ISSN: 2278-0181
ETEDM - 2019 Conference Proceedings

# Design & Analysis of Diffuser Section of Four Wheeler

Mr. M. Viswanath [1], Mr. S. Sri Gowtham Raj Kumar [2], Mr. K. Thamarai Mohan [3], Mr. J. Yaseef [4].
1-Assistant Professor, 2,3&4-Students.
Department of Mechanical Engineering, Hindusthan Institute of Technology,
Othakkalmandapam, Coimbatore-641008, Tamilnadu, India

***Abstract*:- The aim of the paper is to examine the aerodynamic behaviour and performance of the diffuser. A diffuser in an automotive context is a shaped section of the car underbody which improves the cars aerodynamics properties by enhancing the transistion between the high velocity airflow underneath the car and the much slower freestream airflow of the ambient atmosphere. A two-dimensional analysis is to provide an understanding of the relation between diffuser height and associated diffuser angle for maximum downforce. Based upon the observations from the analysis the diffuser is designed with the efficient diffuser angle which produces maximum downforce.**

INTRODUCTION:

In earlier years study of aerodynamics had primarily been on the upper surface of the car, this provided a certain improvement but was not able to completely minimize the drag. Thus study of underbody aerodynamics gained momentum. Due to increasing demand for higher performance with lower emission in recent times, underbody aerodynamics of cars are playing a crucial role in improving the performance characteristics. This trend is seen in all major sport cars, and many studies have been conducted on the underbody aerodynamic characteristics of these cars.

AERODYNAMICS:

Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is a subfield of fluid dynamics and gas dynamics with much theory shared between them. Aerodynamic forces created by the relative motion of the vehicle are drag force, lift force and down force which produces noise by the air flowing around the car body. This air flowing within the car’s body is used for cooling the engine. Influence of drag and lift are the important parameters to study the aerodynamics of the cars

Drag is the aerodynamic force that is opposite to the velocity of an object moving through air or any other fluid. The aerodynamic drag force can be calculated using the formula below.

$$Fd = Cd * 1/2\, \rho\, v^2 * A$$

Where,

Fd = drag force, ρ = density of the air, v2= speed of the object relative to the fluid (m/s), A = reference surface area, Cd= drag coefficient.

Lift is the component of the pressure and wall shear force which acts normal to the moving body. The pressure difference between the top and bottom surface of the object generate an upward force that tends to lift.

Computational Fluid Dynamics:

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse the problems that involve fluid flows.The analysis of system is associated by means of computer based simulation.
Main elements of CFD:

- Pre-processor
- Solver
- Post-processor

-These three modules constitute the core of CFD.

Diffuser

A diffuser is a shaped section of the car underbody which improves the car's aerodynamic properties by enhancing the transition between the high-velocity airflow underneath the car and the much slower free stream airflow of the ambient atmosphere. It is a design that is installed under the car at the rear and is considered as a part of underbody tray. In the diffuser, the cross-section area increases, a bigger area will decrease the velocity of the fluid and increase the static pressure. A high static pressure recovery in the diffuser will lead to a higher base pressure. There are three non-dimensional parameter that affect the properties of a diffuser, the area ratio, the non-dimensional diffuser length and the diffuser angle.

CAD diffuser model:

Computational Fluid Dynamics simulation provides a numerical approximation to the equations that run for fluid motion.Use of the CFD to investigate a fluid problem, the steps followed are starting with writing the

ISSN: 2278-0181
ETEDM - 2019 Conference Proceedings

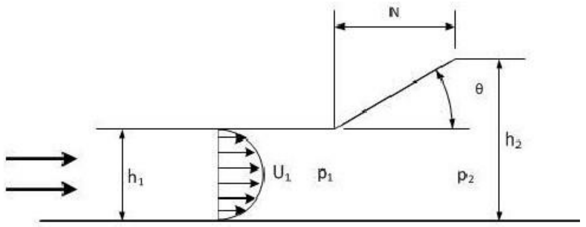



Figure 1. Shows velocity profile of diffuser.

fluid flow, generally partial differential equations.
Ultimately, the boundary conditions and the initial conditions of the specific problem are used to solve these equations.

The solution technique may be direct or iterative. Actual CFD modeling and simulation is carried in the ANSYS CFX12.0 Workbench Environment with ANSYS system of fluid flow (CFX).

