ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 4, April 2018

Computational Analysis of Aerodynamics Effects of a Rear Wing/Spoiler of Formula –1Car.

[1] Piyush Chavda, [2] Darshan Ajuida.
[1] M.Tech student, Department of Mechanical Engineering, Marwadi Education Foundation, FOPG, Rajkot, Gujarat, India. [2] Assistant Professor, Department of Mechanical Engineering, Marwadi Education Foundation, FOPG, Rajkot, Gujarat, India,

***Abstract*— Formula vehicle becomes very known to almost anyone due to peoples having so much love and passion on Racing and Automobiles vehicles. Aerodynamics plays a important role in efficiency of the vehicle and engine performance. To maximize the performance of the vehicle, the aerodynamics forces acting on the automobile vehicle and how to utilize those forces for increasing the performances and stability. A wing or car spoiler is accessory that generally attached to the rear end of the automobiles vehicles like car, and normally mounted on top of a car's trunk or positioned under the front bumper. The low pressure zone at back end creates drag force on vehicle is overcome by using rear Spoiler. The different designs of rear spoiler used are based on the different type of the automobile vehicles used, therefore aerodynamic shape of the automobile bodies and the point of the rear spoiler is important in this analysis. In this study, we have selected formula-1 car spoiler for our analysis purpose. To perform analysis we use commercial software CREO for solid modeling of F1 car body. After that for analysis we use CFD tools. By this analysis we can find out lift and drag forces, pressure and velocity distributions. Possibly we may improve aerodynamics of F1 car body.**

***Index Terms*—CFD analysis, Co-efficient of drag and lift, Formula 1 rear wing, NACA 4412 Aerofoil, Turbulence Analysis/**

I. INTRODUCTION

Formula one is the one of event of motorsport currently referred to as the FIA Formula One World Championship. Increasing competition, changes in regulations, the ever increasing need to cut costs by reduced track testing and, maybe in the future, a freeze on design changes collectively contribute to the need of dependence on computer simulations for various aspects of performance enhancement. The aerodynamic design has two primary concerns. First one, the creation of downforce to help improve the cornering force and pushes the car's tires onto the track. Second one, to minimizing the drag that caused by turbulence and shape of car body or automobile vehicle and trying to slow the car down. Due to high strength-to-weight ratio, fiber-reinforced composites are often more profitable and efficient than conventional materials. The wing function with air flows at different speeds over the two different sides of the wing aerofoil by having to travel different distances over its contour, this creates a difference in pressure. This pressure helps to create the downward force to the car. This pressure can make the wing tries to move in the direction of the low pressure called negative lift in this case of aerodynamics. As flow of air over the wing, it's disturbed by the shape of aerofoil wing causing a drag force. Although this drag force generated due to shape of wing in air flow is generally less than lift or downforce. Francesco Mariani, Claudio Poggiani, Francesco Risi, Lorenzo Scappaticci [1] was studied external aerodynamics of Formula SAE car racing both experimentally and as well as numerically. They have found that use of wing reduce total drag forces and increases downward forces. S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam [2] was Numerically Studied Aerodynamic Drag Reduction of Racing Cars. Drag coefficient was found to be 0.3233. Whereas, for Rear under- body diffuser gives 9.5% drag coefficient reduction. Sneh Hetawala, Mandar Gophaneb, Ajay B.K.c, Yagnavalkya Mukkamalad, [3] researched on Aerodynamic Study of Formula SAE Car. Drag co-efficient is reduced from 0.85 to 0.70. Negative lift is increased from 0.2 to 0.25. Overall pressure around the driver head region is dropped from 340Pa to 80 Pa. C.V.Karthick, Bala Murugan, P.A.Nigal Ashik, P.Raju [4] researched numerically about optimization of a car spoiler. The change in Cd with respect to the speed of the racing automobile car is negligible. The downforce acting on the racing car with the rear spoiler increases notably lower as increase in the speed of the racing car. Additionally, we get the higher Cl and Cd for lower spoiler height. Jasminder Singh, Dr. Jaswinder Singh, Ampritpal Singh, Abhishek Rana, Ajay Dahiya. [5] has conducted numerical of study of NACA 4412 and Selig 1223 airfoils. Found that NACA 4412 gives better left force at high speed. Xu-xia Hu, Eric

