ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Analysis on Simulations of Automobile

Dunna Sandeep
M.Tech (CAD/CAM Student),
Department of Mechanical Engineering,
Sree Dattha Institute of Engineering and Science,
Ibrahimpatnam, Hyderabad, India.

D.Rahul JI, M.E
Assistant Professor,
Department of Mechanical Engineering,
Sree Dattha Institute of Engineering and Science,
Ibrahimpatnam, Hyderabad, India.

ABSTRACT:
In this paper, ANSYS CFX method is used to simulate a single car model with and without a spoiler and by using two types of mesh. The solution of the Reynolds average Navier Stokes equations (RANS equations) has been achieved by using two models such as K-Epsilon and K –Omega Turbulence model will be analysed. In this report, mesh quality, boundary layer and turbulent y+ value simulation has been thoroughly analysed and solution for both the models has also been compared and discussed the results. We use the ANSYS software to determine the drag and lift forces at different turbulence kinetic energy variables k-Epsilon and K-Omega for the given vehicle domain. Further, the effects of aerodynamic are verified with and without the spoiler.

1. INTRODUCTION:
Computational Fluid Dynamics (CFD) is a combination of computer Science, Numerical Mathematics and modern Fluid Dynamics. CFD uses numerical methods to analyse and solve the fluid flows problems. CFD is a method to research and examine wide range of problems in heat transfer and fluid flow. Automotive aerodynamics is one of the important applications of CFD. The development of a car involves a wide range of key components, material selection for lightweight body and consideration of aerodynamics for these components is important aspects to design. In automotive industry CFD used to minimize the drag force and increase the down force (negative lift) which helps to stabilize the vehicle, which will leads to decrease in fuel consumption and shape design.

In racing car industry such as Formula 1 a common interest is the necessary to keep a car on the ground and this could be obtained by attaching a spoiler at the rear of the car body. Most of the aerodynamic evaluation of air flow has been carried out computationally by computational fluid dynamics software by ANSYS.

Aim of stimulation models
• To create a quality Computational Fluid Dynamics simulation of a single car model and extract meaningful data.

• To create quality Computational Fluid Dynamics simulation of a single car with a spoiler and to study the aerodynamic effects of the spoiler on the car.

• To create quality CFD simulations of two cars closely follow each other to study drag phenomenon.

2. METHODOLOGY:
1. The physical boundaries are defined for the given problem.

2. The volume occupied with the fluid is divided by discrete cells and meshing can be uniform or non-uniform.

3. Boundary condition is defined and involves specific fluid behaviour and properties at boundaries of the problem.

4. The simulation is started and the equations are solved at steady state.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

5. Finally a postprocessor is used for analysis and visualization of the results.

2.1.Car Model without Spoiler:

A single car model has been created by using the given construction points and designed the model in a specified computational domain by using ANSYS software. Construction points used to construct the car model are as shown below.

X(m)	Y(m)
0	0.075
0	0.113
0.025	0.063
0.025	0.119
0.025	0.176
0.189	0.057
0.189	0.214
0.302	0.050
0.302	0.220
0.440	0.050
0.440	0.289
0.566	0.050
0.566	0.302
0.755	0.057
0.755	0.277
0.818	0.214
0.956	0.069
0.975	0.082
0.975	0.126
0.975	0.182
1	0.082
1	0.113

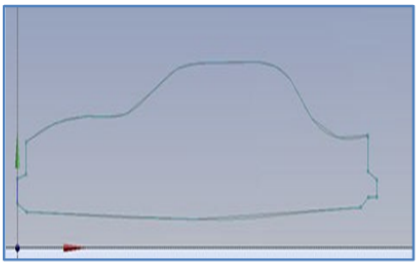


Table 1 construction points for the car model without spoiler

Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

2.2. Car Model with Spoiler:

Construction points used to construct spoiler:

X (m)	Y (m)
0.8800	0.2663
0.8868	0.2595
0.8868	0.2718
0.8978	0.2567
0.8978	0.2711
0.9252	0.2595
0.9252	0.2690
0.9526	0.2670
0.9526	0.2718
0.9800	0.2800

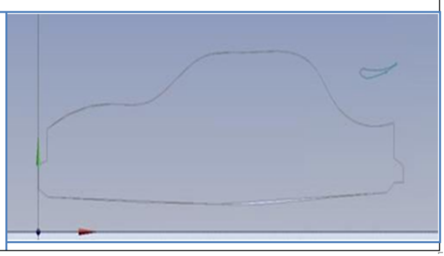


Table 2 construction points for the car spoiler.

