Computational study on an Ahmed Body equipped with simplified underbody diffuser

Filipe F. Buscariolo, Gustavo R. S. Assi and Spencer J. Sherwin

Abstract The Ahmed body is one of the most studied 3D bluff bodies used for automotive research and was first proposed by [1]. The variation of the slant angle of the rear upper surface on this body generates different flow behaviours, similar to a standard road vehicles. In this study we extend the geometrical variation to evaluate the influence of a rear underbody diffuser which are commonly applied in high performance and race cars to improve downforce. We perform parametric studies on the rear diffuser angle of two baseline configurations of the Ahmed body: the first with a 0° upper slant angle and the second with a 25° slant angle. We employ a high-fidelity CFD simulation based on the Spectral/hp element discretisation that combines classical mesh refinement with polynomial expansions in order to achieve both geometrical refinement and better accuracy. The diffuser length was fixed to the same length of 222 mm similar to the top slant angle that have previously been studies. The diffuser angle was changed from 0° to 50° in increments of 10° with an additional case considering the angle of 5°. For the case of an 0° slant angle on the upper surface the peak values for drag and negative lift (downforce) coefficient were achieved with a 30° diffuser angle, where the flow is fully attached with two streamwise vortical structures, analogous to results obtained from [1] but with the body flipped upside down. For diffuser angles above 30°, flow is fully separated from the diffuser. The Ahmed body with 25° slant angle and a diffuser achieves a peak value for downforce at a 20° diffuser angle, where the flow on the diffuser has two streamwise vortices combined with some flow separation. The peak drag value for this case is at 30° diffuser angle, where the flow becomes fully separated.
1 Introduction

Among the standard automotive bluff bodies in literature, the most studied one is the Ahmed body, firstly proposed by Ahmed [1]. It is based on the geometry designed by Morel [13], with the main dimensions highlighted in Figure 1. The proposed geometry of the Ahmed body aims to reproduce the main features of road vehicles, such as the frontal stagnation, ground effect and well-defined separation points.

![Ahmed body schematic drawing.](image)

The upper slant length of 222 mm is fixed, independent of its inclination angle $\varphi$.

The most emblematic characteristic of the Ahmed Body is a angled upper back section with fixed length, here referred as slant, on the upper rear portion, allowing the simulation of different automotive body styles. According to [8], it has been shown that the flow over the slanted surface back section is dependent on specific inclination angle. Two critical angles, at $12.5^\circ$ and $30^\circ$, have been observed, in which the flow structure changes significantly, and where a change of curvature of the drag coefficient is also evident. For angles below $12.5^\circ$, the airflow over the slant remains fully attached before separating from the model when it reaches the rear of the body. The flow from the angled section and the side walls produces a pair of counter rotating vortices, which then persist downstream.

For angles between $12.5^\circ$ and $30^\circ$, the flow over the slant becomes highly complex. Two counter-rotating lateral vortices are shed from the sides of the angled section with increased size, which affects the flow over the whole back end, specially the previously existing three-dimensional wake. These vortices are also responsible for
maintaining attached flow over the rest of the angled surface up to a slant angle of $30^\circ$, and it has been shown that they are extended up to half of the length of the model beyond the trailing edge, as discussed in [18] and [11]. Close to the second critical angle, a separation bubble is also formed over the inclined slant. The flow separates from the body, but re-attaches before reaching the vertical back section.

Above a $30^\circ$ slant angle, the flow over the slant is fully separated. However, there is a weak tendency of the flow to turn around the side of the model, as a result of the relative separation positions of the flow over model top and those over the slant side edges. When the flow is in this state, a near constant pressure is found across that region. To characterize all three flow configurations here discussed, representative slant angles are commonly used in literature with $0^\circ$ (or squared-backed), $25^\circ$ and $35^\circ$.

The first experimental study on Ahmed Body [1] was with static floor conditions, considering a Reynolds number $Re = 4.29 \times 10^6$ based on its full length. In this study, results for the drag coefficient were obtained for different slant angles, ranging from $0^\circ$ to $60^\circ$, in increments of $10^\circ$ with an additional measurement at $12.5^\circ$. Due to limitations on the wind tunnel setup, only drag force measurements and a few flow visualization tests were performed. In order to better understand the flow phenomena and turbulence structures around the model, a complementary study was performed by [11] using laser Doppler anemometry (LDA), hot-wire anemometry (HWA) and static pressure measurements.

Aiming to reproduce realistic road conditions and understand the phenomena associated to flow fields close to the ground, the authors of [18] performed an Ahmed Body wind tunnel test using moving ground and acquired both the aerodynamic forces and the flow characteristics by employing time-averaged LDA. The flow conditions were also slightly different from those used on Ahmed’s first test, since it had a Reynolds number of $Re = 1.7 \times 10^6$. Nevertheless, similar flow behaviour were observed on the slant, despite the quantitative results being slightly different. One of the most interesting features found in this flow visualization results is the lower vortex system, a pair of vortices that appears close to the ground interface, which were absent in the fixed-ground studies. According to [18], this could be attributed to the interference caused by the four studs used to support the model on the floor. Figure 2 illustrates this phenomenon on an Ahmed Body with a squared-back.

