ISSN : 2248-9622, Vol. 6, Issue 7, (Part -1) July 2016, pp.47-53

RESEARCH ARTICLE OPEN ACCESS

Comparison of CFD Simulation of a Hyundai I20 Model with Four Different Turbulence Models

M. Vivekanandan*, R. Sivakumar**, Aashis. S. Roy***
*(Uttam Industrial Engg. Pvt. Ltd., Tiruchirapalli, Tamil Nadu, India,)
**(Mechanical Engg. Dept., Addis Ababa Science and Technology University, Addis Ababa, Ethiopia,)
***(Industrial Chemistry Dept., Addis Ababa Science and Technology University, Addis Ababa, Ethiopia,)

ABSTRACT
This article describes the CFD analysis of a Hyundai i20 car Model. The focus of this study is to investigate the aerodynamics characteristics of Hyundai i20 car model and the flow obtained by solving the steady-state governing continuity equations as well as the momentum conservation equations combined with one of four turbulence models (1.Spalart-Allmaras 2.k-ε Standard 3.Transition k-kl-ω 4.Transition Shear Stress Transport (SST)) and the solutions obtained using these different models were compared. Except transition k-kl-ω model, other three models show nearly similar velocity variations plot. Pressure variation plot are almost similar with K-ε and transition-SST models. Eddy viscosity plot are almost similar with K-ε and transition k-kl-ω models.

I. INTRODUCTION

Designing the shape of a car is very important in achieving a good performance aerodynamically. Computational Fluid Dynamics has played an important role in a car to determine the flow features and performance such as lift, drag, vehicle stability and cross wind parameters. The 3D numerical simulation method is used for various turbulence models such as Spalarat Allmaras, k- Standard, Transition K-KL-ω, and Transition Shear Stress Transport (SST) to simulate the aerodynamic performance of an i20 car model.

Spalart et al [1] reported a model to solve a transport equation. It is a one equation model formulated for the kinematic eddy turbulent viscosity. The model was designed specifically for aerospace applications where wall-bounded flows is involved and is shown to give good results for boundary layers which are subjected to pressure gradients adversely. This model is an effective low-Reynolds number model in its original form. It requires the viscosity-affected region of the boundary layer to be resolved properly. It is also gaining popularity in turbo-machinery applications.

Spalart et al developed a Spalart Allmaras model to solve aerodynamic flows. It is not calibrated for general industrial flows, and errors produced for some free shear flows is relatively large, especially for plane and round jet flows. Further, prediction of the decay of homogeneous, isotropic turbulence cannot be relied upon.

K-ε turbulence model proposed by Lauder and Spalding [2] is the model most commonly used in Computational Fluid Dynamics (CFD) for simulating mean flow characteristics for turbulence flow conditions. It is a two-equation model and gives a general description of turbulence by means of two transport equations. The original objectives of the k-ε model were to improve upon the mixing length model and to find an alternative to algebraically prescribing turbulence length scale in flows with moderate to high complexity.

The first transports variable determines the energy in the turbulence and is called turbulence kinetic energy.

The second transports variable is the turbulence dissipation which determines the rate with which the turbulence kinetic energy is dissipated.

The exact k-ε equations contain many unknown and immeasurable terms for a practical approach. The standard k- ε turbulence model used is based on our best understanding of the relevant process minimizing unknown set of equation which can be applied to large number of turbulence applications.

Transition k-Kl-Ω Turbulence Model is three-equation turbulence model was developed and tested by Walters and Leylek [3] first using the far-field dissipation rate and then later implemented using the inverse turbulence time scale (ω). The original concept for the addition of the laminar kinetic energy (kl) equation was to capture the effects of bypass transition only but now extended to the natural and mixed transition cases also.

The transition shear stress transport (SST) model was developed by Menter [4] to effectively blend the robust and accurate formulation of the model in the region near the wall region with the free stream independence of the model in the far field. To achieve this, the model is formulated into equations. The SST model is similar to the standard model, but includes some refinements such as adverse pressure gradient flows and transonic shock waves than the standard model.

Sneh Hetawal et al [5] carried out CFD simulation of a formula SAE car and suggested modifications to minimize lift and drag.