T.T. Wong [6] were numerically researched for Rear-spoiler Of Passenger Vehicle, Drag reductions for case 1 - 0.574 and Drag reductions for case 2 – 0.564. Development of aerodynamic in F1 cars now a day, almost all the team uses the advantages of CFD by means some CFD software packages. The results are produced without manufacturing or construction of the required F1 car and its components prototype, which is a main advantage of CFD. Analyze flow around F1 car rear wing with particular wing profile and its drag and lift generation to improve its performance. Computational fluid dynamics (CFD) is widely used in the field of aerodynamics for design and analysis of aerospace and automobile vehicles. Different rear wing design gives different flow structure around body with respect to various parameters like speed, AOA, height, etc. In this study, we have used Solid modeling software of preparation of model and for meshing purpose ICEM CFD is used. Further, to perform analysis of rear wing of formula one car ANSYS FLUENT 17.2 was used.

II. OBEJCTIVES AND METHODOLOGY

A. Objectives

1. Design of aerodyAnamic rear wing of formula one's car NACA 4412.
2. Analyse the drag coefficient, Cd, lift coefficient, Cl, pressure and velocity variations at different speed for the designed aerofoil.
3. With 1 camber length height and AOA(°) -2,0,2,6,10

B. Methodology

The model of formula one car is prepared with commercial software using CREO. Then various aerofoils with different angle of attack will made with base model.

- Pressure based solver
- Time – steady flow
- K- ω model
- Solution methods – SIMPLE

– Gradient – least square call based
– Pressure - second order
– Momentum – second order upwind
– Turbulent kinetic energy - Second order upwind
– Turbulent Dissipation rate – Second order upwind

- Initialization- hybrid
- Initialization iteration - 200
- Solution iteration – 750.

SST k-ω turbulent model is selected for these analyses. K-ω includes two extra transport equations to represent turbulent properties. K-ω model is reported to perform better in transitional flows and in flows with adverse pressure gradients. The model is numerically very stable, especially the low- Re version, as it tends to produce converged solutions more rapidly than the k -ε models. K-ω model predicts well near wall So SST K- ω model is selected due to combines the k -ω and K- ε turbulence model such that the k-ω is used in the inner region of the boundary layer and switches to the K- ε in the free shear flow.

III. NUMERICAL MODEL AND SET-UP

A. Physical model.

As we mention earlier, the physical model of rear wing of formula one car is modeled with help of commercial software CREO and the profile of wing is selected as NACA 4412. Which was also generated with help of CREO modeling software as per regulations of 2018 FIA FORMULA 4 TECHNICAL REGULATIONS[7]. We generated different model on the basis of different AOA.

- Chord length (c) = 235 mm
- Thickness (t) = 40 mm

B. Mesh generation and Domain Set-up.

Domain is main part of analysis problem. We have selected domain area around rear wing to capture proper fluid flow phenomena. The size of domain is

- Inlet distance from leading Edge = 1 c.
- Outlet distance from trailing edge = 3 c.
- Domain = 1235mm x 360mm x 360mm.

The generation of mesh in domain area is done by using ICEM CFD. The generated elements are tetrahedral in nature.