An important development in automotive industry directly associated with the flow near the ground is the introduction of underbody diffusers, initially for high performance race vehicles with relatively low ground clearance. By providing a smoother transition from the underbody flow to the base of the car body, the strength of the rear wake can be reduced, contributing to drag reduction. In addition, it was found that at slightly inclined angles, the underbody diffuser also increases the downforce generated, assisting the acceleration and handling.

To explore detailed features of the near-ground vortices, and to examine the potential benefits of implementing underbody diffusers, we propose a series of computational studies considering same simulation conditions as the experiment from [18], with moving ground. The Ahmed bodies used in the study are the squared-back and the slant angle of $25^\circ$, representing respectively estate car/station-wagons
(attached flow) and performance cars (vortex generation with flow detachment). The length of the underbody diffuser is set to be the same as the classic Ahmed body slant length, with angles ranging from $10^\circ$ to $50^\circ$, in increments of $10^\circ$. An additional case also considers a diffuser angle of $5^\circ$, a setting commonly found in racing vehicles.

CFD has become an underpinning technology for most automotive companies to reduce development times and costs. Since the Ahmed Body is a widely studied bluff body, it has become a test case to validate new CFD codes, specially for applications in the automotive industry. Lower vortices observed by [18] were not present in CFD simulation studies with fixing studs modelled. They were first observed by [10], where an Ahmed Body with slant angle of $25^\circ$ was simulated without the fixing studs. Nevertheless, the location where these vortices are generated and possible interactions with underbody components were not highlighted.

We utilise a high fidelity spectral/hp element method simulation using under-resolved direct numerical simulation (uDNS) also known as implicit large eddy simulation (iLES). The spectral/hp elemental method combines the advantages of higher accuracy and rapid convergence from the spectral (p) methods, while maintaining the flexibility of the classical finite element (h) complex meshes, allowing unsteady vortical flows around geometries to be effectively captured.

Most computational studies on the Ahmed Body employ simplified Reynolds Averaged Navier-Stokes (RANS) solution. This approach is very reliable for simple stable flow problems, however it is not suitable to correctly predict unstable phe-
nomina around complex geometries. In the study by [10], for the first time for an Ahmed Body, a LES methodology was used yielding solutions of higher flow details, especially for the critical slant angle of $25^\circ$. A major limitation of running LES or detached eddy simulation (DES) for this kind of geometry is the requirement of high mesh resolution, with considerably higher simulation cost and time.

The latest achievements in the high-fidelity turbulence models around an Ahmed body are found mainly for the slant angle of $25^\circ$ and are summarized in the compilation work of [17] in which a comparative analysis of recent simulations, conducted in the framework of a French-German collaboration on LES of Complex Flows at Reynolds number of 768,000. The study offers a comparison between results obtained with different eddy-resolving modelling approaches: three classical h-type method (LES with Smagorinsky model and wall function (LES-NWM), Wall-resolving LES with dynamic Smagorinsky model (LES-NWR), and DES with shear stress tensor (DES-SST)) and one spectral element method (implicit LES with spectral vanishing viscosity (iLES-SVV)). The iLES-SVV simulation in [17] work was conducted in various two-dimensional planes along the span-wise direction, and subsequently constructed into three-dimensional flow fields (commonly known as 2.5D simulation). A new Implicit Delayed DES (IDDES) methodology is proposed by [7] to solve the flow around the Ahmed body. The study presents a comparison between quantitative and qualitative results obtained with different methodologies previously presented with this newly proposed methodology. The IDDES case is the one that most closely correlates the flow behaviour and structures with experimental reference. However, results of the aerodynamic quantities are different, such as the drag coefficient with approximately 27% difference from same experimental reference.

2 Ahmed body equipped with rear underbody diffuser

Bluff bodies equipped with rear underbody diffusers are being studied by several researchers, especially from the automotive industry, to maximise the performance of the vehicle. The study of [4] identified three important characteristics on a body underbody diffuser. The first is a diffuser pumping effect, which occurs once the outlet of the diffuser is set as the base pressure of the body, as identified by [9]. The diffuser recovers pressure along its length, considering continuity and applying an inviscid, steady argument of constant total pressure using Bernoulli’s equation implies that the diffuser inlet pressure should be reduced, causing a suction effect. The second characteristic is the interaction with the road, in which as the ground clearance between the floor and the underbody becomes smaller, flow velocity in that region increases and pressure drops, following the same continuity and Bernoulli’s equation. The third characteristic is the angled upsweep, which generates vortices on the diffuser up to a certain critical angle, creating an upwash of the flow, aiding flow attachment and increasing downforce.