Z. Y. Chao et al [6] carried out the transient numerical simulation studies on aerodynamics of a simple shaped car when they cross side by side in opposite direction and concluded that the aerodynamic forces have a significant effect on the stability in handling the car.

Mattias Olander [7] carried out CFD Simulation of a Volvo Car in Slotted Walls Wind Tunnel and reported that the drag and lift coefficient were lower in the CFD simulations of the wind tunnel compared to the ordinary CFD simulations. The results of standard CFD simulations were opposite to that of real experiments studied in the tunnel. This was opposite to the real experiments when compared with the standard CFD simulation. The reason was mainly because of simplified geometry of the simulated tunnel and the uniform boundary condition at inlet, which resulted in a more uniform flow in the CFD simulations when compared with experiments.

Fischer et al [8] reported that the variation between experimental results and that of simulation is partly due to wind tunnel effects and ground simulation effects. A CFD study on Sedan scale model in an open wind tunnel was reported with and without wind tunnel effect which resulted in different pressure distribution and hence difference forces and surface pressure.

Kieffer et al [9] reported the CFD simulation on Formula Mazda race car using star CD CFD code. The simulation of turbulent air flow was performed on the front and rear wings using k-ε model. Coefficient of lift and drag had a significant effect on the ground effect. Stalling condition was observed for angles less than or equal to 12^o, to the horizontal.

Singh R [10] carried out CFD simulation studies on NASCAR racing car and reported that lift decreased because of rotating wheel and moving ground effect. In contrast, drag increased because of smaller interaction of the underbody flow with wake flow.

In this paper, authors report the CFD simulation solutions of an i20 car model with four different turbulent models that would enable to compare and choose a model that suits the simulation studies better and converge with a real time solution.

II. COMPUTATIONAL METHOD

For analysis purpose, CFD solver ANSYS FLUENT is used for calculation. The free steam temperature is 300 K, which is same as the environmental temperature. Figure 1 shows the three dimensional view of the i20 car model.

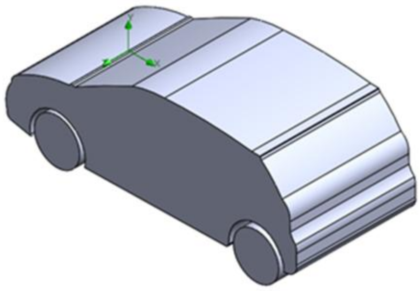

Figure 1 3D view of the i20 car model

The car profile, boundary condition and meshes were all created in the pre-processor. The pre-process is a program that can be used to produce two and three dimensional models using structure or unstructured meshes which can consists of variety of elements such as quadrilateral, and triangular in regions where greater computational accuracy is needed, such as the region closed to the profile.

The first step in performing a CFD simulation should be to investigate the effect of the mesh size on the solution result. Generally, a numerical solution becomes more accurate as more nodes are used but computation time and memory requirement increases with increase in nodes. The appropriate number of models can be determined by increasing the number of nodes until the mesh is sufficiently fine so that further refinement does not change the result.

III. RESULTS AND DISCUSSION

Figures 2, 3, 4 and 5 show the pressure cut plot with Spalart Allmaras, K-E, K-KL-Ω and SST models respectively for turbulent flow. All four models show a similar pressure variation at the front end and at the bottom of wheels of the car.