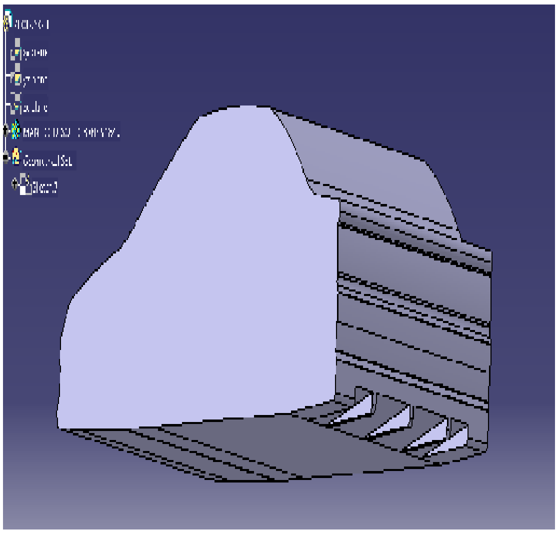


Fig 2.

ANSYS MODEL:
Geometry created using ANSYS Design Modeler software which is specifically designed for the creation and preparation of geometry for simulation. In engineering simulations, the geometry includes details not needed for simulation. Only the physics involved is to be included, simulating such a fully detailed model will increase solver run times. Design modeler and outline tree with a perforated fin with cross fin at center and with 12 mm size of perforation.
As per requirement of the present study, a domain has to be built around the fin to study mass flow and thus the heat flow from the fin, because the area of interest is the outside of fin, which is the interface between the air and fin surface. Thus, connections are required between the solid fin surface and the fluid domain consisting of air. Initially, fin and base was created. Around fin, a fluid domain of size 172 x140x130 mm was created. The idea to include the full fin the system in these domains is similar to the behavior of the whole actual model which will reflect the effect of major parameters like fin height, fin inter spacing, thickness of the fin and outside environment etc.

| | |
|---|---|
| Object Name | *Geometry* |
| State | Fully Defined |
| Definition | |
| Source | |
| Type | DesignModeler |
| Length Unit | Meters |
| Bounding Box | |
| Length X | 1437. mm |
| Length Y | 1394.6 mm |
| Length Z | 2424.1 mm |
| Properties | |
| Volume | 4.836e+009 $mm^3$ |
| Scale Factor Value | 1. |
| Statistics | |
| Bodies | 1 |
| Active Bodies | 1 |
| Nodes | 82340 |
| Elements | 452271 |

| | |
|---|---|
| Buoyancy Model | Non Buoyant |
| Domain Motion | Stationary |
| Reference Pressure | 1.0000e+00 [atm] |
| Heat Transfer Model | Isothermal |
| Fluid Temperature | 2.5000e+01 [C] |
| Turbulence Model | k epsilon |

Settings.

**RESULT & DISCUSSION:**

## Result Of Diffuser section at angle of 10 Degree:

## Velocity:

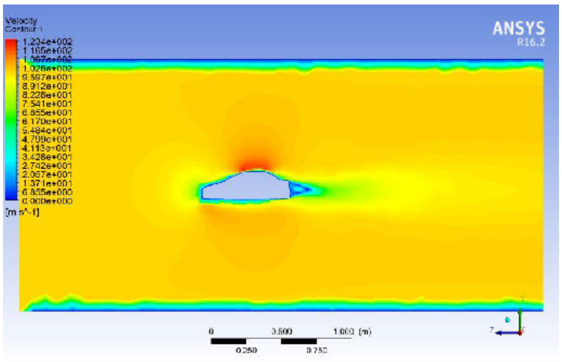



Fig 4(a).

Maximum velocity is **123.4 m/s** for the ZY middle plan section and for this the drag force is **117.315 N.**