The car model geometry is normalised by its length from nose to tail of the car model. The characteristics length of the car model is 1 and with the free stream velocity 100 Km/hr (27.78m/s), Reynolds Number is 1.84 x106 and turbulence intensity assumed is 5%.

2.3. Creating computational domain:

The computational domain means airflow in following sections which is created by using given boundary points.

X	Y
-8	0
-8	3
18	0
18	3

Table 3 Construction points for computational domain

3. PROCEDURE TO CREATE CAR MODEL:

Step 1: Create 3D Geometry in Computational Domain On generate the car geometry form the given construction point and then suppressed onto the 3D air domain in design model. The assumption is that the car is moving forward with a constant velocity 'U' through the air. But this is interpreted in simulation, if the air is moving towards the car with the same velocity but the opposite direction the car is stationary. The entire computational domain is done in the air domain. Therefore, the car is suppressed in the fluid domain and suitable boundary condition is assigned. The surface of the car body: inlet, sky, side plane, symmetry plane, ground, outlet, car body and spoiler.

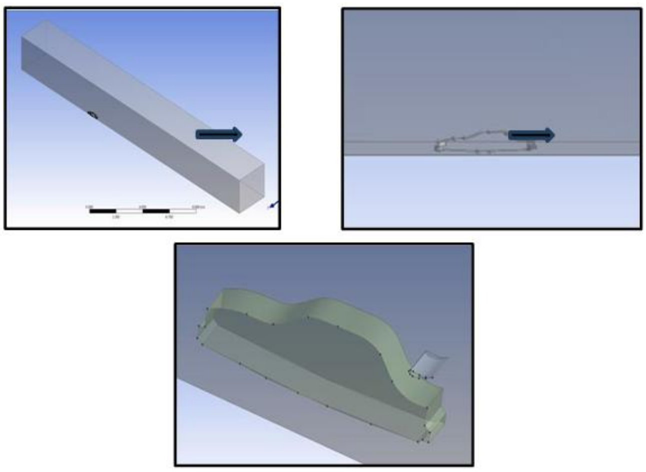


Figure 1 3D geometry

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Step 2: Meshing the Model

Meshing is one of the important steps for simulation in ANSYS. A suitable mesh will give meaningful and accurate solution whereas wrong meshing will give inaccurate or wrong answers.

In order to know the turbulence properly the mesh should be grid independently and also it should be fine as much as possible especially at the boundary layer. On the other hand, greater number of mesh element in the car model which leads to finer meshing and this will increase the computational cost.

Therefore, selecting mesh element size and applying suitable inflation on boundary layer will optimize in the computational cost and accurate results. Turbulence can be properly obtained when the mesh is done independent and fine including boundary layers. The flow domains are split into smaller subdomain. The subdomains are often called elements or cells and collections of these cells called as mesh.

Mesh Sizing: To account all the changes of the flow at the car model, the mesh size of all the surface of the car body has been taken 2.5cm or mesh to element with size 0.025m. On the other hand, the air domain is not that important except the near region of the car body, so it is kept default setting for meshing.

The element of sizing is kept soft for smooth transition between the coarse to fine elements of the car model. The inflation is created on the body domain and on the surface of car body and spoiler. The inflation option to first layer thickness is 0.0013m and maximum layer is 16. The face sizing is created by changing the scoping method to name selection for the car and spoiler model.

Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

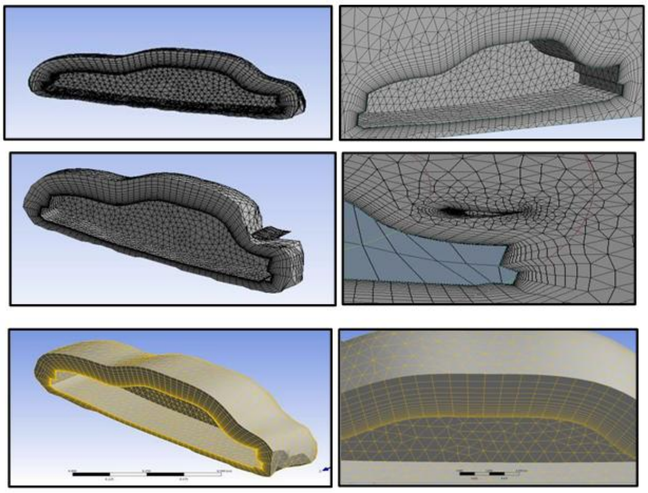


Figure 2 Mesh of the car model.

Step 3: Boundary layer mesh

In the ANSYS CFX, the method provided for turbulence models involving the boundary layer simulation. To know all the effect in the boundary layer of the car body, it is important to use inflation layer into the boundary layer with finer mesh. In the turbulence model, the first layer thickness setting is useful to save time in meshing because the value used by constant inflation layer. The quality of the mesh is determined by the shape of the individual cells. The key factor that affects the quality of the cell is skewness, aspect ratio, angle between the adjacent element of the cells and determinants. Skewness is primary quality measure for a mesh. It determines how close to idea a face or cell is. In single car model the skewness average of k-epsilon model is 0.26648 and k-omega is 0.26611. The mesh element models have the shape to meet the requirements.

Step 4: Mesh element requirements

• Triangular $45< \theta< 135$
• Rectangular 1< Ratio of adjacent side < 5
•The shape of the mesh element is qualified for the requirements shown in the above figures.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Step 5: Boundary Condition and Setting

NAME	BOUNDARY TYPE	LOCATION	BOUNDARY DETAILS
Inlet	Inlet	Inlet	Normal speed 27.78 m/s, Flow Regime: Subsonic. Turbulence: Medium (Intensity = 5%)
Outlet	Outlet	Outlet	Average static pressure= 0pa
Symmetry	Symmetry	Symmetric plane	
Car body	Wall	Car body	Mass and Momentum: No slip wall. Wall Roughness: Smooth Wall
Ground	Wall	Ground	Wall U=27.78m/s, wall V =wall W=0
Far field	Opening	Sky and back plane	Flow Regime: subsonic, Turbulence: Zero Gradient Mass and Momentum: Entrainment, 0.

Table 4 Boundary condition

According to the data from the table and figure, in reality the car is travelling at a constant speed U forward and the air is assumed to be still. In this case, it is equivalent to placing a stationary car in the air with a free stream speed U moving against the car towards the positive x direction. The ground also moves together with air in same speed and direction as the air.

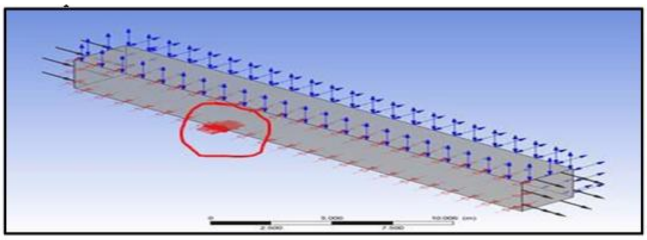


Figure 3 Boundary condition

Boundary condition

•The car is simulation to the drive at 100 km/h.

•The car is assumed to be fixed or set as wall to the model.

•The outlet boundary condition is set to pressure outlet with the gauge pressure of 0pa.

•The car contour, the top and the bottom of the wind tunnel are set as walls.

•The ground moves in the opposite direction at the same speed. Ground is set as wall, U velocity same as air is 27.78 ms^-1, V and W velocity are 0.

•The density of air is set as 1.185 kg/m^3.

•The dynamic velocity of air is 1.831 x 10^-5 kg/ms.