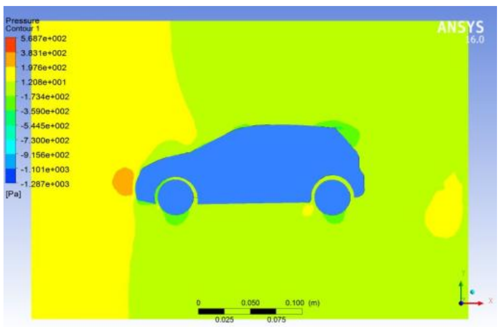

Figure 2 Pressure cut plot with Spalart Allmaras Model

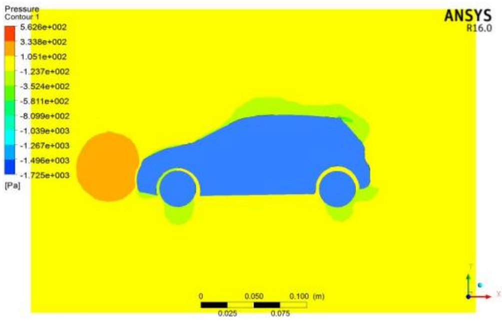

Figure 3 Pressure cut plot with K-E Model

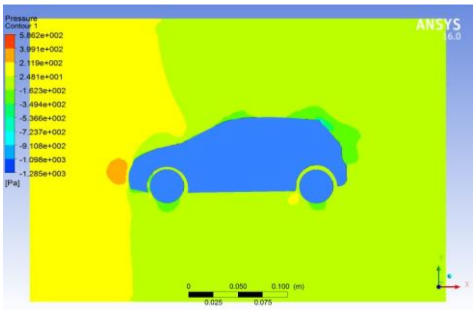

Figure 4 Pressure cut plot with Transition K-KL-Ω Model

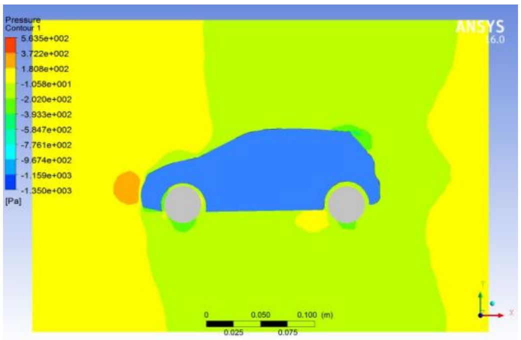

Figure 5 Pressure cut plot with Transition SST

Figures 6, 7, 8 and 9 show the surface pressure plot with Spalart Allmaras, K- ε, and K-KL-Ω and SST models respectively for turbulent flow. Green colour / blue colour shade represents negative pressure of decreasing order and hence boundary layer separation at the surface while yellow / orange / red colour shade represents positive pressure of increasing order. More variation in surface pressure is observed with transition K-KL-Ω turbulence flow model.

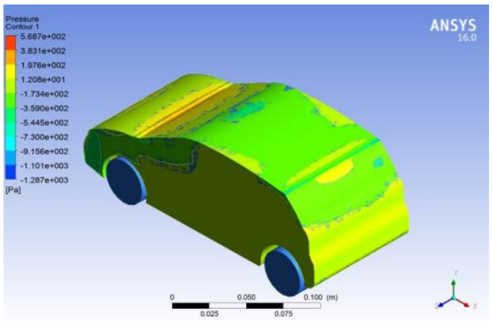

Figure 6 Surface Pressure plot with Spalart Allmaras Model

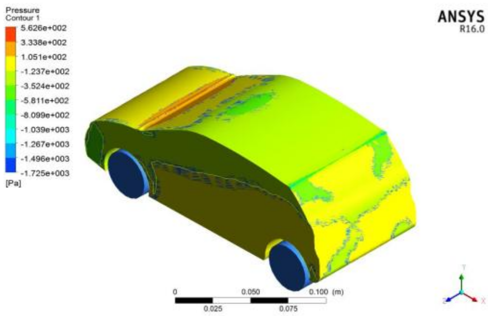

Figure 7 Surface Pressure plot with K- ε Model

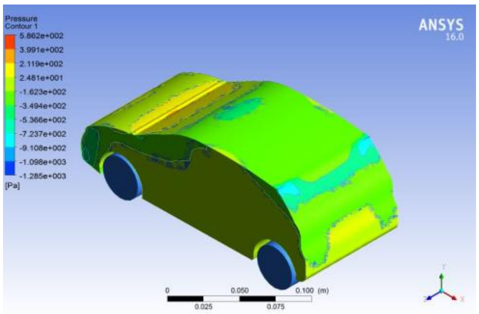

Figure 8 Surface Pressure plot with K-KL-Ω Model

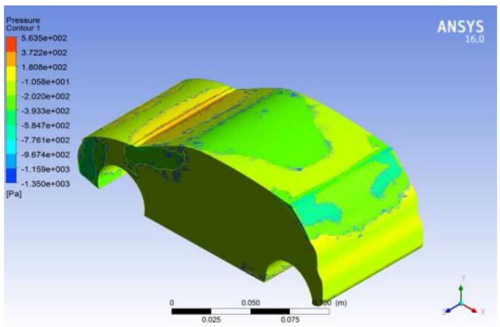

Figure 9 Surface Pressure plot with SST Model

Figures 10, 11, 12 and 13 shows the velocity cut plot with Spalart Allmaras, K- ε, K-KL-Ω and SST models respectively for turbulent flow. All four models show low velocities at the rear end of the car. K-KL-Ω model showed much lower velocity compared to the other three models.

