

# A Numerical Study On Rear-spoiler Of Passenger Vehicle

Xu-xia Hu, Eric T.T. Wong

**Abstract**—The simulation of external aerodynamics is one of the most challenging and important automotive CFD applications. With the rapid developments of digital computers, CFD is used as a practical tool in modern fluid dynamics research. It integrates fluid mechanics disciplines, mathematics and computer science. In this study, two different types of simulations were made, one for the flow around a simplified high speed passenger car with a rear-spoiler and the other for the flow without a rear-spoiler. The standard k-ε model is selected to numerically simulate the external flow field of the simplified Camry model with or without a rear-spoiler. Through an analysis of the simulation results, a new rear spoiler is designed and it shows a mild reduction of the vehicle aerodynamics drag. This leads to less vehicle fuel consumption on the road.

**Keywords**— CFD, vehicle rear-spoiler design.

## I. INTRODUCTION

AFTER years of development in road vehicle aerodynamics, the ideal teardrop body shape, smooth underbody, less sharp corner and other methods have been applied in designing today's vehicle bodies. Take the vehicle spoiler as an example. The spoiler is used as a tool to minimize unfavorable air movement around the vehicle and can be divided into the front spoilers and the rear spoilers. A front spoiler, connected with the bumper, is mainly used to direct air flow away from the tires to the underbody. A rear spoiler is commonly installed upon the trunk lid of a passenger vehicle. The added spoiler can diffuse the airflow passing a vehicle, which minimizes the turbulence at the rear of the vehicle, adds more downward pressure to the back end and reduces lift acted on the rear trunk [1]. The rear spoiler is no longer just a decoration and they do have measurable effect on aerodynamic drag reduction and vehicle stability. This paper focuses on optimizing the selected rear spoiler parameters of a selected vehicle model to reduce aerodynamic drag. Sedan cars are the most common driven vehicles in modern cities; therefore, a simplified sedan model was used for the present simulation and experimental study investigation and an attempt was made to re-design a new rear spoiler.

## II. SIMPLIFIED PASSENGER VEHICLE

The car body experiences aerodynamic forces and moments

X. X. Hu is a postgraduate student of the Hong Kong Polytechnic University, Hung Hom, Kowloon, Hong Kong (e-mail: nicolehkpolyu@hotmail.com)

Eric T.T. Wong is with the Hong Kong Polytechnic University, Hung Hom, Kowloon, Hong Kong (email: mmttwong@polyu.edu.hk)

from the air when it moves through the air. The force on the vehicle in the direction opposite to its moving direction is called drag, and the force perpendicular to the drag and normal to the ground is called lift. The higher the drag force is, the more the horsepower is required.

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:

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

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

Where  $\rho$  is the air density,  $V$  is the vehicle velocity,  $A$  is the frontal projected area of the vehicle [2].

Use of Computational Fluid Dynamics (CFD) software greatly reduces time-to-market by reducing the need for costly physical testing and prototyping. Hence in the present study ANSYS Fluent [3] was used to build a virtual prototype and the performance of proposed and existing designs was simulated.

Fig. 1 shows the simplified car model. In this case, the model is simplified by removing wheels and rear-view mirrors and the bottom portion of the car was assumed to be a flat surface.

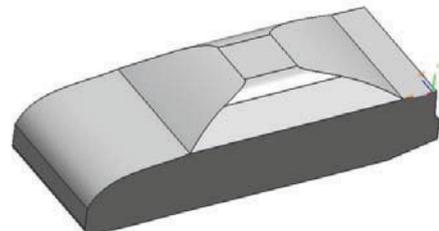


Fig. 1 Simplified 3D car model

Low-skewness triangular element was first created over the car surface. Curvature of every surface was fitted with smaller triangles when necessary. Smooth transitions are guaranteed between areas meshed with triangles of different sizes to avoid excessive numerical dissipation. Great care was taken for surface meshing as volume mesh quality is directly correlated with its quality.

Layers of prisms were extruded from the surface mesh to capture flow features in the boundary layer. The extrusion was performed through Gambit 2.0, a general-purpose preprocessor for CFD Analysis, which allows engineers easier access into the CFD world.

Fig. 2 shows four layers of prism elements created around the car surface to capture the boundary layer.

