



# Numerical Analysis of the effect of Diffuser Angles in the Ahmed Body

**Khandakar Ahmed Rhine, Shreyans Jain, Uthra Anuradha Kawdaullage**

*Department of Mechanical Engineering, Delhi Technological University, India*

*†Corresponding Author Email: shreyansjain60201@gmail.com*

## Abstract

The Ahmed Body is a comprehensive model designed to capture and study distinguishing characteristics that are pertinent to bodies in the vehicle industry despite having a relatively simple shape, which then is used to create new car models after it has been verified. As the changes in diffuser angle ( $\beta = 0^\circ$  to  $25^\circ$ ) significantly affects the flow characteristics downstream of the Ahmed Body. Our study is aimed at examining the effects of the change in diffuser angle on incompressible flow over the Ahmed body to determine the optimum angle of the diffuser. The commercial CFD tool ANSYS is used because of its accurate capability in replicating aerodynamic force coefficients. The data then obtained is verified against similar verified studies. The change in Diffuser Angle ( $\beta$ ) against a fixed Slant Angle ( $\alpha$ ) is expected to provide an insight into the use of diffusers in consumer vehicles.

**Keywords:** *Ahmad body; diffuser angle; Computational fluid dynamics (CFD); ANSYS*

## 1.1 Introduction

To achieve better fuel efficiency, low road noise, and lower emissions while driving, the aerodynamics of the vehicle (Sykes & Scibor-Rylski, 1984,) needs to be optimized. This involves achieving better vehicle 'ground effects'. Ground effects of a car are achieved by reducing the air flowing underneath the vehicle, which can be achieved by the addition of ground effect devices; front and rear spoilers, side skirts and diffusers (Marklund et al., 2013,). The air that travels underneath a car when it's moving at high speeds builds up pressure causing a lift effect, which in turn reduces stability. The above mentioned ground effect devices help to create a low pressure area underneath the car; increasing the downforce and improving grip by channeling airflow away from the car's underbody. The air that travels underneath a car when it's moving at high speeds builds up pressure causing a lift effect, which in turn reduces stability. The above mentioned ground effect devices help to create a low pressure area underneath the car; increasing the downforce and improving grip by channeling airflow away from the car's underbody. The diffusers of a vehicle employ the 'Venturi Effect'<sup>1</sup>; the constricted air flow results in accelerated local airflow, which according to Bernoulli's principle reduces pressure improving downforce on the vehicle. The improved airflow around the vehicle resulting from ground effects plays an important role in improving handling, stability, and traction at high speeds.

The objective of the study is to carry out tests on the Ahmeds body aerodynamic behavior under various geometric

---

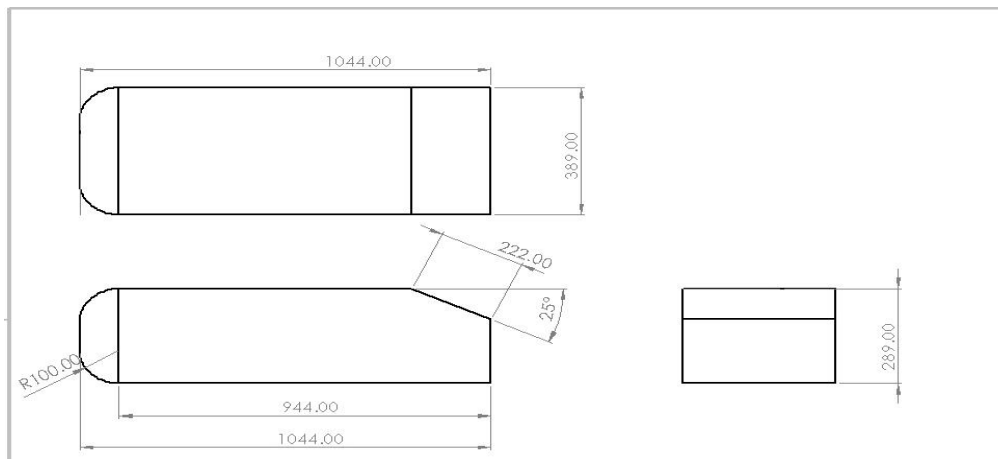
<sup>1</sup> "Venturi effect - Wikipedia." [https://en.wikipedia.org/wiki/Venturi\\_effect](https://en.wikipedia.org/wiki/Venturi_effect). Accessed 20 May. 2023.

parameters using CFD simulation tool (FLUENT, 2023) to see the flow and determine the coefficient of drag at each diffuser angle.