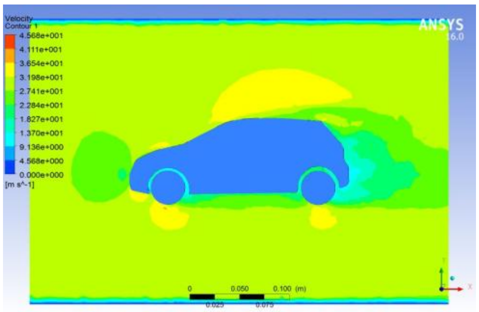

Figure 10 Velocity cut plot with Spalart Allmaras Model

Figures 14, 15, 16 and 17 show the surface velocity plot with Spalart Allmaras, K- ε, K-KL-Ω and SST models respectively for turbulent flow. All models show maximum velocity at the top and minimum velocity at the rear end of the car. The surface velocity variations are almost similar with all four models

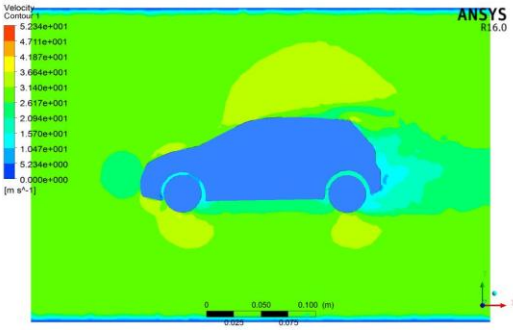

Figure 11 Velocity cut plot with K- ε Model

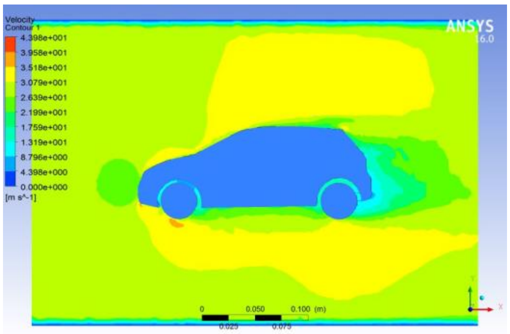

Figure 12 Velocity cut plot with Transition K-KL-Ω Model

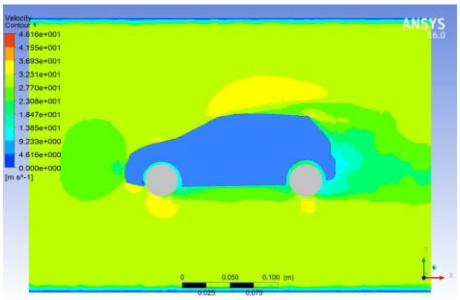

Figure 13 Velocity cut plot with Transition SST Model

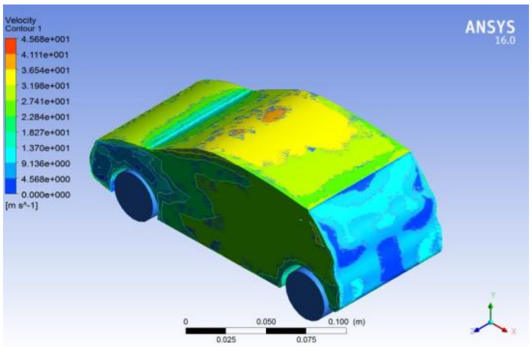

Figure 14 Surface velocity plot with Spalart Allmaras Model

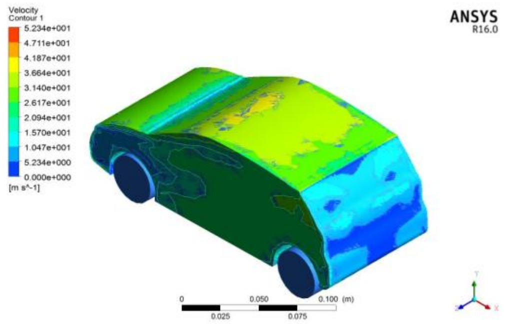

Figure 15 Surface velocity plot with K- ε Model

