692 respectively. It is known that, drag force is created opposite of the direction that the vehicle is moving towards.

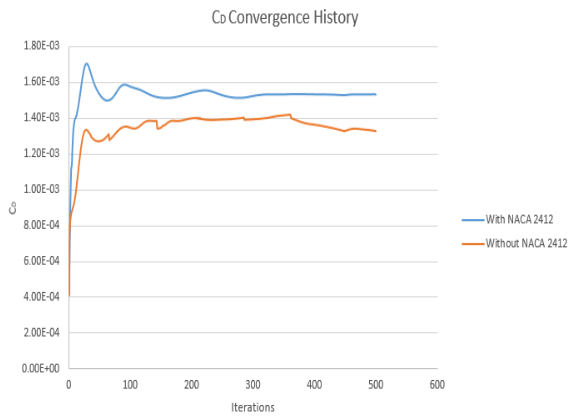

Fig.12 C_D vs. iterations for both conditions

The value of C_D has been increased when the NACA 2412 airfoil rear wing has been installed. From our hypothesis, the increment of drag force has occurred due -3° angle of the rear wing, which has created extra drag force. Our future research will try to identify the definite cause of such behavior and come out with a better design and solution through CFD analysis.

Table 2 below summarizes the result of plotted graphs of C_L & C_D, drag force and lift force for both conditions:

Table 2 Results obtained from CFD study

Sports car	C_L	Lift force (N)	C_D	Drag force (N)
Without rear wing	0.000782	1.70455	0.00133	2.92702
With rear wing	-1.688e-05	-0.0372	0.00153	3.36685

5.5 Validation of the CFD analysis

To validate the CFD analysis, the study has been compared to several published scholarly articles. The solid geometry model of this analysis is scaled down to 1:24. In case of solid model without NACA 2412 rear wing, the pressure coefficient and viscous coefficient are found to be 0.00114069 and 0.000186748 respectively. Therefore, 85.9% of the total drag is pressure drag and rest is friction drag. This result has been compared to the popular Ahmed Body aerodynamics experiment for the validation purpose. According to the research conducted by Ahmed et al. on 1984, for a basic bluff vehicle body, up to 85% of the total drag is pressure drag and rest is friction drag. The wind tunnel speed was kept 60m/s [9]. The Mathematical model and physics setup which are used in the present analysis have followed several literatures [8][10-14]. The inlet velocity condition and specifications of the solid geometry model in those studies were different from our present research. So, in case of comparison, the main focus has been given to the flow visualization and the effect of NACA 2412 rear wing on the sports car.

6. Conclusion & future research possibilities

The results of this CFD study have been compared to the similar studies that have been done previously [9-14]. Our main research objective has been achieved as lift is reduced by creating downward force in case of the sports car model with NACA 2412 airfoil made wing. From the CFD analysis carried on two different models of sports car (with or without rear wing), the following conclusions can be drawn.

(1) Vehicle stability is increased on high speed by the reduction of significant amount of lift force. It has been achieved by the installation of NACA 2412 airfoil as the

rear wing. Thus, safety is ensured highly and tendency to lift over at high speed has been minimized. This research also focuses to reduce accidental risks of high speed sports car. Leading sports car companies such as Bugatti, Porsche and Mercedes have been using different technologies for rear wing and trying to maximize the efficiency by eliminating the side effects at low speeds and increase the benefits on high speeds. For them, the key reason is safe driving. Also, the recirculation zone at the rear end is minimized, which ensures the cleanliness of the rear wind shield.

(2) Secondly, the feasibility of installing NACA 2412 airfoil as the rear wing of a sports car has been studied from aerodynamics perspective. Further studies can be done to find the efficient angle for the NACA 2412 airfoil made rear wing so that lift to drag ratio can be maximized, which has been a popular research topic for past few years.

NOMENCLATURE

ρ : Density of the fluid, kgm^{-3}
V : Velocity of the flow, m/s
A : Projected area, m^{2}
C_D : Coefficient of Drag, no units
C_L : Coefficient of Lift, no units

REFERENCES

[1] Xu-xia Hu, Eric T.T. Wong, A Numerical Study on Rear-spoiler of Passenger Vehicle, *World Academy of Science, Engineering and Technology, International Journal of Mechanical and Mechatronics Engineering*, Vol:5, No:9, 2011.

[2] G.Ganesh, V.Vasudevan, Analysis of Effects of Rear Spoiler in Automobile Using Ansys, *International Journal of Scientific & Engineering Research*, Volume 6, Issue 6, June-2015 762, ISSN 2229-5518.

[3] Ridhwan Bin Che Zakem, Aerodynamics of Aftermarket Rear Spoiler, Malaysia Pahang University (2008).

[4] Dan Barbut, Eugen Mihai Negrus, CFD analysis for road vehicles - case study, *Incas Bulletin*, 3(2011) 15-22.

[5] M. Rouméas, P. Gilliéron, A. Kourta, Drag Reduction by Flow Separation Control on a Car after Body, *International Journal for Numerical Methods in Fluids*, 60(2009) 1222–1240.

[6] Emmanuel Guilmineau, Computational Study of Flow around a Simplified Car Body, *Journal of Wind Engineering and Industrial Aerodynamics*, 96(2008) 1207–1217.

[7] Md. Munir Islam, M.Mamun, Computational Drag Analysis over a Car Body, *International Conference on Marine Technology*, Dhaka, Bangladesh, 2010.

[8] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, Numerical Study on Aerodynamic Drag Reduction of Racing Cars, *10th International Conference on Mechanical Engineering, ICME 2013*, Dhaka, Bangladesh.

[9] Ahmed et al., Some Salient Features of Time-Averaged Ground Vehicle Wake, *SAE Technical Paper Series*, *International Congress & Exposition*, Detroit, Michigan.

[10] Cakir, Mustafa, CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle (2012), *Mechanical Engineering Masters Theses. Paper 1*, Santa Clara University.

[11] Bhamre et al., Experimental and numerical aerodynamics investigation of car, *International Journal of Engineering Development and Research*, Volume 4, Issue 2, ISSN: 2321-9939.

[12] A. Kourta, P. Gilliéron, Impact of the Automotive Aerodynamic Control on the Economic Issues, *Journal of Applied Fluid Mechanics* , Vol. 2, No.2, pp. 69-75, 2009, ISSN 1735-3645.

[13] A R S Azmi et al., Study on airflow characteristics of rear wing of F1 car, *IOP Conf. Ser.: Mater. Sci. Eng.*, 243 012030.

[14] Philipp Epple et al., Aerodynamic Devices For Formula Student Race Cars, *Proceedings of the ASME 2014 International Mechanical Engineering Congress and Exposition*, Montreal, Quebec, Canada.