Then, we outline the numerical simulation<sup>2</sup> methodology. Next, we present and compare the results obtained from the simulations. Finally, we summarize our findings and draw conclusions in the last section.

### 1.2 Ahmed body and its dimensions:

The Ahmed body's geometrical shape is depicted in Figure 1. Despite the body's obvious geometric deviation from that of normal automobiles, it incorporates the key aerodynamic features of a vehicle, notably in terms of its aerodynamic performance. It is crucial to consider the rear of the car while designing it because it contributes significantly to the drag profile of the vehicle.



**Fig:1 Dimensions of the Ahmed Body used for simulations**

Therefore, the fastback vehicles are represented by an Ahmed Body with a slant angle of 25 degrees, this model exhibits very potent inward rotating longitudinal vortices at the C-pillars.. These vortices create a field of downwind in the contained area and extend far into the wake.

The aluminum Ahmed body that was utilized for the experiments was constructed in a simulator using the original data depicted in Figure 2. To provide a fair comparison of the diffuser angles, the front portion was kept constant in its dimensions.

### 1.3 Diffuser angle:

Diffusers are designed to utilize the kinetic energy of a flow to create a pressure difference (L, n.d.) and to alter the airflow around the vehicle in the vertical plane so as to minimize the air drag. Furthermore it aims to increase the cross sectional area. The diffuser is a passage between an upswept surface and the ground and is shown the

<sup>2</sup> "Direct numerical simulation - Wikipedia." [https://en.wikipedia.org/wiki/Direct\\_numerical\\_simulation](https://en.wikipedia.org/wiki/Direct_numerical_simulation). Accessed 20 May. 2023.



diagram below.

This passageway is present at the rear of the vehicle. The airflow follows the upswept surface accelerating the flow from the underbody into a high-pressurized flow, therefore increasing velocity flow and lowering lift and thereby minimizing the drag forces.

The simulations were carried out at an air velocity of 40 m/s, keeping the dimensions of the Ahmed body constant excluding the angle of the diffuser. A measure of consistency was taken in that the start point of the diffuser was also kept constant across all the models generated. The slant angle was kept constant at 25 degrees and the diffuser angles were varied by 5 degrees. With each model having a 5 degree increment in diffuser angle.

To enable a direct comparison the velocity streamline figures were produced along with the graphs for cd values in order to directly compare the effects of the change in diffuser angle on the Ahmed body and thereby find the change in what might be the case if such a diffuser was placed in a car.

## **2.1 PRIMARY COMPONENTS OF A CFD SIMULATION SOFTWARE:**

CFD software or tools has three main processes that are carried out to analyze the object from start point to the final simulation.

This process are explained in brief as follows:

- **Pre-Processing:**In this process, the geometry is first defined or created, followed by meshing which breaks the computational domain into small volumes. These control volumes are the areas which then interact with the flowing fluid (in this article, air).
- **The Solver:** During this stage of the CFD simulation, the discretized and solved flow regulating equations will be performed.
- **Post-processing:**The results of the simulation are created at this stage of the CFD simulation, together with other crucial flow characteristics like velocity, density, pressure, and forces that were calculated during the simulation.

## **2.2 Simulation Assumptions:**

The following assumptions will be used throughout the simulation process:

- There cannot be any exchange of thermal energy or heat between the flowing air and the object being meshed.
- Air flow speed from the inlet shall remain constant during the simulations.
- The boundaries of interaction between the body and the walls will be simulated with no slip shear conditions.
- The fluid will be considered as incompressible as the fluid velocity Mach number is less than 0.3.



**Numerical Equations:**

**Primary Equations:** To simulate the incompressible flow, Navier-Stokes equations for n compressible flow<sup>3</sup> will be utilized.