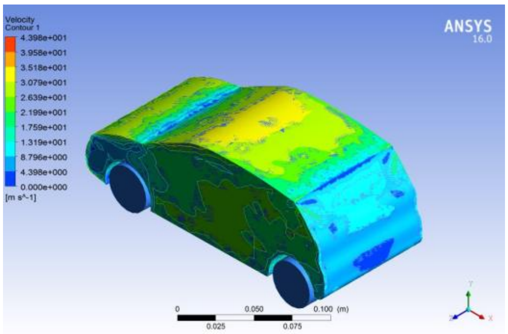

Figure 16 Surface velocity plot with Transition K-KL-Ω Model

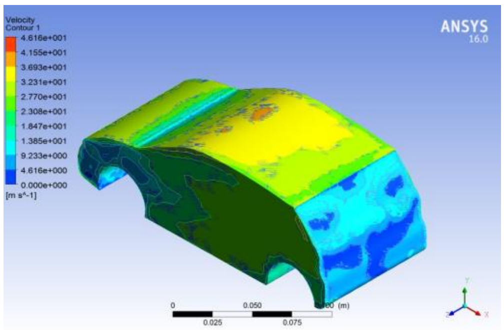

Figure 17 Surface velocity plot with Transition SST Model

Figures 18, 19, 20 and 21 show the eddy viscosity cut plot with Spalart Allmaras, K- ε, K-KL-Ω and SST models respectively for turbulent flow. All four models show variations in eddy viscosity behind the car. K-KL-Ω model showed more turbulent eddies and hence more variations in eddy viscosity than the other three models.

Figures 22, 23, 24 and 25 show the eddy viscosity plot at the surface with Spalart Allmaras, K-ε, K-KL-Ω and SST models respectively for turbulent flow. Spalart Allmaras and SST models show no variations at the surface. K-ε and K-KL-Ω models show small variations in eddy viscosity on the sides of the car.