## Pressure Distribution :

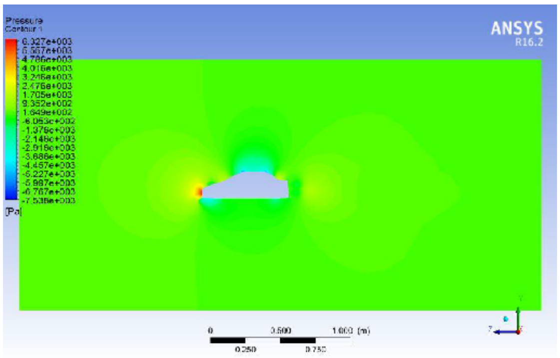
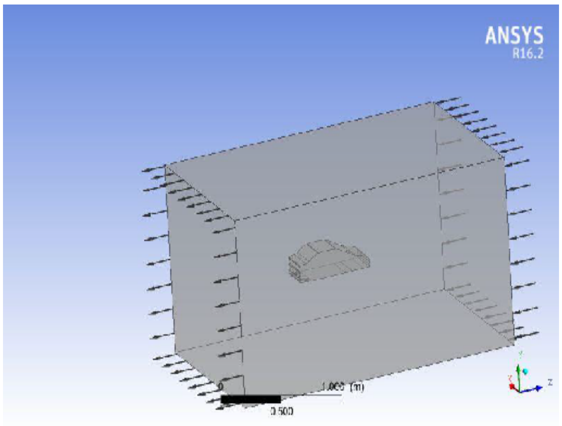



Fig 4(b).

## Turbulence kinetic Energy:



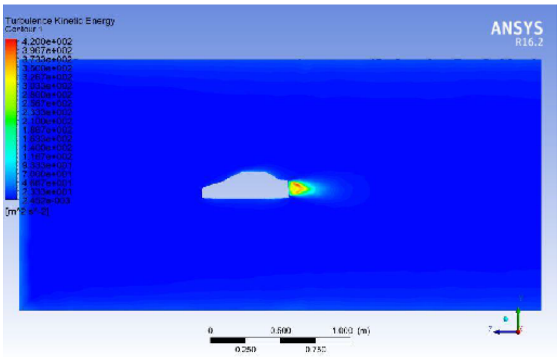
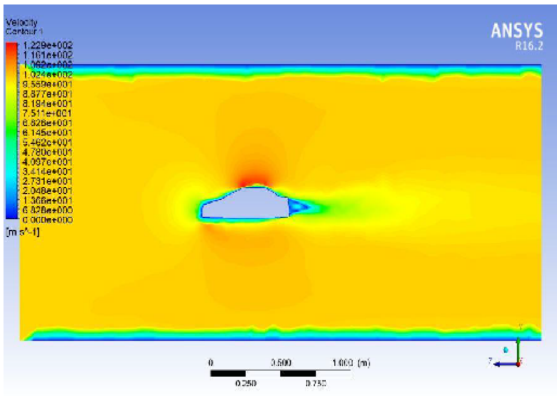




Fig 4(c).




## Result for when Diffuser section at 11 Degree:

## Velocity distribution:

Fig 5(a).

## Pressure Distribution:

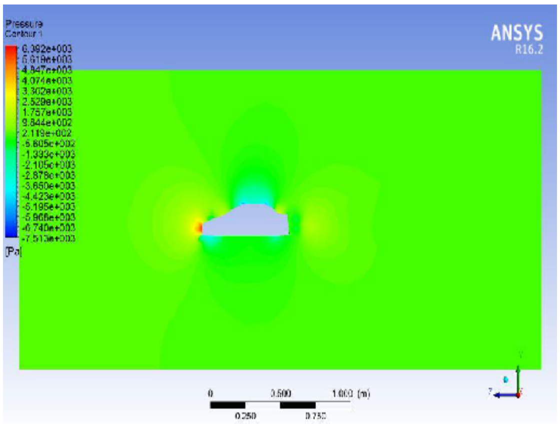



Fig 5(b).

## Turbulence Kinetic Energy:

ISSN: 2278-0181
ETEDM - 2019 Conference Proceedings

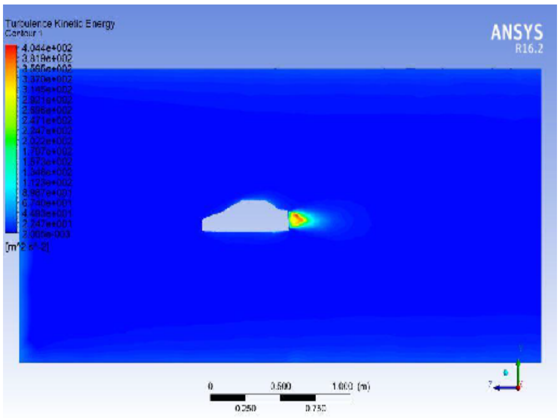



Fig 5(c).