Based on the assumptions, the following equations will be used:

1) Continuity Equation:

$$(1) \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

For an incompressible flow

$$(2) \frac{\partial \rho}{\partial t} = 0, \quad \rho \cdot (\nabla \cdot \mathbf{v}) = 0$$

Thus;

$$(3) \frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} = 0$$

Where,

$$\rho = \rho, \quad \rho \cdot \nabla = 0$$

2) N-S Equations:

a) X-direction component:

$$(4) \frac{\partial(\rho u)}{\partial t} + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) = - \frac{\partial p}{\partial x} + \frac{\partial^2 \tau_{xx}}{\partial x^2} + \frac{\partial^2 \tau_{yy}}{\partial y^2} + \frac{\partial^2 \tau_{zz}}{\partial z^2} + \rho g_x$$

b) Y-direction component

$$(5) \frac{\partial(\rho v)}{\partial t} + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) = - \frac{\partial p}{\partial y} + \frac{\partial^2 \tau_{xx}}{\partial x^2} + \frac{\partial^2 \tau_{yy}}{\partial y^2} + \frac{\partial^2 \tau_{zz}}{\partial z^2} + \rho g_y$$

c) Z-direction component

$$(6) \frac{\partial(\rho w)}{\partial t} + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) = - \frac{\partial p}{\partial z} + \frac{\partial^2 \tau_{xx}}{\partial x^2} + \frac{\partial^2 \tau_{yy}}{\partial y^2} + \frac{\partial^2 \tau_{zz}}{\partial z^2} + \rho g_z$$

*2.3 Conditions at the boundary or interaction walls for the simulation*

<sup>3</sup> "Navier–Stokes equations - Wikipedia."

[https://en.wikipedia.org/wiki/Navier%E2%80%93Stokes\\_equations](https://en.wikipedia.org/wiki/Navier%E2%80%93Stokes_equations). Accessed 20 May. 2023.



The following boundary conditions (T, 2017) shall be maintained during the simulation in ANSYS-Fluent:

- Velocity-inlet: Inlet surface.
- Symmetry: Enclosure surfaces (maintaining no-slip conditions).
- Walls: Road and enclosure faces.
- Pressure-outlet: Outlet surface of the enclosure.

Flow Inlet Parameters (FIUENT, 2017):

$$v=15.13 \times 10^{-6} \text{ m}^2/\text{s}.$$

$$\rho= 1.225 \text{ kg/m}^3.$$

$$V = 40\text{m/s}.$$

$$Re = 768,000 \text{ (based on object's height)}.$$

$$I \leq 0.25\%$$

2.4 Meshing (FIUENT, 2017):

To properly compute the wall shear stress during the simulation, the initial cell's distance from the surface (y) must be accurately estimated. To do this, before beginning the meshing process, Y+ computations should be run to determine the value of (y).

1. Reynolds Number (Re):

$$Re = \frac{V \cdot L}{\nu} = \frac{40 \times 0.288}{15 \times 10^{-6}}$$

$$Re = 768,000$$

Skin friction coefficient(Cf):

$$C_f = [2 \log_{10}(Re) - 0.65]^{-2.3}$$

$$C_f = [2 \log_{10}(768,000) - 0.65]^{-2.3}$$

$$C_f = 3.925 \times 10^{-3}$$

Wall Shear Stress (w):

$$\tau_w = C_f \times \frac{1}{2} \rho V^2$$

$$\tau_w = (3.295 \times 10^{-3}) \times \frac{1}{2} \times 1.225 \times 40^2$$

$$\tau_w = 3.847 \text{ Pa}$$



4) Friction Velocity:

$$\tau_w = \mu \frac{du}{dy}$$

$$u^* = \sqrt{\frac{3.847}{1.225}}$$

$$u^* = 1.772 \text{ m/s}$$

For most near-wall modeling,  $Y^+ \approx 1.0$ . (Salim & Cheah, 2009)

Therefore,

5) Wall Distance (y):

$$y^+ = \frac{y u^*}{\nu}$$

$$y = \frac{1.0 \times (1.789 \times 10^{-5})}{1.225 \times 1.772}$$

$$y = 8.241 \times 10^{-6} \text{ m}$$

herefore, the boundary layer thickness (wall Distance) for this analysis is  $8.241 \times 10^{-6} \text{ m}$

## 2.5 Base value Establishment

### Cd value obtained: 0.33347

The Cd value or drag coefficient (dimensionless quantity) is an important metric. The pressure drag that results from flow separation is a significant factor in a vehicle's drag. With a 25 degrees slant angle, the drag coefficient of the Ahmed body is equal to 0.33347, which is in agreement with the findings of the reference papers used in the project. Thereby the base model is established, based upon the same parameters further experiments will be carried out.