Complementing the work of [4], [16] investigated a new bluff body equipped with a diffuser which extended over 41% of the body length and with inclination angle
of 17° and endplates in different ground heights. The result was the identification of four distinct regions of diffuser performance, all related to the model ground height. The first region from non-dimensional ride height $h/H$, where $h$ is the distance from the body to the ground and $H$ is the total height of the body, is defined from 0.76 to 0.38 and is defined as downforce enhancement, region where the flow on the diffuser is symmetric with some separation on the diffuser inlet. The second region, referred as maximum downforce, from $h/H$ 0.38 to 0.22, with similar flow behaviour as the first region, except for the formation of a separation bubble at the center of the diffuser. The third and fourth regions are referred both as the downforce reduction from $h/H$ 0.22 and low downforce region from $h/H$ 0.16. The third region is characterized by a sudden drop of downforce performance and the fourth region shows that further ground height reduction causes the downforce to be reduced and this fact is explained by an asymmetric and separated flow behaviour at the diffuser inlet.

With substantial literature on the diffusers, such as the works from [4] and [16], we notice that each study has employed different bluff body geometries, that have not been previously studied without a diffuser. The work of [8] was the first to propose a computational study of diffusers on an Ahmed body with slant angle of 35° with similar conditions of [18] study. This body style has a quasi-2D behaviour and a combination of five diffuser lengths with eight different angles were evaluated to predict drag and lift coefficients. The flow physics was not fully evaluated and no conclusions were found mainly due to the nature of the averaged flow solution employed on the study.

By evaluating the Ahmed Body equipped with a rear underbody diffuser without endplates, we also offer an interesting and simple test case, especially for the squared-back case, as it can be evaluated using a regular Ahmed body which has been flipped upside-down.

3 uDNS/iLES simulations using spectral/hp element method

For both Ahmed body styles with diffuser, we performed implicit LES simulations based on a spectral/hp element approach. Classical h-type method is based on dividing the domain into non-overlapping elements of the same type, offering geometric flexibility, a key factor for many complex industrial cases. To improve the accuracy of the solution, the mesh characteristic length (h) is reduced in order to capture smaller flow features, generating a finer mesh, with larger element density of the same type. The p-type method focus on improving results by increasing the degree of the polynomial expansion used to approximate the solution on each element on a fixed mesh. The spectral/hp element method used in this work combine both spatial approximations and the flow solution is obtained by using the incompressible Navier-Stokes solver with a velocity correction scheme as proposed by [6]. The elliptic operators were discretised using a classical continuous Galerkin (CG) formulation and all this formulation is encapsulated in the open source package Nektar++ [3].
In this work we adopt an equivalent of the Taylor Hood approximation, approxi-
mating the velocity by continuous piecewise quadratic functions and the pressure by 
continuous piecewise linear functions. Therefore we use one higher polynomial order 
for velocity than the pressure. The polynomial order for velocity is also referred to 
as the simulation expansion order in this study.

For simulations with higher Reynolds numbers ($10^5$ and above), such as the cases 
presented here, the flow is typically only marginally resolved, which means that 
the ratio of subgrid scale (SGS) and resolved dissipation is relatively small. Such a 
marginal resolution can lead to numerical instabilities related to wave interaction and 
wave trapping. To reach a stable solution, we employ both dealiasing and spectral 
vanish viscosity (SVV) stabilization techniques.

The aliasing errors related to the Navier-Stokes equations appears when handling 
its quadratic non-linearity term by using the Gauss integration orders $Q$ similar to the 
solution polynomial order $P$. This is usually present in simulations considering under-
resolved turbulence, such the iLES, which leads to a significant error increment in 
high-frequency modes of the solution and typically cause the simulations to diverge. 
To avoid the aliasing errors, we employed a quadrature order consistent with the 
polynomial order and non-linearities of the equation. In areas of non-linear geometry 
deformation we also have to be mindful of geometric aliasing (aliasing arising from 
geometric mapping). We refer the interested reader to [12] for more details.

For the SVV operator in this study, we run the simulation using a novel CG-
SVV scheme with DGKernel as proposed by [14]. The fundamental idea of this 
implementation is based on the fixing of the Péclet number, which can be understood 
as a numerical Reynolds number based on local velocity and mesh spacing. This is 
achieved by making the viscosity coefficient of the SVV operator proportional to 
both a representative velocity and a local mesh spacing. Once the Péclet number 
is the same for the domain, the authors in [14] proposed a SVV kernel operator 
for CG methods that mimics the properties of DG-based discretisation where there 
is natural damping of high frequency and reflected waves. In this approach the 
dissipation curves arising from spatial eigenanalysis of CG of order $p$ are matched 
to those of DG with order $p - 2$. Matching both curves offers benefits such as the 
numerical stability of simulations at very high Reynolds number.