•Relative pressure of sky and side plane is 0; average static pressure of outlet is 0.

•Car body is set as wall.

•Drag coefficient: force_x()@carbody*2/1.185[kg m^-3]/27.78[m s^-1]^2/0.252[m]/0.17[m].

Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

Boundary Flows for CFX:

LOCATION	TYPE	MASS FLOW	MOMENTUM		
			X	Y	Z
Car body	Boundary	0.0000e+00	-7.0039e+00	4.3995e+00	1.8647e+01
Far field	Boundary	-2.3439e+00	-6.5293e+00	-4.2522e-03	3.9834e-03
Ground	Boundary	0.0000e+00	3.9507e-02	-4.6187e+00	-3.8233e-03
Inlet	Boundary	2.9627e+02	8.2306e+03	-3.0941e-08	3.1031e-02
Outlet	Boundary	-2.9604e+02	-8.2175e+03	2.3078e-01	-8.2808e-02
Symmetry	Boundary	0.0000e+00	0.0000e+00	0.0000e+00	-1.8604e+01

Table 5 Boundary flows

4. RESULTS ANALYSIS AND DISCUSSIONS FOR CAR MODEL WITHOUT SPOILER

4.1. Pressure Contour, Velocity Vector, Streamline, Turbulence Kinetic Energy for Car without Spoiler

Note: K-Epsilon and K-Omega are two approaches to simulate the model and have two different simulating results.

4.1.1. Pressure Contour for K- Epsilon Model and K-Omega Model

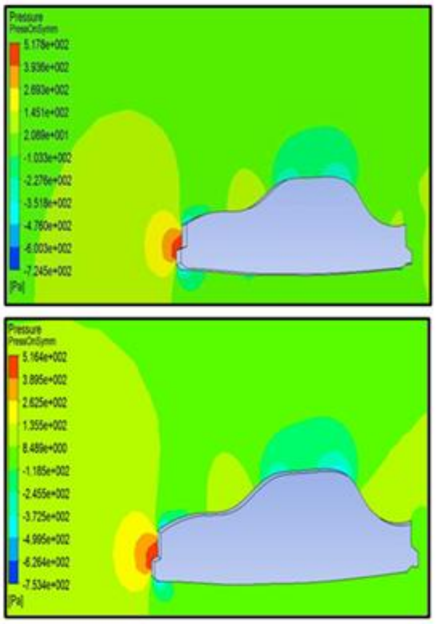


Figure 4 pressure contour for K-Epsilon model and K- Omega model

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

According to the above figure, for both models the high pressure at the front of the vehicle is due to the flat and projected front portion. This can be reduced by making the design more streamlined. The pressure level varies behind the rear windshield and it is lower in the K-Epsilon model and the K-Omega model has lower pressure behind the car. The pressure for the K-Epsilon model range varies from 549pa to -852pa and K-Omega model pressure range varies from 548 pa to -879 pa, hence the K- Omega model does not account for wall boundary conditions. Different calculation methods are used for two different models so the results for both the models are alike.

4.1.2 Velocity Vector for K- Epsilon Model and K-Omega Model

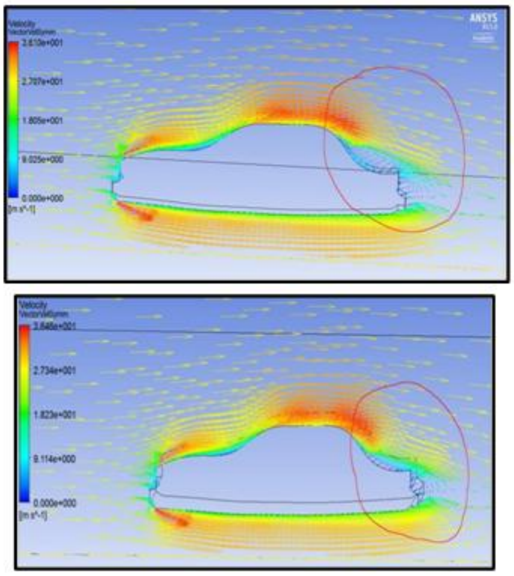
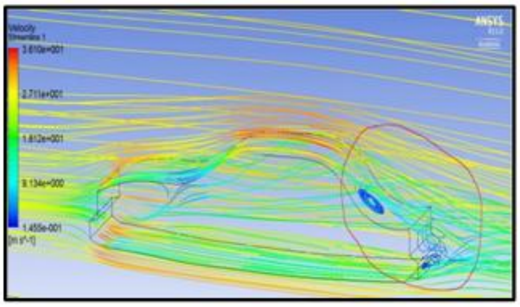