For the project, using the same parameters used for establishing the baseline value, 5 separate models were made. Each of these models had the same dimensions as the Ahmed body with the difference being in the diffuser angle.

The cd values for each, at (constant Reynolds number) was calculated based on 100 iterations as the same as the baseline. The number of nodes, cells and faces for each model subsequently changed.

3.1 Results for Ahmed body with diffuser of 5 degrees:

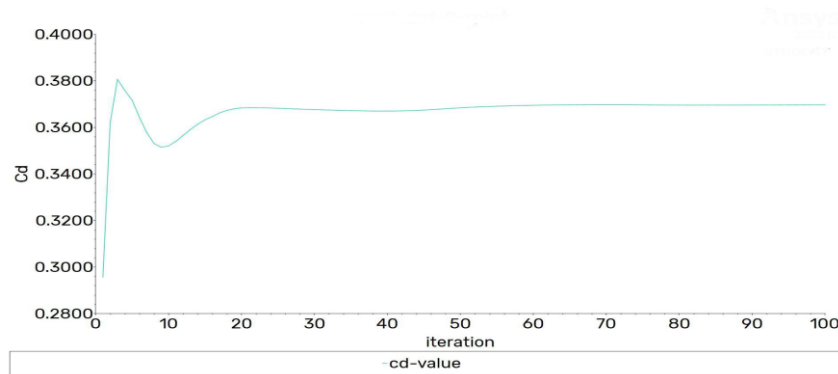


Figure 2: Cd coefficient value with 5 degree diffuser

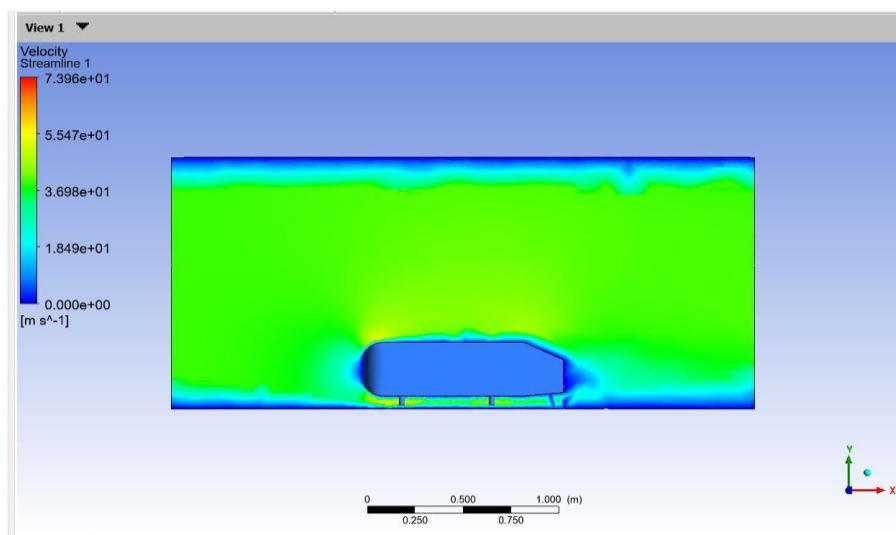


Figure 3: Velocity diagram of ahmed body with 5 degree diffuser

**Cd value obtained:0.3357**

The comparatively marginal change in diffuser angles as seen has had an impact on the airflow over the Ahmed Body demonstrated by the velocity flow diagrams.