4 Simulation methodology

In this simulation study, we use a coordinate system with $X$ as the streamwise 
direction, $Y$ as the vertical direction and $Z$ as the spanwise direction. Diffuser length 
$D_L$ is set to be at a fixed value, which is the same as the upper slant length $S_L$ of 222 
mm, regardless of the inclination angle changes.

The Ahmed body model is positioned with its back end on the coordinate $X = 0$ 
and at a distance $h$ of 50 mm from the ground ($Y = 0$). The wind tunnel test section 
size is defined as 1660 mm $\times$ 2740 mm, with the inlet positioned at $X = -2L$ and 
outlet at $X = 2L$ with a total $X$ length of $4L$. 

The Reynolds number for all simulated cases is $Re = 1.7 \times 10^6$, based on the Ahmed body total length $L$ of 1044 mm. With this Reynolds number value and by imposing moving ground condition, we aim to reproduce similar conditions employed by [18].

The high-order meshes for all cases presented in this work were generated by the mesh generator module of Nektar++: NekMesh ([19]). The pipeline to create a high-order mesh starts by designing the geometries for the computational simulations on a Computer Aided Design (CAD) software, exported in STEP format. Subsequently, we generate a linear mesh using a classical h-type method. We considered an average mesh element size of approximately $0.0075L$ on the surface of the body and on ground immediately below the Ahmed body. Linear mesh generation on NekMesh also incorporates an optimisation step to avoid irregular and low quality elements once the surface mesh is projected into its 3D form using the CAD surface.

Once the linear mesh was generated, the next step is to convert it into a high-order mesh which is geometry conforming. The generation of the high-order mesh requires the addition of extra points to represent the polynomial discretization (with order $P_M$), which are added along the curved edges, CAD surface geometry and in the interior of the domain. For this study, we selected polynomial $P_M$ of 6th order. The processes then follows by the generation of a macro boundary layer on user-defined wall surfaces together with volumetric high-order mesh on rest the domain. The final operation on the mesh generation is the boundary layer split, by using the isoparametric approach as proposed by the authors of [15] considering previously inputted parameters.

Mesh parameters inputs first generates a macro boundary layer with total size of $0.022L$, which is further divided into 10 layers with a growth rate of 1.6. The volumetric mesh also incorporates two refinement zones applied over the Ahmed body length as illustrated with Figure 3 following a similar strategy to the mesh setup of [2]. The first refinement zone is around the Ahmed Body with a mesh size of $0.035L$, starting from $X = -1.3L$ with a total length of $1.6L$ and a height of $0.55L$. The second refinement box, defined as the wake refinement region, has the mesh element size defined as $0.020L$. The total length and the height is the same as the first refinement zone, but with the starting location offset to $X = -0.3L$, covering the slant and extends further up to $X = 1.3L$. The refinement zones are aimed to help capture the flow phenomena over the slant and diffuser areas as well as the flow features on the far field, based on a h refinement approach of the spectral/hp element method.

With the use of p-refinement on the spectral/hp element method, the volumetric mesh elements are approximated by polynomials of 6th order, which increases the resolution of the solution by converting a mesh with 250,000 (h-type equivalent) elements into 19.8 million degrees of freedom (DOF) per variable.

The boundary conditions for the computational study were as follows and shown on Figure 4:

- Ahmed bodies with diffuser are set as wall with no-slip condition;
- A half model of the geometry is used;
- Symmetry condition imposed at $Z = 0$;
Fig. 3: Mesh refinement regions on both Ahmed bodies squared back (up) and with slant angle of 25°. Refinement region highlighted in yellow is defined as the Ahmed Body refinement and region highlighted in black is defined as Wake Refinement.

- Uniform velocity profile at the inlet;
- High order outflow condition at the outlet (as proposed by [5]);
- A moving ground condition on the floor with speed $U$ in the $X$ direction, as used by [18].

Fig. 4: Boundary condition setup.
Simulations were performed for 7 convective time units (CTU), which can be understand as the free stream flow has been advected for a total length of $7L$.

5 Results

The key findings of the study are presented as follows. We initially present a comparison of drag and lift coefficients for both Ahmed body styles when the diffuser angles changes. Both quantities are averaged from the 5th to the 7th convective length where we assure to have a fully converged physical solution. Flow structures comparison for the planes $X/L = 0$ and $X/L = 0.096$ and wall shear stress lines are both averaged from the same period and presented in order to complement the findings.