Figure 5 Velocity vector for K-Epsilon model and K-Omega model

The above figure shows velocity vector of K-Epsilon and K-Omega. The K- Omega model does not involve complex non-linear damping functions required for the K-Epsilon model and is more accurate. According to the velocity plots when compared with K- Epsilon and K-Omega models, the negative area in K-Epsilon is larger than that in K-Omega model. The results using K-Epsilon modal is more accurate than using K-omega model.

4.1.3 Streamline Plots for K- Epsilon Model and K-Omega Model



Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

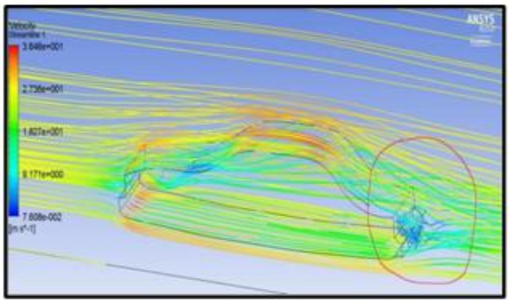


Figure 6 Streamline plots for K-Epsilon model and K- Omega Model

The above figure shows a vortex can be observed at the rear of the vehicle due to back pressure for the k-Epsilon model. The streamline flow over the rear of car body for k-Omega is more smooth and convergent. This proves k-Omega is most preferred for near wall problems. The high velocity at the top of the vehicle is due to the sudden change in profile.

From the streamline plots it can be seen that, the turbulence behind the rear windshield in the K-Epsilon is larger than K-Omega. For the area behind the car, the turbulence in K-Epsilon is smaller.

4.1.4. Turbulence Kinetic Energy for K-Epsilon Model and K-Omega model

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

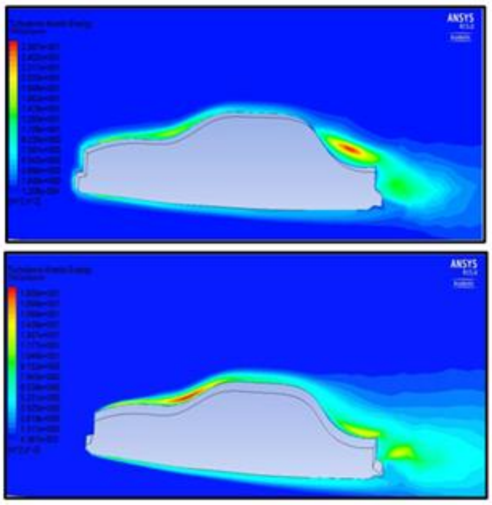


Figure 7 Turbulence kinetic Energy for K-Epsilon model and K-Omega model

The vehicle turbulence air is considered as a fluid medium through which our current model moves at constant speed U. There are many types of turbulence produced on roadways. The K-Epsilon model has become one of the most widely used turbulence models as it provides economy and accuracy for a wide range of turbulent flows. It is used for fully turbulent flow (high Reynolds number). It cannot handle low Re. and also flow separations. K-Omega is majorly used for near wall problems and damping functions are not required. The turbulence frequency is used as a variable for detecting length scales.

According to the above figure, the turbulence kinetic energy behind the rear windshield area in k-Epsilon model is around about 20 Kg m2/s2 and 6 Kg m2/s2 in the area behind the car. The above figure illustrates high level of turbulence at the rear end of the vehicle because of taper and small turbulence occurs over a larger distance in k-omega model. The kinetic energy behind the rear windshield does not change significantly because of the turbulence in K-Omega model. Nevertheless, the turbulence area behind the car is larger than in present K-Epsilon model. The K-Epsilon model, the turbulence kinetic energy behind the area of car is higher than K-Omega model.