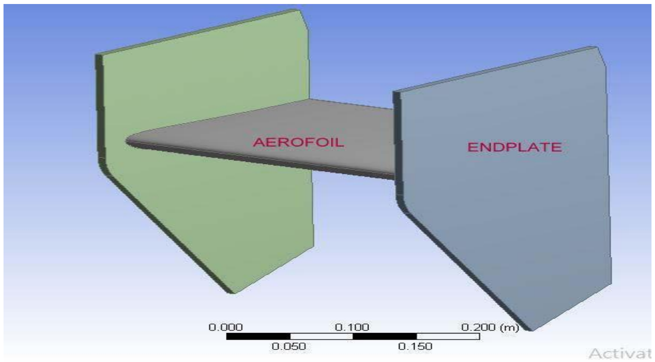

Fig. -1 Physical model of rear wing

IFERP connecting engineers... developing research

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 4, April 2018

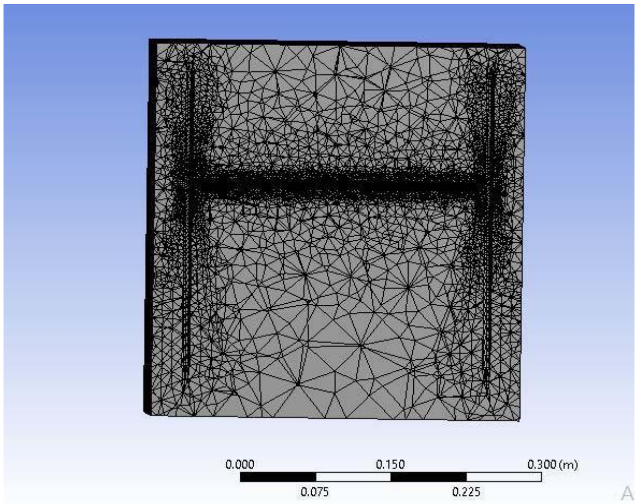

Fig.-2 (a) the cross-sectional view of generated mesh.

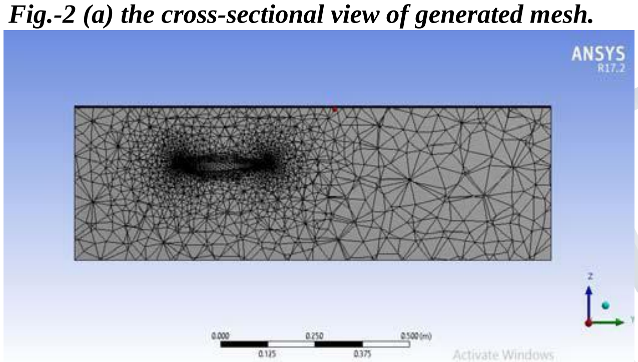

Fig.-2 (b) the cross-sectional view of generated mesh

C. Boundary Conditions

- Inlet boundary – Velocity Inlet (180 KMPH and 120 KMPH)
- Outlet boundary – pressure outlet
- Top and sides Boundaries – symmetry
- Car wing body – no slip condition
- Bottom – No Slip Conditions.

D. Governing Equations.

1. Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0; \; i = 1,2,3;$$

2. Navier-Stokes equations

$$\frac{\partial (u_i)}{\partial t} + u_j \frac{\partial (u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \left(\nu \frac{\partial u_j}{\partial x_j}\right) + F_i; \; i, j = 1,2,3;$$

ν = The effective viscosity. **P** = Pressure of fluid.

u = Velocity of fluid in x direction. **ρ =** Density of fluid.

3. Lift co-efficient, Cl

$$Cl = \frac{L}{\left(\frac{\rho V^2 A}{2}\right)}$$

4. Drag Co-efficient, Cd

$$Cd = \frac{D}{\left(\frac{\rho V^2 A}{2}\right)}$$

IV. VALIDATIONS

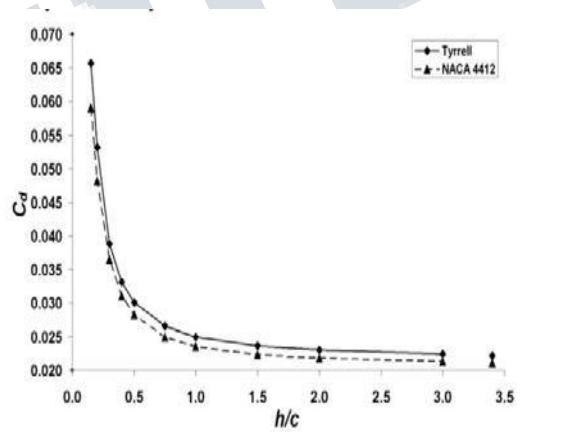

Fig. – 3 Cd vs h/c, data of Jonathan W. Vogt el at. [8]