5.1 Drag coefficient results

Drag coefficient results with both Ahmed body slant angles are presented in Figure 5. Ahmed body squared back results indicate that the drag coefficient initially rises as the diffuser angle increases, reaching the maximum value at the diffuser angle of $30^\circ$. The rising drag trend suddenly breaks for diffuser angles higher than $30^\circ$, where results for further angles are similar to the diffuser with $20^\circ$ inclination. Such behaviour indicates similar trends as earlier verified in studies on Ahmed body slant angle variations of [1], [11] and [18]. The drag breaking point for the diffuser angle is similar to slant angle which is an indication of flow structure change on the diffuser region and is further discussed below. We conclude that any diffuser added to the Ahmed body squared back has a penalty in terms of drag performance.

The drag coefficient results for Ahmed body with $25^\circ$ slant angle show different trend from the squared back case. Applying the diffuser with angles of $5^\circ$, $10^\circ$ and $20^\circ$ angle leads to drag reduction with the optimum angle at $10^\circ$. For the other diffuser angles, the drag coefficient recovers to a similar value of the Ahmed body with $25^\circ$ slant angle without a diffuser. We also notice that the drag performance enhancement region might be related to flow behaviour change on the diffuser region, which is further presented. Diffuser application has no negative impact on drag coefficient for this Ahmed body style.

5.2 Lift coefficient results

Lift coefficient values for both Ahmed body cases at different diffuser angles are presented in Figure 6. When analysing the lift coefficient results for the Ahmed body squared back, downforce enhancement is obtained as the diffuser angle increases from $0^\circ$, reaching maximum downforce value at $30^\circ$. The downforce trend breaking
phenomenon is observed for diffuser angles higher than 30°, similar as observed for the drag coefficient. For diffuser angles higher than 30°, downforce force values are similar to the diffuser of 5°, indicating saturation of downforce enhancement with this diffuser geometry.

Ahmed body with 25° slant inclination lift coefficient results shows similar trend of previous squared-back results. We’ve noticed the positive lift on the baseline case without diffuser, which might compromise gripping on performance and race cars. By implementing the diffuser, downforce performance starts to increase, where the first proposed diffuser angle of 5° take the lift coefficient to equilibrium. Downforce increment is noticed until the diffuser angle of 20°, where the maximum performance is reached. The diffuser loses its performance for diffuser angle of 30° and above, where the first has performance similar to the 5° whereas the 40° and 50° diffusers have similar performance as the baseline.

Fig. 5: Drag coefficient comparison for Ahmed Body squared-back (blue line) and 25° slant inclination (orange line) considering standard configuration and evaluated diffuser angles: 5°, 10°, 20°, 30°, 40° and 50°.

Ahmed body with 25° slant inclination lift coefficient results shows similar trend of previous squared-back results. We’ve noticed the positive lift on the baseline case without diffuser, which might compromise gripping on performance and race cars. By implementing the diffuser, downforce performance starts to increase, where the first proposed diffuser angle of 5° take the lift coefficient to equilibrium. Downforce increment is noticed until the diffuser angle of 20°, where the maximum performance is reached. The diffuser loses its performance for diffuser angle of 30° and above, where the first has performance similar to the 5° whereas the 40° and 50° diffusers have similar performance as the baseline.

Fig. 6: Lift coefficient comparison for Ahmed Body squared-back (blue line) and 25° slant inclination (orange line) considering standard configuration and evaluated diffuser angles: 5°, 10°, 20°, 30°, 40° and 50°.
5.3 Flow Features Analysis

Comparative results for the flow structures found on the Ahmed body equipped with a diffuser are now presented and discussed. The comparisons are presented at two planes, one at $X/L = 0$, where the back end of the Ahmed body is placed and one downstream at $X/L = 0.096$, in order to evaluate how the flow structures develop as they separate from the body. Contours of Q-criterion, $U$ (streamwise) and $V$ (vertical) velocities are provided for the inspection planes, aiming to identify flow structures and define its interactions with the rear wake. We also provide a flow topology comparison on the diffuser surface by plotting averaged wall shear stress lines from the 5th to the 7th convective length, same period as the flow quantities and aerodynamic forces. Results for the Ahmed body squared-back are firstly presented, followed by the Ahmed body with 25° slant angle with all plane views observed from downstream. We define the simulation cases by the abbreviation of diffuser angle, DA, followed by the inclination angle so diffuser angle of 30° is defined as DA30.

5.3.1 Ahmed body squared back with diffuser

The first consideration of Figure 7(a) shows a vortex arising from side of the diffuser up to the angle of 30°. The vortex intensity and core size increase as the diffuser angle becomes more inclined. A wake structure can also be noticed for the DA30 case, indicating that there is separated flow on the diffuser. Similar flow behaviour is observed on the classical Ahmed body slant variation experiment for slant angles ranging from 12.5° to 30°. For diffuser angles above 30° only a few structures are observed, such as weak vortices on the lower part part of the body, near the floor. This is an indicative of fully separated flow, however the Q-Criterion image alone is not conclusive.