4.2. Calculate the Lift and Drag Coefficient of Car Model without Spoiler

4.2.1. Drag

Drag is the force that acts opposite to the path of the vehicle's motion. Drag is detrimental to vehicle performance because it limits the top speed of a vehicle and increases the fuel consumption. Reducing drag in road vehicle has led to increasing the speed of vehicle, fuel efficiency handling, and acceleration

Drag coefficient (CD) = FD / (½ * ρ * V2 * AF) (1)

Where, FD is the drag force

ρ is the air density,

V is the free stream velocity

AF is the frontal area of the vehicle.

4.2.2. Lift

Lift can be manipulated to enhance the performance of a race car and decrease lap times. Lift is the force that acts on a vehicle normal to the road surface that the vehicle rides on.

Lift coefficient (CL) = FL / (½ * ρ * V2 * AT) (2)

Where FL is the lift force

ρ is the air density

V is the free stream velocity

AT is the area of the top surface of the vehicle

The forces and coefficient of the two models are as following below:

	K-EPSILON	K-OMEGA
DRAG FORCE	7.25692 N	8.05381 N
DRAG COEFFICIENT	0.375363	0.420639
LIFT FORCE	-3.96126 N	-4.87552 N
LIFT COEFFICIENT	-0.2245	-0.24306

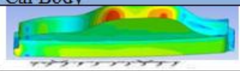
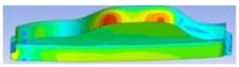
Table 6 Force and coefficient

The table shows, Drag coefficient and lift coefficient for the two turbulence car models. The two turbulence models have two different results. The drag force and coefficient in K-Omega are larger than in K-Epsilon model and the lift force and coefficient in K-Omega are both smaller than compared to K-Epsilon model. According to the results, it is understood that the car can hold the ground better in K- Omega model than in K-Epsilon model.

ISSN No: 2348-4845
International Journal & Magazine of Engineering,
Technology, Management and Research
A Peer Reviewed Open Access International Journal

Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

4.2.3. Calculate Co-efficient drag (Cd) and Co-efficient lift (C)

	Car Body	C	C	
Generic Automotive		0.32	0.43	
Model		-0.046	0.36	K-Epsilon
		-0.049	0.41	K-Omega

The coefficients in the model are lower than the generic car because of different shape of car body. The model is developed on spline, so angle is blunter, area smoother and drag is lower than generic. The bottom of the model is smoother and speed flow is slower than generic car. A symmetric model cannot cause to happen lift force and round body has lowest drag coefficient. The lift force of the model car is negative and the force is acting down towards the ground and this is useful for stability of the vehicle and driving.

5. RESULTS ANALYSIS AND DISCUSSIONS FOR CAR MODEL WITH SPOILER

5.1. Y Plus Value, Pressure Contour, Velocity Vector, Streamline, Turbulence Kinetic Energy with Spoiler for K-Epsilon Model

NOTE: Spoiler reduces the drag force and the lift force on the road.

5.1.1 Y Plus Plot with Spoiler for K-Epsilon Model

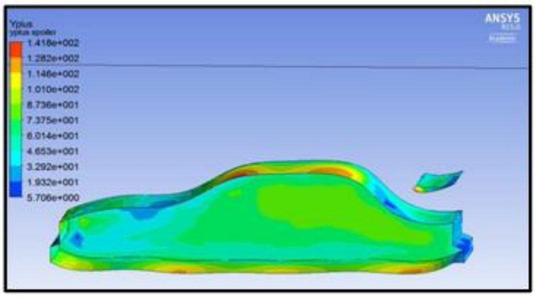


Figure 8 y plus plot with spoiler for K-Epsilon

According to the results, the Y plus values should not exceed 300 and by analysis the results we got: K-epsilon model car body is 143, K-omega model car body is 135 and K-epsilon model car body with spoiler is 142. So it meets all the following conditions. It is important to have the y+ values less than the allowed limit because it determines the quality of the mesh for the flow pattern and to have appropriate Reynolds number for the turbulent flow.