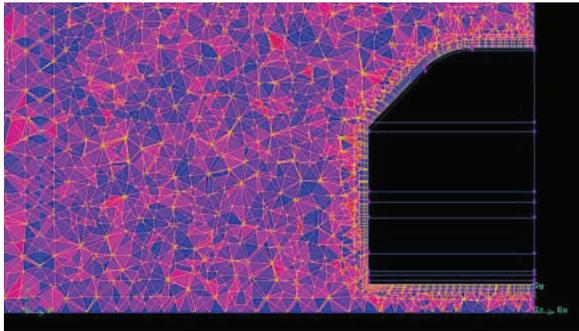


Fig. 2 Layer of prism element in Gambit

Once the prism layers had been created around the vehicle body, the rest of the fluid domain, representing the numerical wind-tunnel, was built using fully automated procedure. This procedure has been robustly developed using scripting and batch capabilities of Gambit. The dimensions were chosen to ensure a low blockage ratio. The full size of the 3D grid was about 1.37 million grid cells. Automatic mesh refinement around the car and in the wake was also implemented to guarantee sufficient mesh quality for satisfactory results and numerical stability. Tetrahedral mesh was carried out for the remaining entire domain and the boundary conditions were assigned. The meshed domain of simplified car surface model was then imported to Fluent.

Three-dimension incompressible viscid Navier-Stokes equation was used and the SIMPLE algorithm was selected for pressure/velocity coupling. Standard  $k-\epsilon$  turbulence model was selected for further simulation. The discretization scheme for convection terms of the momentum and turbulence equations adopted was second-order upwind. The standard method was used for the pressure equation. The solution was initialized with an X velocity of 40m/s. Airflow was assumed to be incompressible and heat transfer was considered insignificant in this study.

Flow velocities and properties were monitored during the simulation, until there was no appreciable change upon further iteration. Additionally, the total momentum residuals were allowed to fall by a minimum of three orders of magnitude [5]. Then, the calculations were terminated.

Obviously, two positive pressure areas can be found: one is at the front of the vehicle body, and the other is between the hood and the windshield. At the same time, high negative pressure areas can be found at the front and rear end of the roof and a small area at the front end of the hood; as the pressure coefficient increases to more than 0.415 the flow accelerates over the bonnet and due to this acceleration the pressure drops and becomes negative.

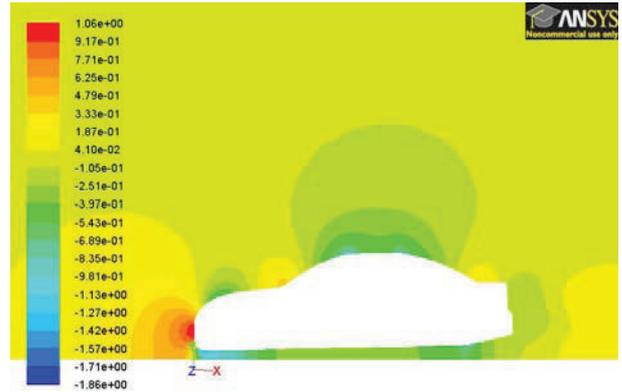


Fig. 3 Distribution of Pressure on the symmetry plane

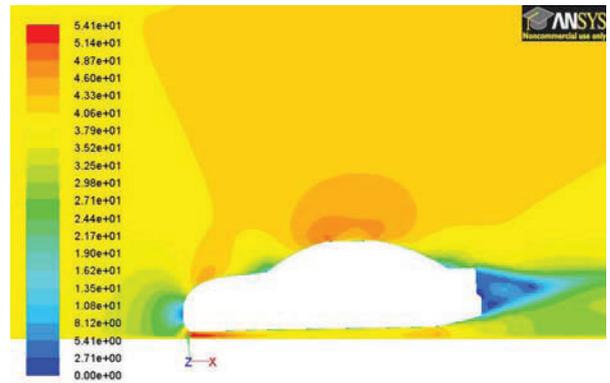


Fig. 4 Distribution of Velocity on the symmetry plane

It is believed that positive pressure at the bottom and negative pressure at top will make the rear portion of the car to lift up from the ground and reduce the pressure between the wheels and ground, which causes loss of steering on the front axle and loss of traction on the rear axle. Pitching moment is usually negative i.e. the nose down, and this makes the rear axle lift off the ground, which further reduces the available traction. The pressure and velocity contours are shown in Fig. 3 and Fig. 4. The predicted drag coefficient is 0.574 at 40m/s.