Moving to plane $X/L = 0.096$, shown Figure 7(b) the main difference noticed is on DA30 case where the diffuser vortex has merged with the rear wake. We observed that the diffuser vortex in DA5, DA10 and DA20 cases are weaker and shifting both upwards and in the spanwise direction towards the centre plane.

Flow separation is observed for DA40 and DA50 case on plane $X/L = 0$. The large negative velocity area shown in Figure 8(a) indicates that the flow is already separated at the outlet of the diffuser. The slight higher drag for DA50 is explained by a more intense contour of negative $U$ velocity when comparing to DA40. The negative velocity area on DA30 at mid-span, shows a combination of flow separation with vortex generation. With both flow features, the DA30 case can characterized as a highly energetic flow.

Flow structures evolution is presented in Figure 8(b) reaching the plane $X/L = 0.096$. Low velocity zones are observed on the rear vertical portion of the body, with a different contour position for DA30. The DA30 shifts the base wake upward as it moves to a inner spanwise direction. The diffuser vortex intensity is probably the main cause of this translation and $V$ velocity results are next presented to complement the explanation the this phenomenon.
from normalized vertical velocity $V$ contours in Figure 9 we extract the diffuser vortex rotation direction. The diffuser vortex rotates anti-clockwise, as the most inner spanwise vortex component has positive vertical velocity and the outer negative. Vertical velocity contour for DA30 on plane $X/L = 0.096$ explain the wake moving upwards on the spanwise direction from Figure 8(b). A strong positive vertical velocity zone is observed at the mid-span behind both the diffuser and back of the Ahmed body.

Averaged wall shear stress lines for the diffuser cases evaluated are next presented. The bottom view of the diffuser, with flow direction coming from the top are shown
Analysing results in Figure 10, we observe different flow behaviours on the diffuser surface, detailed as follows. The DA5 and DA10 cases have the diffuser vortex influencing the diffuser surface up to the mid-span. There is a clearly defined separation area at the diffuser inlet, responsible for the pumping effect observed by [4]. The flow keeps attached on the diffuser surface until reaching the diffuser outlet. A combination of both separated flow on the diffuser surface and diffuser vortex is observed for the DA30 case. The diffuser vortex size increases, reaching almost
mid-span distance at the diffuser outlet, while fully separated flow is also observed on the diffuser surface.

The pattern evident in DA20 is a combination of the DA30 and DA10 diffuser flow regimes. There is indication of the diffuser vortex, with defined separation area at the diffuser inlet followed by a small recirculation bubble due to flow separation. The flow then reattaches and follow this pattern until reaching the diffuser outlet. Considering the more extreme diffuser angles (DA40 and DA50), had no previous references, however the low performance is expected. Results show a chaotic behaviour on their
Fig. 10: Wall shear stress lines (black) on the diffuser surface for the Ahmed body squared-back considering the proposed diffuser angles: $5^\circ$ (DA5), $10^\circ$ (DA10), $20^\circ$ (DA20), $30^\circ$ (DA30), $40^\circ$ (DA40) and $50^\circ$ (DA50), bottom view, incoming flow direction from top.

surfaces due to the separated flow and a recirculation zone is observed at the diffuser outlet.

5.3.2 Ahmed body $25^\circ$ slant inclination with diffuser

We now analyse flow structures found for the Ahmed body with slant angle of $25^\circ$ equipped with underbody diffuser. On the upper part of the body, the slant vortex is clearly defined in all diffuser angles evaluated. As presented for previous case, contours of Q-Criterion for planes $X/L = 0$ and $X/L = 0.096$ are shown in Figure 11. The diffuser vortex appears on DA5, DA10 and DA20 together with an additional small intensity vortex on the plane $X/L = 0$. The diffuser vortex has similar intensity and size as the slant vortex in the DA20 case. The other three diffuser cases, DA30, DA40 and DA50 indicate no evidence of the diffuser vortex but only the same lower side vortex, originated on the frontal part of the Ahmed body. Further downstream on
Fig. 11: Contours of Q-Criterion ($Q_{\text{Crit}} = 100$) for the Ahmed body with slant angle of 25° considering diffuser angle of 5° (DA5), 10° (DA10), 20° (DA20), 30° (DA30), 40° (DA40) and 50° (DA50) for planes $X/L = 0$ and $X/L = 0.096$.

plane $X/L = 0.096$, the vortical system have moved inward in the spanwise direction on the first three cases. The last three cases indicate that the lower side vortex gets weaker as it moved downstream.

Contours of normalized $U$ for both $X/L = 0$ and $X/L = 0.096$ planes are presented in Figure 12. On plane $X/L = 0$, we observe that the velocity contour on the upper slant changes as the diffuser angle becomes more inclined until the case DA20. The three other cases from DA30 to DA50 however, have similar velocity profiles. We conclude that the diffuser influences the flow over the upper slant whenever we have evidences of the diffuser vortex. From observation of the first
Ahmed body case, we have indication of attached flow for diffuser angle up to 20°, whereas for higher angles, a significant wake contribution can be seen from the fully separated flow from the diffuser.