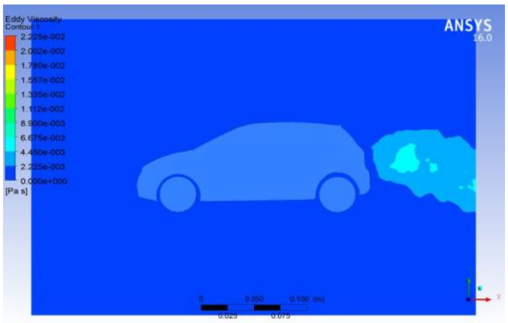

Figure 18 Eddy viscosity cut plot with Spalart Allmaras Model

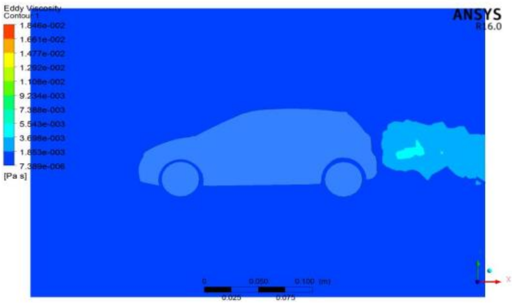

Figure 19 Eddy viscosity cut plot with K- ε Model

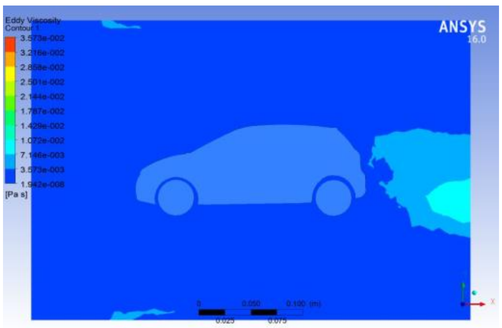

Figure 20 Eddy viscosity cut plot with Transition K-KL-Ω Model

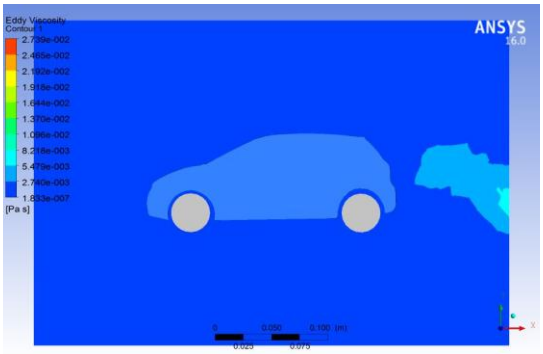

Figure 21 Eddy viscosity cut plot with Transition SST Model

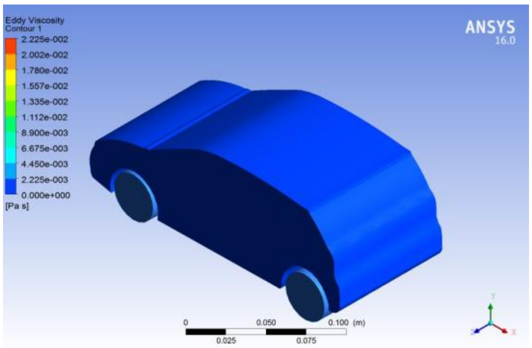

Figure 22 Eddy viscosity plot at the surface with Spalart Allmaras Model

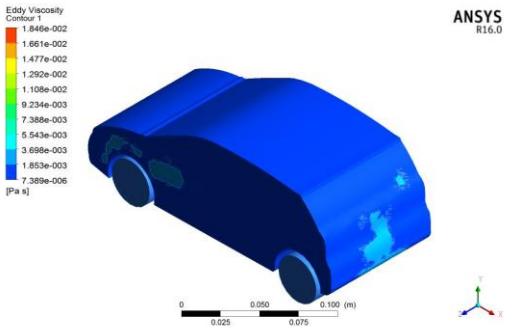

Figure 23 Eddy viscosity plot at the surface with K-ε Model

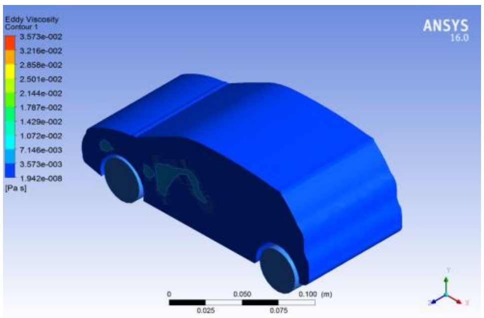

Figure 24 Eddy viscosity plot at the surface with Transition K-KL-Ω Model

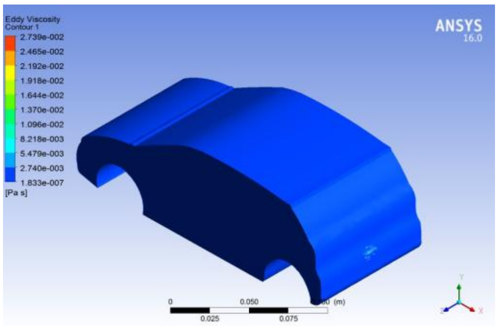

Figure 25 Eddy viscosity plot at the surface with Transition SST Model

Figures 26, 27, 28 and 29 show the velocity trajectory plot with Spalart Allmaras, K- ε, K-KL-Ω and SST models respectively for turbulent flow. The trajectory obtained with Spalart Allmaras and K-KL-Ω models are similar. Flow trajectory observed with K-ε and transition SST models are having similarities with mixed stream lines behind the car.