Fig. 5 shows the sections of velocity distribution along the axle direction. At the Z=0 section (symmetry plane), flow passing the front body goes backward along the car body, and separates at the boundary area at front of the windshield, and then separates again at the middle point of rear window. As can be seen that, two types of flow are formed (upward flow and downward flow) based on bumper area.

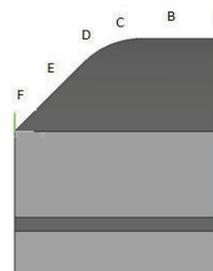


Fig. 5 Position of 6 sections along the axle direction

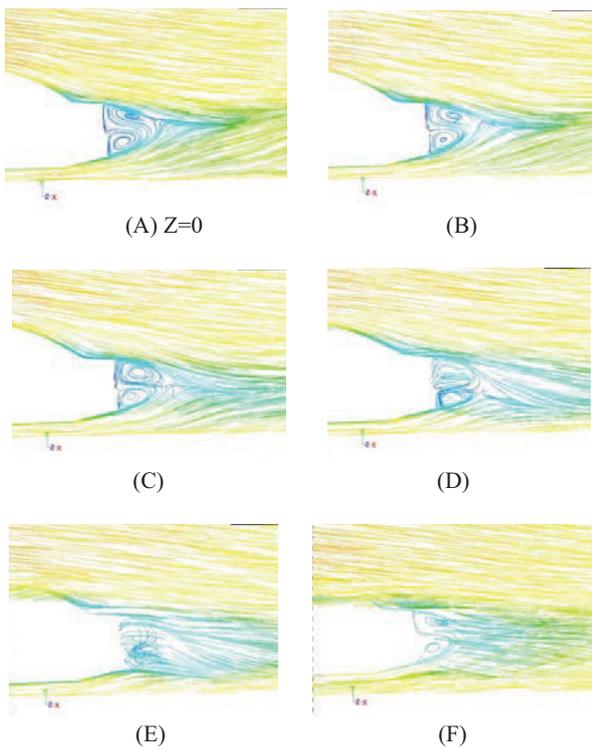


Fig. 6 Path line of velocity distribution around a car body

It is easy to find that the upward flow along the upper surface of the trunk and the separated flow meet at the end of the tail. The flow direction is changed immediately and contributes to a large re-circulation flow which existed in the upper part of the rear flow field. Since the flow towards the lower part of the vehicle and the relatively fast flow from the underbody meet together at the lower region. That is why an anti-clockwise vortex can be found in the lower part near the bumper.

As mentioned before, two re-circulation flows formed in the rear flow field. Fig. 6 (A) shows that the upper re-circulation region is more expanded than other sections and the flow moves near the vehicle tail surface. Because of the flow component of the upper part which goes downwards to the under body surface is larger, an anti-clockwise vortex formed at the lower part of the vehicle tail region. This is similar to section A ( $Z=0$ ), though the effect of upwash from the bottom was much larger. The two vortex regions became weak along Z direction, for the upper flow goes directly to the ground and the upper one even extinct. Another factor cannot be ignored is that the fast side flow passing from the side curvature body controls the upper and lower flows.

### III. NUMERICAL SIMULATIONS

The design of the new rear-spoiler was aimed to reduce both the drag force and the lift force. The aerodynamic drag force consists of two components, one is the friction drag (skin drag) and the other is pressure drag (form drag). The friction drag is the result of shear stresses on the body, and the pressure drag is

due to the pressure differences between the front region and the rear region of the vehicle.

At high Reynolds number, the major contribution of the total drag on a streamlined body such as an airfoil is the friction drag and is very low ( $C_D \approx 0.01$  at  $Re \approx 10^6$ ) [5]. Since there is no or almost no flow separation at the front of the streamline body, the pressure drag takes low percentage of the total drag. However, the drag on a body with a bluff back such as a half cylinder is contributed mainly by the pressure drag and is very high ( $C_D \approx 1.2$  at  $Re \approx 10^6$ ) [5]. Because of the boundary layer flow separates from the rear top of the body, the pressure magnitude at the rear of the body is much smaller than that at the front.