Moving downstream, normalized $U$ velocity on plane $X/L = 0.096$ shows the evolution of the turbulent wake and vortices. Base pressure turbulent wake with the slant and diffuser vortices are the main structures seen on cases DA5 and DA10. The $U$ velocity contours on this plane for the DA20 case shows a very small negative velocity zone, indicating an energetic wake, together with the slant and diffuser vortices moving downstream. On DA30 case, wake profile and vortical system is similar to DA40 and DA50, except by the fact of a distortion on the lower outer area of the diffuser. At this point, the flow distortion could be caused by a vortical flow structure and further plots will provide evidences to confirm this assumption.

Normalized vertical velocity $V$ is presented in Figure 13 where we observe similar contour on the slant as presented by the experimental reference of [11] on plane $X/L = 0$. When analysing the diffuser area, two vortices are identified at similar spanwise coordinates but different heights on DA5 case. From bottom to top, the lower side vortex and the diffuser vortex are in the same region, however it is not possible to assure they are merging at this point. Only the diffuser vortex is observed for DA10 and DA20 with anti-clockwise rotation direction. The case considering diffuser angle at 30° (DA30) also has an indication of two vortices, however only the lower side vortex (bottom) can be confirmed at this point. For the DA40 and DA50 cases, the lower side vortex has similar intensity in both case and a slightly different $V$ velocity distribution on the diffuser, with higher velocities in the most inclined diffuser.

Analysing plane $X/L = 0.096$, the inner spanwise component of the slant vortex is shifting downwards, once this flow structure starts to interact with the wake. On the diffuser area of DA5, the pair of vortices is merged into one single structure, with high $V$ velocity on the positive component of the new merged vortex. DA10 and DA20 cases maintain only one single diffuser vortex structure highlighted, moving slightly up and into the spanwise direction. The DA30 case still maintain the lower side vortex in similar position as in plane $X/L = 0$ and the additional structure does not behave as a vortex. The diffuser wake structure is still similar on DA40 compared to DA50, where the main difference relies on a low velocity zone at the mid-span of the diffuser, caused by the interference of slant vortex on the diffuser wake.

Wall shear stress lines on the surface of each diffuser case evaluated are presented in Figure 14. This analysis follows similar terminology and setup as presented for the Ahmed body squared-back.

The flow structure on the diffuser indicates a vortex touching the diffuser surface up to angle of 20° (DA20) and separated flow from DA30 onward. We identified three flow behaviour on the diffuser surface as the previous Ahmed body squared-back, detailed as follows. The DA5 and DA10 cases have the side diffuser vortex touching the diffuser surface together with a separation area at the diffuser inlet. The flow remains attached on the surface until reaching the outlet. Separated flow on the diffuser surface and diffuser vortex is observed in the DA20 case, with a partial
Computational study on an Ahmed Body equipped with simplified underbody diffuser

Fig. 12: Contours of normalized streamwise velocity $U$ for the Ahmed body with slant angle of 25° considering the proposed diffuser angles: 5° (DA5), 10° (DA10), 20° (DA20), 30° (DA30), 40° (DA40) and 50° (DA50) for planes $X/L = 0$ and $X/L = 0.096$.

Reattachment at the outlet. The last three diffuser angles of 30° (DA30), 40° (DA40) and 50° (DA50) have a fully separated flow all over the diffuser surface.
Fig. 13: Contours of normalized vertical velocity $V$ for the Ahmed body with slant angle of $25^\circ$ considering the proposed diffuser angles: $5^\circ$ (DA5), $10^\circ$ (DA10), $20^\circ$ (DA20), $30^\circ$ (DA30), $40^\circ$ (DA40) and $50^\circ$ (DA50) for planes $X/L = 0$ and $X/L = 0.096$.

6 Conclusions

A parametric study on the effect of diffusers is conducted on the Ahmed body at two slant cases: squared-back and $25^\circ$ angle. The diffuser length is fixed with the same dimension of the slant and cases are evaluated at a Reynolds number of $Re = 1.7 \times 10^6$ with moving ground condition. Diffuser angles considered range from $10^\circ$ to $50^\circ$ in increments of $10^\circ$, including one additional angle of $5^\circ$ for both cases.
Ahmed body squared-back results indicate two different flow behaviours. The first is observed in diffuser angles from 5° up to 30°, indicating that downforce increment leads to higher drag coefficient. Flow structure for this regime is composed of a lateral vortex with fully attached flow on the diffuser surface for diffusers up to 20°. The critical angle has similar structure considering the same diffuser vortex however the flow is partially separated on the diffuser surface. The second flow regime is found for diffuser angles higher than 30° where downforce increment efficiency is lost, and the flow is fully separated on the diffuser surface.