3.2Results for Ahmed body with diffuser of 10 degrees:

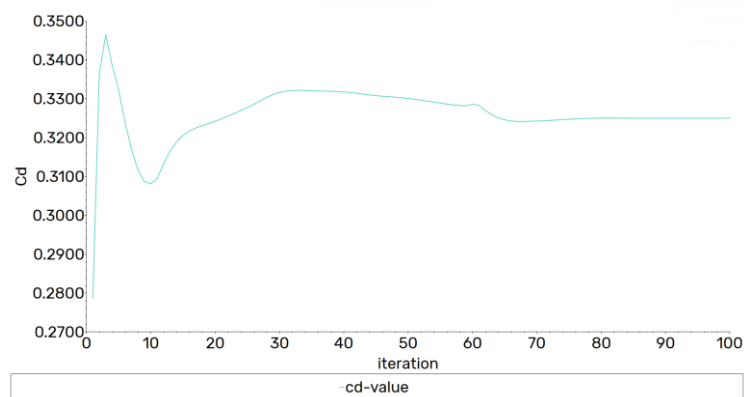


Figure 4: Cd coefficient value with 10 degree diffuser

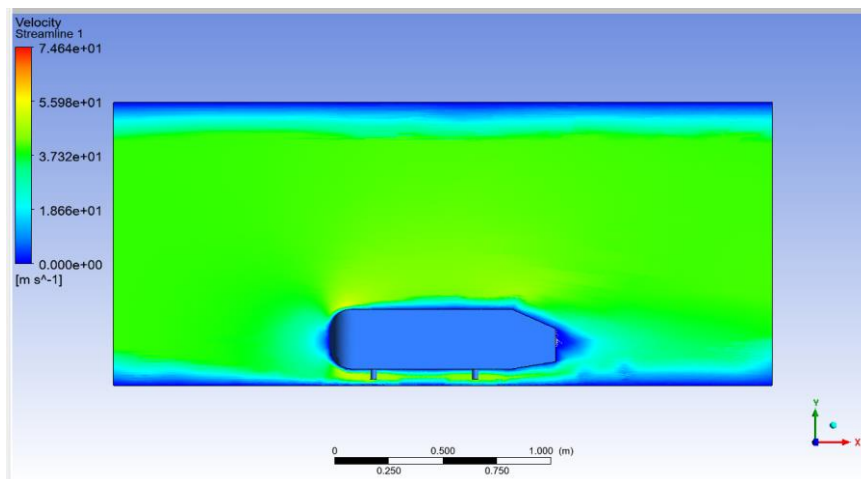


Figure 5: Velocity diagram of Ahmed Body with 10 degree diffuser

Cd value obtained:0.3244

### 3.3 Results of Ahmed body with 15 degree diffuser

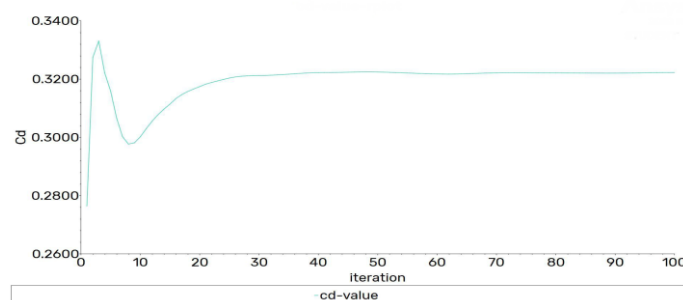


Figure 6: Cd plot for Ahmed Body with 15 degree diffuser