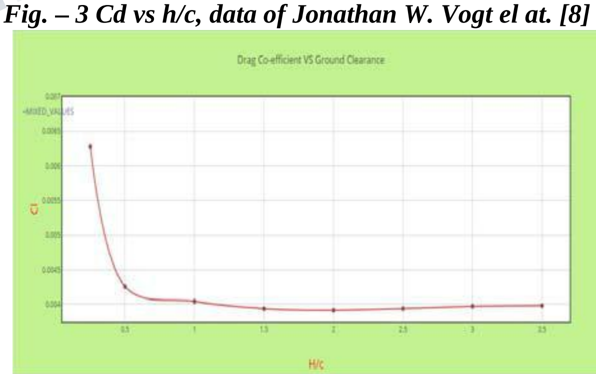

Fig. – 4 Cd vs h/c of calculated data for rear wing of car.

The present numerical results in terms of co-efficient of drag (Cl) and various ground clearance (h) of NACA 4412 at 6 AOA (°) have been evaluated. Then, it compares data of Jonathan W. Vogt el at. [8], they have been researched about Cd vs. ground clearance of NACA 4412 as well as

IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 4, April 2018

Tyrell profile. They have same geometry as we have but they are calculated of 2-dimensinal flow. They perform various analyses for both inverted NACA 4412 and Tyrell aerofoil at AOA is equal to 6°. As from above plot, we get almost same fashion as Jonathan el at. Performed but, there is somechange in value of Cd at different ground clearance. This happens due to we have done analysis in 3-dimensional geometry.

V. RESULTS AND DISCUSSIONS

The various analyses performed on model of different AOA chosen at two different speeds of 120KMPH and 180 KMPH. Different values of Cl and Cd are obtained.

Table-I Cl and Cd w.r.t Different AOA At 120 KMPH

AOA (°)	Cl	Cd
-2	-0.013659	0.0014486
0	-0.027757	0.0017693
2	-0.042516	0.0022399
6	-0.071377	0.0039871
10	-0.099952	0.0068085

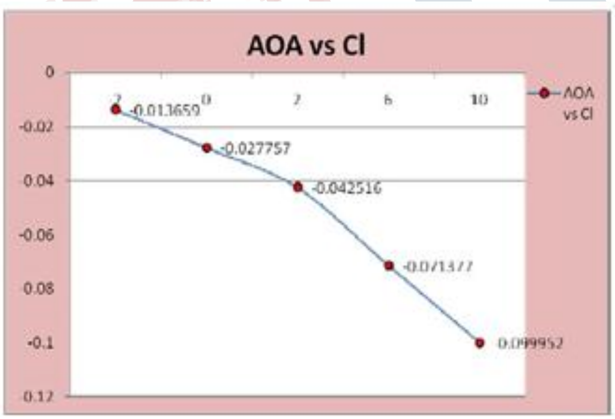

Fig. – 5 AOA vs Cl for 120 KMPH

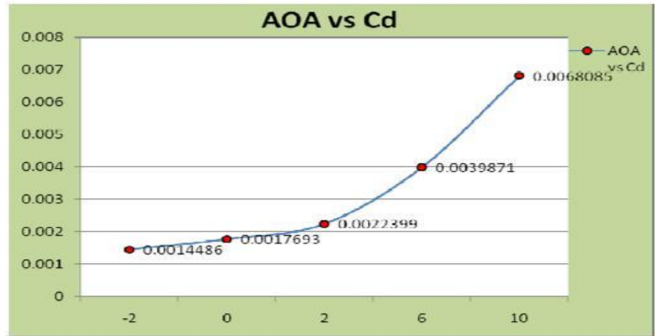

Fig. – 6 AOA vs Cl for 120 KMPH

From calculated data for NACA 4412 aerofoil at different AOA at 120 KMPH, It shows that there is steady increase in Negative lift of rear wing of F1 car. The fashion is observed in co-efficient of drag Cl w.r.t increase in AOA (°) -2 to 10. The values Cl of rear wing is negative due to reverse arrangement of aerofoil. The values of Cl are decreases from -0.013659 to -0.099952, when AOA (°) are change from -2 to 10. Whereas the values of Cd are increase from 0.0014486 to 0. 0068085, when AOA (°) are change from -2 to 10.