The spoiler can affect the flow pattern, and it induces a negative lift force on the car body by modifying the flow over the upper face. As a result, the grip of the cars on the road is increased and so is the driving stability.

The aerodynamic influence of the spoiler was carried out by numerical simulations. In this study, parametric study was carried out on the clearance A between the rear window and the leading edge of the rear- spoiler between the upper surface , and the clearance B of spoiler bottom to the trunk. The distance was varied by 80mm from position 1 to position3. The chord length was used to define the clearance B. Two spoiler types were considered: airfoil and plate (see Fig. 7).

TABLE I  
REAR SPOILER GEOMETRIC CHARACTERISTICS

Case NO.	Spoiler Type	Angle of Attachment (°)	Configuration	
			Clearance A	Clearance B
1	Airfoil	5	Position 1	h= 0.4c
2				h= 0.8c
3				h= 0.4c
4			Position 2	h= 0.8c
5				h= 0.4c
6				h= 0.8c
7	Plate	10	Position 1	h= 0.4c
8				h= 0.8c
9				h= 0.4c
10			Position 2	h= 0.8c
11				h= 0.4c
12				h= 0.8c
13	Plate	5	Position 1	h= 0.4c
14				h= 0.8c
15				h= 0.4c
16			Position 2	h= 0.8c
17				h= 0.4c
18				h= 0.8c
19	Plate	10	Position 1	h= 0.4c
20				h= 0.8c
21				h= 0.4c
22			Position 2	h= 0.8c
23				h= 0.4c
24				h= 0.8c

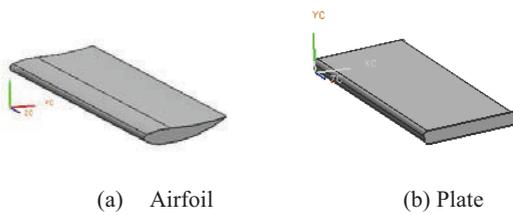


Fig. 7 Two types of rear spoilers considered

The inclinations of the rear surface of rear-spoiler were changed accordingly. According to the variation of parameters,

TABLE II  
DRAG COEFFICIENT REDUCTION RATE OF VEHICLE WITH REAR-SPOILER

Case NO.	Reduction Rate (%)	Case NO.	Reduction Rate (%)
1	-9.99	13	-4.23
2	-6.73	14	-6.73
3	-3.64	15	2.38
4	-11.73	16	0.14
5	-6.65	17	-7.47
6	-10.86	18	-2.80
7	-6.65	19	-5.17
8	-9.23	20	-6.04
9	-11.54	21	-8.90
10	-6.56	22	-4.62
11	-9.65	23	-8.10
12	-10.26	24	-9.83

24 cases were generated and they are shown in Table .

Table shows the drag and lift force of all the 24 cases based on the attachment position and angle of attachment. It can be seen that in most cases, cars with a spoiler actually have a higher drag force than cars without it. However, a maximum reduction 2.38% is achieved in case 15 and case 16 has an available reduction rate of 0.14%. When focusing on the lift reduction, both of the cases are failed. As a result, the plate shape rear- spoiler in Position 2 with an attachment angle of 5 degrees has a good performance in drag force reduction. According to the performance of the airfoil cases, it is obviously that the total drag force is larger than vehicle without rear-spoiler. The smallest drag force occurs in case 9, also in position 2 with a  $0.4c$  ( $c$ : chord length of the spoiler clearance). Consequently, position 2 with height ( $0.4c$ ) is selected as the new rear-spoiler attachment position.

#### IV. DEVELOPMENT OF NEW REAR-SPOILER

To keep a car stable at high speeds and effectively prevent it from flying off the ground the rear spoiler are adopted. Next the simplified car is simulated with the new rear spoiler.

The development of a new rear spoiler should consider the following conditions [2]:

1. The new spoiler should act as a diffuser when it is attached to the upper surface of the trunk so that vehicle with rear-spoiler should have higher pressure in the back compared with the vehicle without the rear-spoiler.

2. Normally an additional attachment to a vehicle causes an

additional drag and lift due to the presence of it attached to a moving body. The new rear-spoiler should be designed in such a way that the additional drag on the spoiler should be much smaller than the total drag on the car. Furthermore, it would be much better if it is possible to design a rear spoiler in such a way that the force on the spoiler along the direction of the vehicle motion acts in the same direction as the vehicle motion.