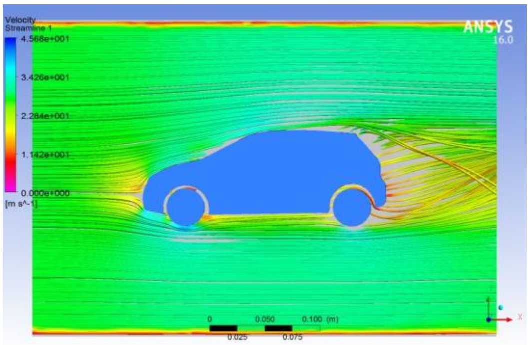

Figure 26 Flow trajectory with Spalart Allmaras Model

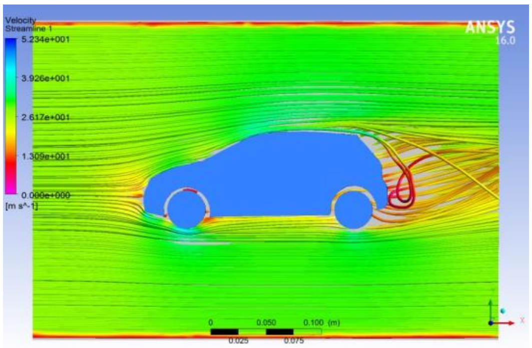

Figure 27 Flow trajectory with K- ε Model

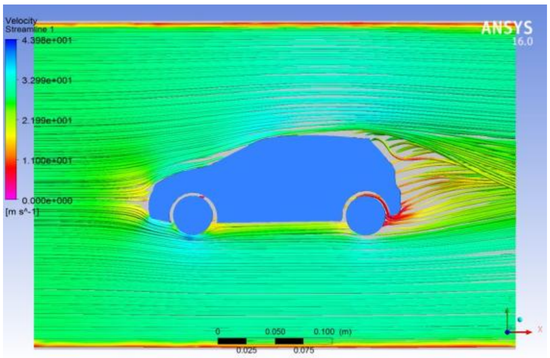

Figure 28 Flow trajectory with Transition K-KL-Ω Model

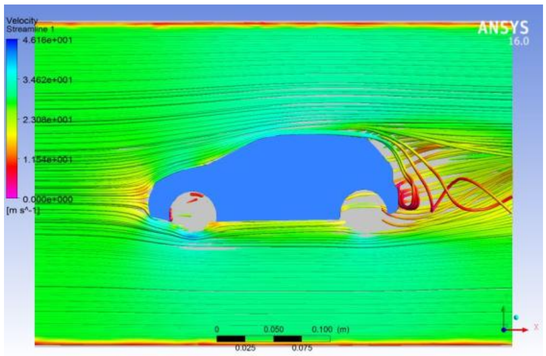

Figure 29 Flow trajectory with Transition SST Model

Table 1 Average values of Pressure, Velocity and Eddy viscosity calculated using four different turbulence Models:

Turbulence Model	Pressure Pa	Velocity ms^{-1}	Eddy Viscosity Pa.s
Spalart Allmaras	-255.529	14.964	0.000234
K-E	-266.533	14.3556	0.000159
Transition K-KL-Ω	-222.889	8.5976	0.000162
Transition SST	-265.724	13.704	0.0000868

The average velocity calculated from the turbulence models is shown in Table 1. K-KL-Ω model showed more variation in average pressure and average velocity compared to other three models, while Spalart Allmaras and transition SST models showed maximum and minimum eddy viscosity values respectively.

Figures 30, 31, 32 and 33 show the variations in residuals with increase in number of iterations and convergence of solution for the four different turbulence models studied.