**Result for when diffuser section at 12 degree :**

**Velocity distribution:**

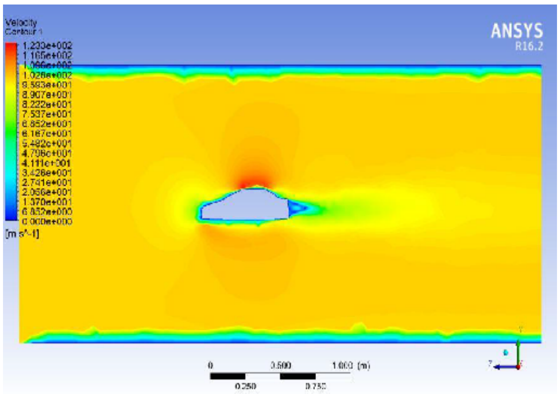



Fig 6(a)

**Pressure Profiles:**

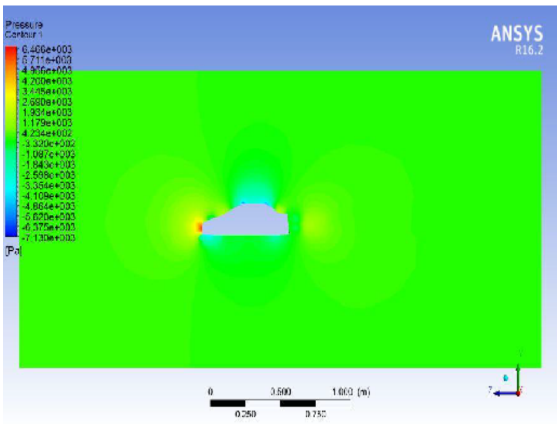



Fig 6(b).

**Turbulence Kinetic Energy :**

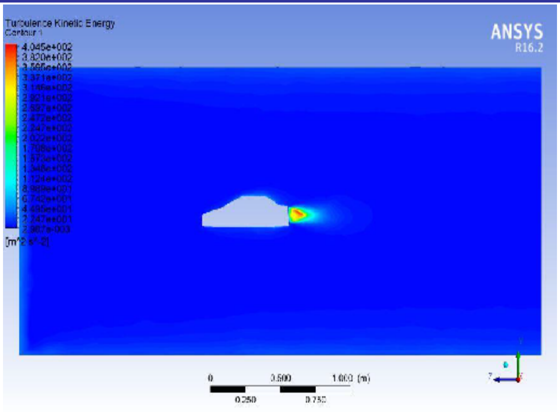



Fig 6(c).

## CONCLUSION:

| Angle | Velocity (m/s) | Turbulence KE($m^2/s^2$) | Drag (N) |
|---|---|---|---|
| 10 | 123.4 | 420 | 117.315 |
| 11 | 122.99 | 404.4 | 128.308 |
| 12 | 123.33 | 404.5 | 103 |

Table 1.

- From the investigation it was observed that the change of diffuser angle (10,11,12,) has a great influence on wake of the underbody flow. The drag force decreasing while increasing diffuser section angle. increasing diffuser angle which is reducing turbulence kinetic energy and drag force values.
- Comparing all these result value angle 12 Degree which give maximum velocity and low turbulence Kinetic energy value which reducing drag force to 103N. This angle give maximum efficiency.

REFERENCES:

[1]. Johan Levin, Rikard Rigdal.Aerodynamic analysis of drag reduction devices on the underbody for SAAB 9-3 by using CFD, Chalmers University of Technology, ISSN 1652-8557, 2011.

[2]. Hu, X., Zhang, R., Ye, J., Yan, X., & Zhao, Z. (2011). Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics. Physics Procedia, 22, 239-245.

[3]. Aljure, D. E., Lehmkuhl, O., Rodriguez, I., & Oliva, A. (2014). Flow and turbulent structures around simplified car models. Computers & Fluids, 96,122-135.

[4]. Huminic, A., Huminic, G., & Soica, A. (2012). Study of aerodynamics for a simplified car model with the underbody shaped as a Venturi nozzle.International Journal of Vehicle Design, 58(1), 15-32.

[5]. Yamazaki, S., Motojima, S., & Koki, Y. (1994). The effect of a moving belt for the ressure on the underbody of a competition vehicle. JSAE Review, 15(2), 171-175.