Table-II Cl and Cd w.r.t Different AOA At 180 KMPH

AOA (°)	Cl	Cd
-2	-0.013929	0.0013536
0	-0.029854	0.0018286
2	-0.042993	0.0021547
6	-0.072049	0.0039178
10	-0.10078	0.0067524

From calculated data for NACA 4412 aerofoil at different AOA (°) at 180 KMPH, It shows that there is steady increase in Negative lift of rear wing of F1 car. The fashion is observed in co-efficient of drag Cl w.r.t increase in AOA (°) -2 to 10. The values Cl of rear wing is negative due to reverse arrangement of aerofoil. The values of Cl are decreases from -0.013929 to -0.10078, when AOA (°) are change from -2 to 10. Whereas the values of Cd are increase from 0.0013536 to 0. 0067524, when AOA (°) are change from -2 to 10.

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 4, April 2018

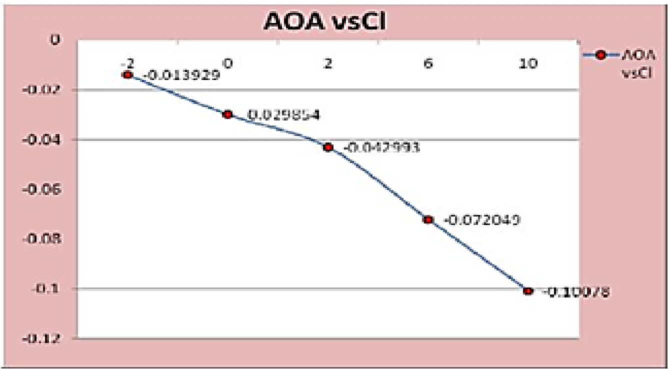

Fig. – 7 AOA vs Cl for 180 KMPH

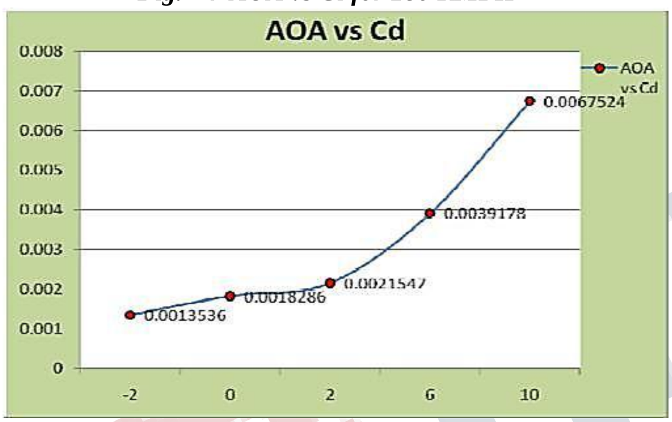

Fig. – 8 AOA vs Cl for 180 KMPH

We can see that as AOA of aerofoil leads to 10°, the pressure distribution n the upper surface of aerofoil is getting higher as compared to lower AOA. That indicates high lift force generated at AOA is at 10°. In velocity distribution, velocity of flow is higher at bottom surface of aerofoil and also getting higher for AOA reaches to 10°.