5.1.2 Pressure Contour Plots with Spoiler for K-Epsilon Model

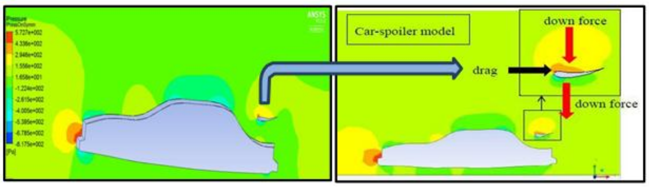


Figure 9 pressure contour plot with spoiler for K-Epsilon model

The above figure shows the pressure in the model of a car-spoiler. The pressure is focused on the front part of the vehicle and the rear part of the car body does not undergo extreme pressure. Furthermore, the area on the top of the spoiler is pushed downwards with approximately 200pa. When the high pressure air approaches in front of the windshield, the air travels over the windshield and accelerates and then decrease in pressure. This lower pressure produces a lift force on the car roof as the air passes over it. The pressure behind the car-spoiler is reduced. There is a large amount of pressure over the spoiler surface and this pressure is generating down force and results in better stability of the vehicle.

5.1.3. Velocity Vector with Spoiler for K-Epsilon Model

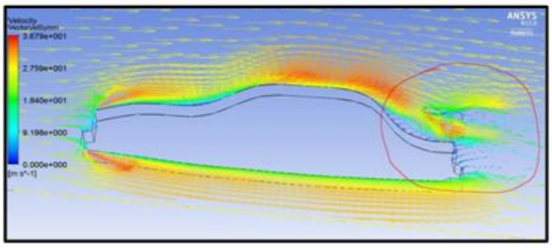


Figure 10 velocity vector with spoiler for K-Epsilon model

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

From the above figure, the velocity on top of the car especially at the spoiler end is more than the velocity under the car. This increases the downward force which helps the car to stay on the road.

5.1.4. Streamline Plot with Spoiler for K-Epsilon Model

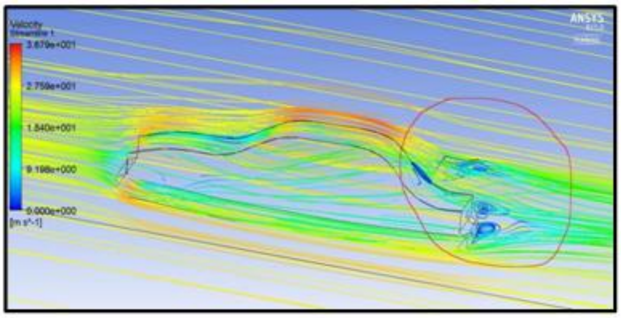


Figure 11 streamline plot with spoiler for K-Epsilon model

Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys

From the above figure, the spoiler splits the air flow at the rear and it consequently reduces the velocity on top of the car and reduces the formation of a vortex at the rear. But there is a high pressure zone created above the spoiler.

5.1.5. Turbulence Kinetic Energy with Spoiler for K-Epsilon Model

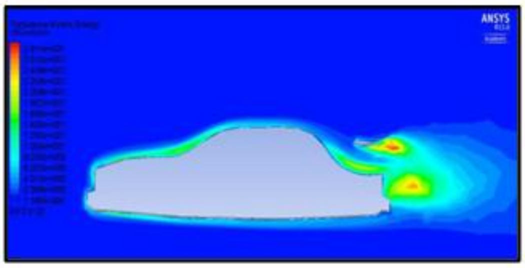


Figure 12 Turbulence kinetic energy with spoiler for K-Epsilon model

In the above figure, the turbulence area in the car spoiler is smaller compare to K- Epsilon model. The turbulence kinetic energy of K-Epsilon car model is approximately 23 Kg m2/s2 compared with K-Epsilon model which is smaller.