3. The value of the additional lift on the new spoiler should be very small if it is positive. Furthermore, it would be much better if it is possible to design a rear spoiler in such a way that the lift on the spoiler is negative.

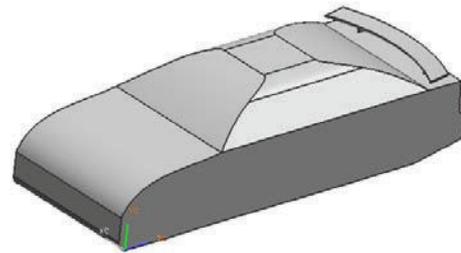


Fig. 8 3D vehicle model with rear spoiler

Fig. 8 shows a new spoiler attached to the rear end of the car surface. Results from the rear spoiler development cases suggested that the most ideal position for spoiler attachment is in position 2 with a clearance  $0.4c$ . As a result, the new spoiler is attached to the position mentioned above.

#### V. COMPARISON OF REAR-SPOILER DESIGNS

In this section, the computational results for the following cases are presented and discussed:

Case 1: results for the simplified car model without rear-spoiler

Case 2: results for the simplified car model with the new rear-spoiler

All the results are obtained with the same meshing resolution, the standard  $k-\epsilon$  turbulence model, and also the same boundary conditions. Drag and lift coefficients for the two cases are presented in Table III. The reduction rate of the drag and lift coefficients between the two cases did not depend appreciably on mesh resolution, turbulence model, and domain size.

Firstly, the predicted drag coefficient with the help of a

TABLE III  
DRAG COEFFICIENT REDUCTION RATE OF VEHICLE WITH REAR-SPOILER

Model	$C_D$	Reduction Rate	$C_L$	Reduction Rate
Case 1	0.574		-0.820	
Case 2	0.564	1.7%	-0.852	4%

spoiler is reduced to 0. 564 and shows an increase in downwards force for car with rear spoiler.

The results for Case 2 (which has the new spoiler attached in the rear end) were compared with the results for Case 1(without

rear- spoiler). Contours of velocity and pressure distribution of the flow field around the car body in the symmetry plane are presented, respectively, Fig. 9 (a) and Fig. 9 (b) for vehicle with spoiler. Fig. 9 illustrates that the boundary-layer flow separates below the middle of the rear window, and the flow slows down from the separation point. A region with relatively low speed forms just above the boundary area between the rear window and the trunk.

Fig. 9 (b) shows the pressure contour over the passenger car, the flow accelerates over the roof of the vehicle and pressure drops to a negative value at the rear of the vehicle and low negative pressure area occurs below the tail end.

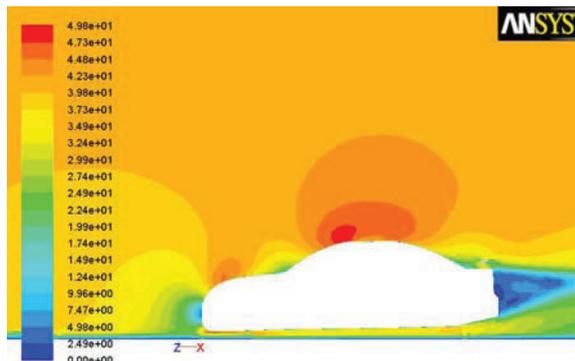


Fig (a) Velocity field around the vehicle rear body in the center plane for Case 2

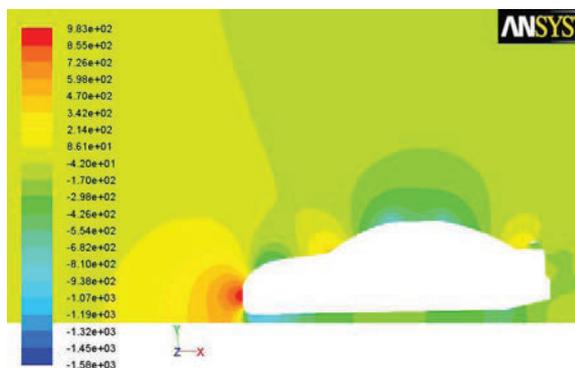


Fig (b): Pressure field around the vehicle body in the center plane for Case 2

Fig.9 Pressure and Velocity around the vehicle with spoiler

Velocity contour for a high speed vehicle with spoiler at the symmetry plane is shown in Fig. 9 (a) and the car with rear spoiler has large and double air swirl at the rear end. This positive pressure at the top and negative pressure at bottom will make the rear portion of the car to create a downward force.