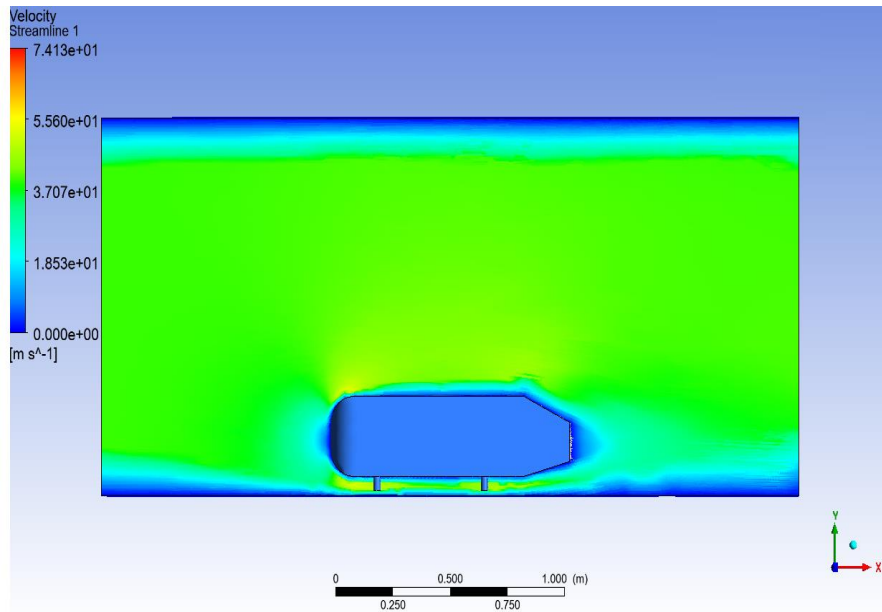


Figure 7: Velocity diagram of ahmed body with 15 degree diffuser

Cd value obtained:0.3207

### 3.5 Results for Ahmed body with diffuser of 20 degrees:

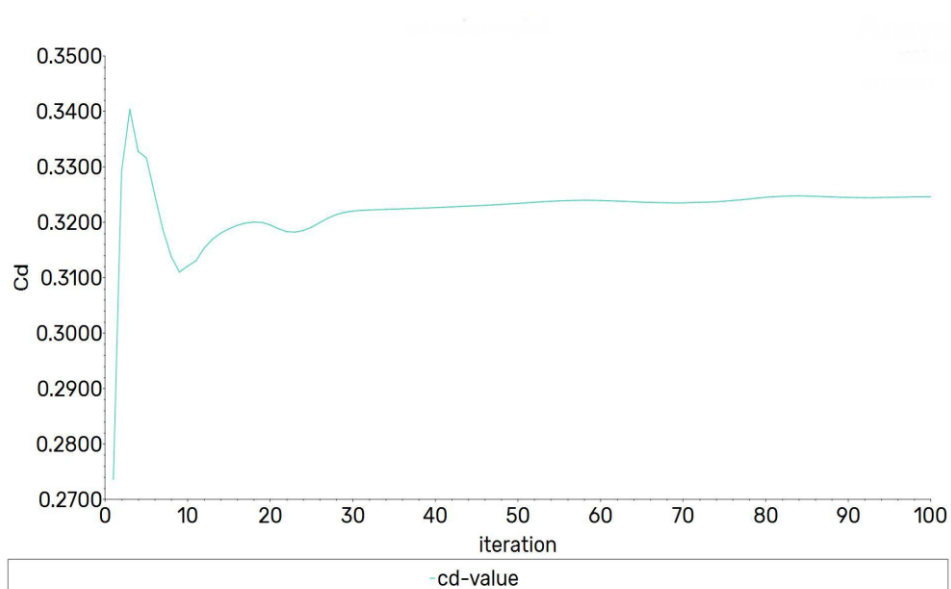


Figure 8: Cd Plot Ahmed Body with 20 degree diffuser

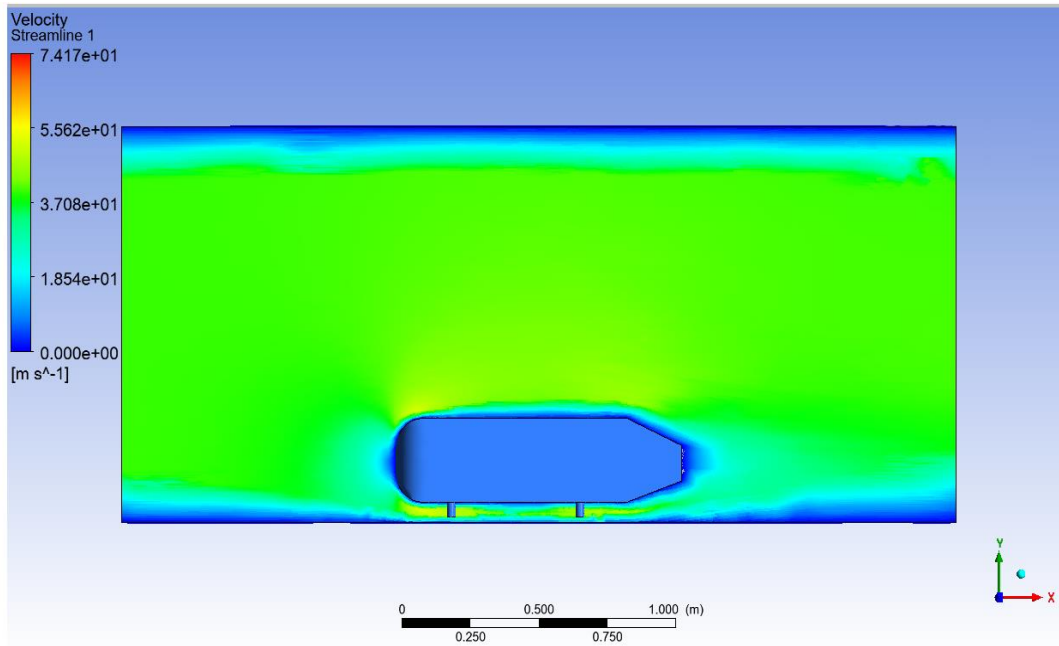


Figure 9: Velocity diagram of ahmed body with 20 degree diffuser

Cd value obtained:0.3112

Results for Ahmed body with diffuser of 25 degrees:

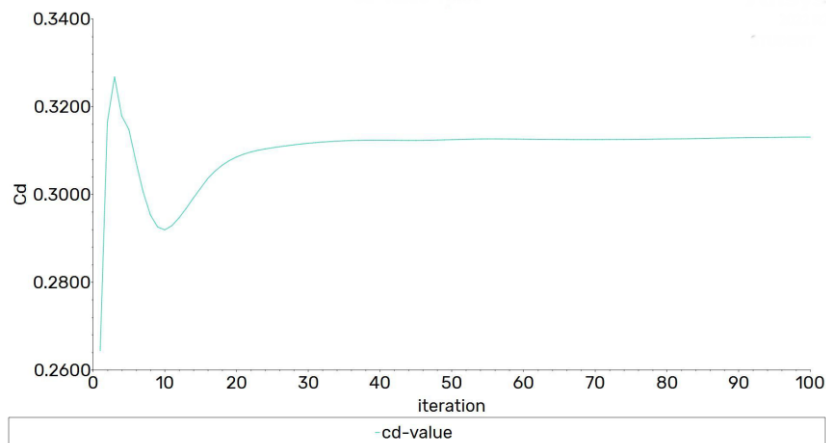
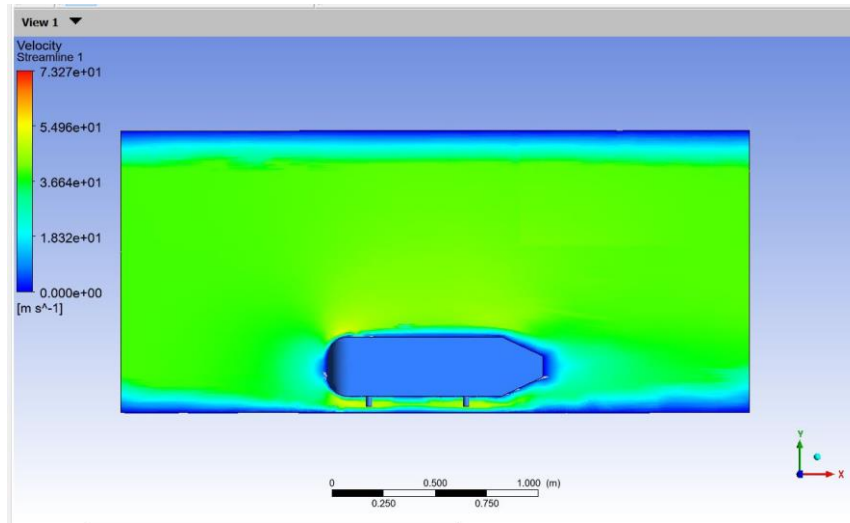


Figure 9: Cd Plot Ahmed Body with 25 degree diffuser



**Figure 11: Velocity diagram of ahmed body with 25 degree diffuser**

**Cd value obtained: 0.3233**

From The results obtained it is evident that the value of cd and hence the drag of the vehicle is positively affected at comparatively lower diffuser angles whilst at large angles it tends to be negatively affected.