5.2. Lift and Drag Coefficient for the Car Spoiler Model

	K-EPSILON CAR	K-OMEGA CAR	K-EPSILON CAR	SPOILER
DRAG FORCE	7.15692 N	8.04382 N	8.70156 N 8.813052 N	0.111492 N
DRAG COEFFICIENT	0.365362	0.410639	0.444216	
LIFT FORCE	-3.86116 N	-4.76551 N	-5.50031 N -12.87884 N	-7.37853 N
LIFT COEFFICIENT	-0.0496725	-0.0613066	-0.66078	

DRAG COEFFICIENT	DRAG FORCE	LIFT COEFFICIENT	LIFT FORCE
0.44	8.81 N	-0.660	-12.87 N

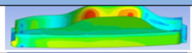
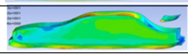
	CAR BODY	Cl	Cd
K-EPSILON		-0.049	0.36
CAR SPOILER		-7.37 N	0.44

Table 8 Lift and Drag Coefficient for the Car Spoiler Model

The Compared with the K-Epsilon car model Cl and Cd are both smaller than the car spoiler model. The drag force differs because of spoiler. The turbulence and flow created by spoiler that increases the drag force of the vehicle. The reason is that the spoiler suffers from turbulence and flow. The drag force of the car model with a spoiler is higher than the without spoiler. When it comes to lift coefficient and force have lots of changes. In the front and rear of the spoiler there is positive pressure and there would be additional drag force when compared with car model. The top aspect of spoiler is positive pressure and bottom receives negative pressure. The increase in the lift force is the main reason that the down force in car spoiler is large than in car model without spoiler.

6. CONCLUSION:

The Reynolds average Navier Stokes equations or single car body model can be solved in two ways by using K-Epsilon and K-Omega turbulence models. The single car with spoiler has been solved using K-Epsilon model. For the different types of mesh K-Epsilon and K-Omega the simulation results are different. The results of K- Omega models provides more smooth of the fluid medium over the car body surface reducing the swirls at the rear window region. The coefficient of drag and lift along with drag and lift forces has been calculated by using the equation.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Cl and Cd in K- omega model are both larger than in K-Epsilon model. K-Omega model have better down force than K-Epsilon model but has larger drag force. The car spoiler is a better way to increase the down force. The results will be different in case of adding a spoiler. In velocity vector plots with spoiler, an excess force acts on rear end resulting in an increase in downwards force which ultimately increases the frag on the car. The state of different car model in fluid domain can be simulated and different model have different actions in fluid domain and also different method of analysis. The computational fluid dynamics provides visual images and numerical data to simulate the car body in the fluid flow. The computational fluid dynamics provides simulations for design which can be used in practical. The results also help to reduce the fuel consumption and increase the stability of the vehicle. CFD contributes vehicle design, visible results and shows approximate results and helps simulations for designers to design the vehicle and improve accordingly.

REFERENCES:

[1] Anderson, ‘Race Car Aerodynamics Part 2: Lift and Drag’, 2014.

[2]http://jeffrin.hubpages.com/hub/Mesh-Generation-in-CFD-An-Overview.

[3] J.Ducoste, ‘An Overview of Computational Fluid Dynamics’, 2008

[4] Katz, J. 1995, ‘Race car aerodynamics: Designing for speed’, 1st edn, Robert Bentley.

[5] M.Cable, ‘An Evaluation of Turbulence Models for the Numerical Study of Forced and Natural Convective Flow in Atria’, 2009.

[6] Rakesh Jaiswal, Anupam Raj Jha, Anush Karki, Debayan Das, Pawan Jaiswal, Saurav Rajgadia, Ankit Basnet and Rabindra Nath Barman, Structural and Thermal Analysis of Disc Brake Using Solidworks and Ansys. International Journal of Mechanical Engineering and Technology, 7(1), 2016, pp. 67–77.

[7]Janvijay Pateriya, Raj Kumar Yadav, Vikas Mukhraiya and Pankaj Singh, Brake Disc Analysis with the Help of Ansys Software. International Journal of Mechanical Engineering and Technology, 6(11), 2015, pp. 114–122.