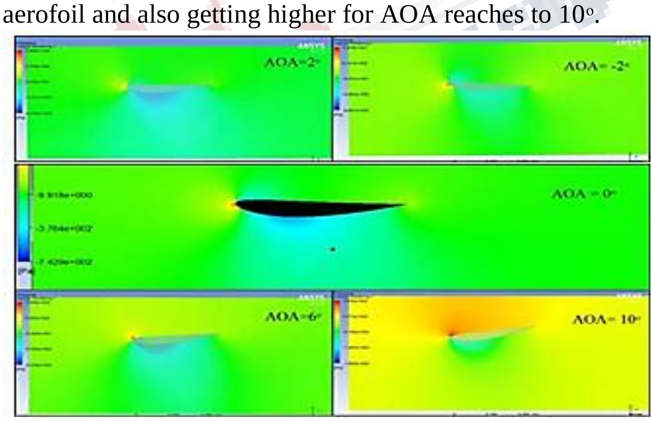

Fig.- 9 Pressure distribution for NACA 4412 at 120 KMPH

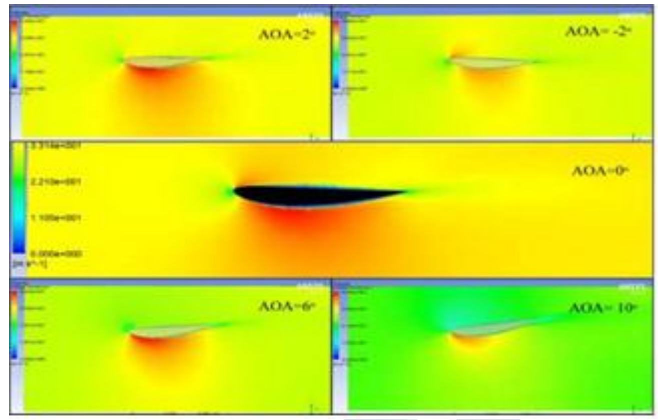

Fig.- 10 Velocity Varation of NACA 4412 at 180 KMPH

For both 120 kmph and 180 KMPH similar effect of pressure and velocity distribution have been observed.

VI. CONCLUSIONS

Computational analysis using ANSYS FLUENT to predict flow around Rear wing of race car has been achieved. 3-D external flows around the rear wing of formula one car were investigated. In this study, three parameters were taken into consideration, AOA, speed of car, NACA4412 Aerofoil. By comparing the results of various cases of different AOA and speed of car, we can get information about aerodynamics of Formula one car. It is clear from the results obtained that at a specific height of spoiler and AOA, the change in CD is increased as the speed of the racing car increases. And also negative lift generate by rear wing of car is increase as the speed of car increases. Notably, the lower spoiler height tends to gives both higher CD and CL. Finally it was found that at a particular racing speed and spoiler height, Cd and Cl both increase as the AOA increases. The aerodynamic of racing cars can be improved by use of computational fluid dynamics. These tools provide results with good accuracy within short time scales. Without validation, results of simulations are indeed of no value. The set of results and observations included in this report suggests that CFD can be a very useful tool to support the aerodynamic design of race cars. .

REFERENCES

[1] Francesco Mariani, Claudio Poggiani, Francesco Risi, Lorenzo Scappaticci "Formula SAE car racing : Experimental and numerical analysis of the external aerodynamics" ELSEVIER-Energy Procedia,Volume 81, December 2015, Pages 1013-1029

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 4, April 2018

[2] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam “Numerical Study on Aerodynamic Drag Reduction of Racing Cars” ELSEVIER - Procedia Engineering Volume 90, 2014, Pages 308-313

[3] Sneh Hetawala, Mandar Gophaneb, Ajay B.K.c, Yagnavalkya Mukkamalad “Aerodynamic Study of Formula SAE Car” ELSEVIER - Procedia Engineering Volume 97, 2014, Pages 1198-1207

[4] C.V.Karthick, Bala Murugan, P.A.Nigal Ashik, P.Raju “CFD Analysis and Optimization of a Car Spoiler” Research India Publications - International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 (2015)

[5] Jasminder Singh, Dr. Jaswinder Singh, Ampritpal Singh, Abhishek Rana, Ajay Dahiya “Study of NACA 4412 and Selig 1223 airfoils through computational fluid dynamics” – SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 6–June 2015

[6] Xu-xia Hu, Eric T.T. Wong “A Numerical Study On Rear-spoiler Of Passenger Vehicle” World Academy of Science, Engineering and Technology 57 2011

[7] ARTICLE 274 - 2018 FIA FORMULA 4 TECHNICAL REGULATIONS

[8] Jonathan W. Vogt* and Tracie J. Barber “Variation of ground effect phenomena about downforce generating Tyrrell and NACA4412 aerofoils” Int. J. Aerodynamics, Vol. 1, No. 1, 2010