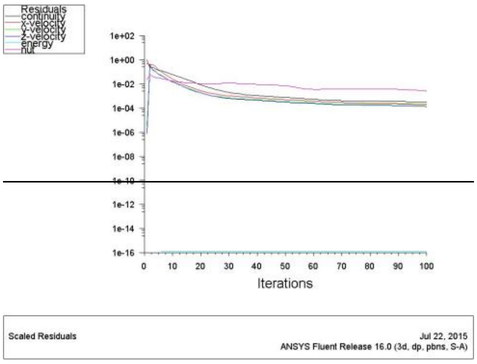

Figure 30 Residuals vs. Iteration graph of Spalart Allmaras Model

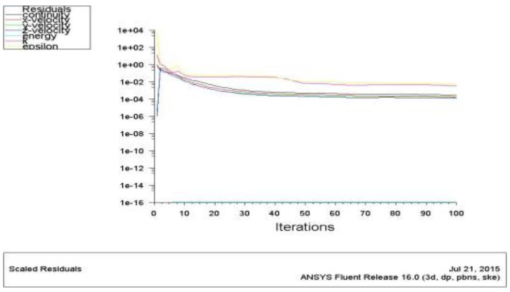

Figure 31 Residuals vs. Iteration graph of K-ε Model

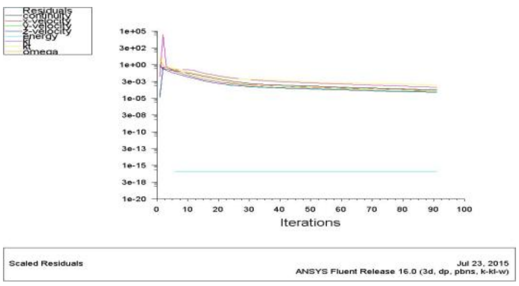

Figure 32 Residuals vs. Iteration graph of Transition K-KL-Ω Model

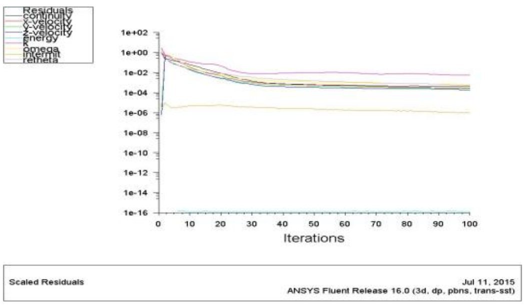

Figure 33 Residuals vs. Iteration graph of Transition SST Model

IV. CONCLUSIONS

An i20 car model was simulated with four different turbulent flow models - Spalart Allmaras, K-ε, K-KL-Ω and SST models respectively. More variation in surface pressure is observed with transition K-KL-Ω turbulence flow model. K-KL-Ω model showed much lower velocity compared to the other three models. The surface velocity variations are almost similar with all four models K-KL-Ω model showed more turbulent eddies behind the car. K-ε and K-KL-Ω models show small variations in eddy viscosity on the sides of the car. K-ε and transition SST models are having similarities with mixed stream lines behind the car. K-KL-Ω model showed least average pressure and average velocity while SST model showed least eddy viscosity.

REFERENCES

[1]. Spalart, PR and Allmaras SR, A One-Equation Turbulence Model for Aerodynamic Flows. AIAA Paper, 1992, 92-439.

[2]. Launder BE and Spalding DB, The numerical computation of turbulent flows. Comp. Meth. Appl. Mechanics Engr., 1974, Vol. 3, 269-89.

[3]. Walters D.H. and Leylek J.H. (2003b) Simulation of Transitional Boundary-Layer Development on a Highly-Loaded Turbine Cascade with Advanced RANS Modeling, Proceedings of ASME Turbo Expo, 2003-38664.

[4]. Menter FR, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA J., 1994, Vol. 32, 1598-1605.

[5]. Sneh Hetawal, Mandar gophane, Ajay B.K. and Yagnavalkya Mukkamala, Aerodynamic study of Formula SAE car, Procedia Engineering, 97, 2014, 1198 – 1207.

[6]. Zhang Ying-chao and Fu Li-min, Transient aerodynamic numerical simulation of simplified shape cars under condition of opposite meet side by side in tunnel, Journal of Jilin University, 2006, 03.

[7]. Mattias Olander, CFD Simulation of the Volvo Cars Slotted Walls Wind Tunnel Master's Thesis in Solid and Fluid Mechanics, Chalmers University of Technology, Goteborg, Swedon, 2011, 33.

[8]. Oliver Fischer, Timo Kuthada, Jochen Wiedemann, Patrick Dethioux, Richa Mann and Brad Duncan, CFD validation study for a Sedan scale Model in an Open Jet Wind Tunnel, SAE International, 2008, 1-0385.

[9]. W. Kieffers, S. Moujaes and N. Ambya, CFD study of section characteristics of Formula Mazda race car wings, Mathematical and Computer Modelling, Vol.43 (11-12), 1275-1287.

[10]. Singh R. CFD Simulation of NASCAR Racing Car Aerodynamics, SAE Technical paper, 2008, 01-0659.