This is in line with current practices in automobile manufacturers who put diffuser angles on average between the ranges of 10 to 20 degrees on average.

| Diffuser Angle | Cd coefficient Value Obtained |
|----------------|-------------------------------|
| 0              | 0.3347                        |
| 5              | 0.3357                        |
| 10             | 0.3244                        |
| 15             | 0.3207                        |
| 20             | 0.3112                        |
| 25             | 0.3233                        |

**Table 1: The obtained Cd values**

**CONCLUSIONS**

The drag coefficient, which measures an object's resistance to airflow, is a crucial total value (dimensionless quantity). The flow separation which occurs at the rear end of the body is a major component of air drag caused



in a vehicle. The objective of a diffuser is to create a surface that assists in the air flow to change direction as it is the tendency of a flowing volume of air to follow a curved surface. Thereby this action reduces the flow separation at the rear end thereby helping to reduce air drag making the vehicle more efficient. As the results of the simulations show the diffuser has a considerable impact on the flow field of the vehicle and

thereby it affects vehicle drag and lift subsequently. Based on the data gathered, it was determined that increasing the diffuser angle causes a reduction in the drag force by reducing the size of the low-pressure recirculating zone behind the body. Weak flow separations are caused by diffuser angles larger than 10. Higher diffuser angles (like 20) will completely separate the flow, causing drag to rise. The emergence of longitudinal vortices (near the side margins of the diffuser) will also play a role, negatively affecting the aerodynamic properties. As a result, the drag forces will rise. The diffuser outlet pressure was also discovered to be constant. The diffuser's inlet pressure will drop when the diffuser angle is increased. As a result, there will be more air being drawn into the diffuser, which will strengthen the zone of low pressure and result in greater negative lift and downforce. The flow separation that occurs at larger diffuser angles, however, has a detrimental impact on the aerodynamics, increases drag, and less downforce

Although it is evident that having a diffuser improves overall aero performance and efficiency of the Ahmed body, however as the Ahmed body is only a standardized representation of a vehicle it cannot be used as direct evidence that all cars will have the same result at same diffuser angles. Therefore the concluded data and results must be taken into context when making design choices based on the results of these project

## References

1. FIUENT, A. (2017, April 19). *Modeling Turbulent Flows Modeling Turbulent Flows*. University of Southampton. Retrieved May 20, 2023, from [http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence\\_Notes\\_Fluent-v6.3.06.pdf](http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf)
2. FLUENT, A. (2023, January 1). *Ansys Fluent | Fluid Simulation Software*. Ansys. Retrieved May 20, 2023, from <https://www.ansys.com/products/fluids/ansys-fluent>
3. L, D. (n.d.). *CFD-based Optimization For Automotive Aerodynamics*. [https://link.springer.com/chapter/10.1007/978-3-540-72153-6\\_7](https://link.springer.com/chapter/10.1007/978-3-540-72153-6_7). Dumas, L., 2011. *CFD-based Optimization for Automotive Aerodynamics*. Paris: Université Pierre et Marie Curie.
4. Marklund, J., Lofdahl, L., Danielsson, H., & Olsson, G. (2013). Performance of an Automotive Body Diffuser Applied to a Sedan and a wagon vehicle. *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(1):293-307, 2013, 6(1), 15. <https://doi.org/10.4271/2013-01-0952>.
5. Sykes, D. M., & Scibor-Rylski, A. J. (1984). *Road Vehicle Aerodynamics*. Pentech Press.
6. T, B. (2017, April 18). *Boundary Layer*. Boundary Layer. Retrieved May 20, 2023, from <https://www.grc.nasa.gov/WWW/BGH/boundlay.html>