The road vehicle fuel consumption is mainly used to provide the tractive power to overcome the resistive forces opposing its motion, with a small portion (estimated to be 8%) to power vehicle accessories. The total tractive power  $P$  needed to drive a vehicle is equal to:

$$P = FT \times V \tag{3}$$

Where  $FT$  is the total traction force and  $V$  is the vehicle velocity. When the vehicle moving on the road, the  $FT$  can be calculated as follows

$$FT = FD + FR + FG + F\alpha \tag{4}$$

$FD$  —Aerodynamic drag;

$FR$  — Rolling resistance;

$FG$  —Climbing resistance (relate to inclination angel of road);

$F\alpha$  — Acceleration resistance.

Considering the fact that as much as 65 % of the power required for ground vehicles to travel on a highway at 70 miles per hour is consumed to overcome the aerodynamic drag, the total power can be simplified by

$$Power = FD \cdot V / 0.65 = \frac{C_D A \rho V^3}{1.3} \tag{5}$$

The relationship between power and fuel economy is

$$MPG = 1.3 / (bsfcv \cdot C_D A \rho V^2) \tag{6}$$

In (6),  $bsfc$  is the Brake Specific Fuel Consumption (BSFC) and is an index of fuel efficiency. It indicates the rate of fuel consumption divided by the power produced. Fuel economy is the distance travelled per unit volume of fuel used and unit is kilometers per liter (km/L) or miles per gallon (MPG). The higher the value, the more economical the vehicle is.

The original  $C_D$  of the simplified car model is 0.574 and the optimized model is 0.564. The MPG reduction is about 1.7%.

## VI. CONCLUSION

The aerodynamic lift, drag and flow characteristics of a high speed sedan car were numerical investigated. There are two additional contributions to the total pressure force on a vehicle without the spoiler. The first contribution is made by the pressure force on the spoiler, and the second contribution is due to the increase in pressure on the back surface of the vehicle due to the diffuse effect of the spoiler. A standard  $k - \epsilon$  model was selected as the turbulence model for numerical simulations. And then three-dimensional external flows around the car body with a rear-spoiler were investigated.

In the present study, three parameters were taken into consideration. The parameters include: the attachment position, the shape of the spoiler, and the variable of clearance A between the rear window and the leading edge of the rear-spoiler and the clearance B of spoiler bottom to the trunk. Through comparing the results of 24 cases, a rear spoiler of better performance has been developed. The high speed passenger car with new rear spoiler shows that the aerodynamic drag is reduced by 1.7% and also it increases negative lift.

The high speed sedan car with a spoiler shows a drag reduction of 1.7%, which would result in fuel saving. Also, it increases the negative force which generates the following advantages:

1. Increases tires capability to produce cornering force;
2. Stabilizes vehicles at high speed;

3. Improves braking performance;
4. Gives better traction and increases the fuel efficiency.

#### ACKNOWLEDGMENT

The authors would like to thank the Department of Mechanical Engineering, The Hong Kong Polytechnic University for the wind-tunnel laboratory and CFD facilities provided for the present study.

#### REFERENCES

- [1] Ridhwan Bin Che Zakem, Aerodynamics of Aftermarket Rear Spoiler, Malaysia Pahang University, 2008.
- [2] Inchul Kim, Huailei Chen and Roger C. Schulze, "A Rear Spoiler of a New Type that Reduces the Aerodynamic Forces on a Mini-Van", *SAE Technical Paper Series*, 2006-01-1631.
- [3] ANSYS Fluent 12.0 Theory Guide, retrieved on July 26, 2011 from <http://www.cadfamily.com/downinfo/285585.html>
- [4] Retrieved on July 26, 2011 from <http://geolab.larc.nasa.gov/APPS/YPlus/>
- [5] Jones, W. P. and Launder, B. E., The Prediction of Laminarization with a Two Equation Model of Turbulence, *International Journal of Heat and Mass Transfer*, 15, 301-314.1972.