On the Ahmed body with 25° slant angle, downforce increases while the drag coefficient is reduced for diffuser angles up to 20°. Maximum downforce is observed at 10°. The flow structure is composed by the diffuser vortex and attached flow on the diffuser surface, however only for the 5° and 10° diffuser angles. The diffuser vortex is present in the 20° case but the flow is mostly separated. Similar to the squared-back case, the highest drag coefficient value is observed for the 30° angle, and flow on the diffuser surface is fully separated from this case onward.
Acknowledgements. We acknowledge the HPC facilities at Imperial College Research Computing Service, DOI: 10.14469/hpc/2232 and also under the UK Turbulence Consortium. CNPq for the sponsorship 202578/2015-1 and EPSRC under the grant.

References

[1] S.R. Ahmed, G. Ramm, and G. Faltin. Some salient features of the time-averaged ground vehicle wake. Technical report, SAE Technical Paper, 1984.
[2] Filipe F Buscariolo, Gustavo R S Assi, Julio R Meneghini, and Spencer J Sherwin. Spectral/hp methodology study for iles-svv on an ahmed body. In Spectral and High Order Methods for Partial Differential Equations ICOSAHOM 2018, page to be released. Springer, 2019.
[3] C.D. Cantwell, D. Moxey, A. Comerford, A. Bolis, G. Rocco, G. Mengaldo, D. De Grazia, S. Yakovlev, J-E. Lombard, D. Ekelschot, et al. Nektar++: An open-source spectral/hp element framework. Computer Physics Communications, 192:205–219, 2015.
[4] K.R. Cooper, T. Bertenyi, G. Dutil, J Sym, and G. Sovran. The aerodynamic performance of automotive underbody diffusers. Technical report, SAE Technical Paper, 1998.
[5] Suchuan Dong, George E Karniadakis, and C Chryssostomidis. A robust and accurate outflow boundary condition for incompressible flow simulations on severely-truncated unbounded domains. Journal of Computational Physics, 261:83–105, 2014.
[6] Jean-Luc Guermond and Jie Shen. Velocity-correction projection methods for incompressible flows. SIAM Journal on Numerical Analysis, 41(1):112–134, 2003.
[7] E. Guilmineau, G.B. Deng, A. Leroyer, P. Queutey, M. Visonneau, and J. Wackers. Assessment of hybrid rans-les formulations for flow simulation around the ahmed body. Computers & Fluids, 2017.
[8] A. Huminic and G. Huminic. Computational study of flow in the underbody diffuser for a simplified car model. Technical report, SAE Technical Paper, 2010.
[9] Lydia Jowsey. An experimental study of automotive underbody diffusers. PhD thesis, © Lydia Jowsey, 2013.
[10] S. Krajnović and L. Davidson. Large-eddy simulation of the flow around simplified car model. Technical report, SAE Technical Paper, 2004.
[11] H. Lienhart and S. Becker. Flow and turbulence structure in the wake of a simplified car model. Technical report, SAE Technical Paper, 2003.
[12] Gianmarco Mengaldo, Daniele De Grazia, David Moxey, Peter E Vincent, and SJ Sherwin. Dealiasing techniques for high-order spectral element methods on regular and irregular grids. Journal of Computational Physics, 299:56–81, 2015.
[13] T. Morel. Aerodynamic drag of bluff body shapes characteristic of hatch-back cars. Technical report, SAE Technical Paper, 1978.

[14] Rodrigo C Moura, Gianmarco Mengaldo, Joaquim Peiró, and SJ Sherwin. On the eddy-resolving capability of high-order discontinuous galerkin approaches to implicit les/under-resolved dns of euler turbulence. *Journal of Computational Physics*, 330:615–623, 2017.

[15] D. Moxey, M.D. Green, S.J. Sherwin, and J. Peiró. An isoparametric approach to high-order curvilinear boundary-layer meshing. *Computer Methods in Applied Mechanics and Engineering*, 283:636–650, 2015.

[16] A.E. Senior and X. Zhang. The force and pressure of a diffuser-equipped bluff body in ground effect. *Transactions-American Society of Mechanical Engineers Journal of Fluids Engineering*, 123(1):105–111, 2001.

[17] E. Serre, M. Minguez, R. Pasquetti, E. Guilmineau, G.B. Deng, M. Kornhaas, M. Schäfer, J. Fröhlich, C. Hinterberger, and W. Rodi. On simulating the turbulent flow around the ahmed body: A french–german collaborative evaluation of les and des. *Computers & Fluids*, 78:10–23, 2013.

[18] R.K. Strachan, K. Knowles, and N.J. Lawson. The vortex structure behind an ahmed reference model in the presence of a moving ground plane. *Experiments in fluids*, 42(5):659–669, 2007.

[19] Michael Turner. *High-order mesh generation for CFD solvers*. PhD thesis, Imperial College London, 2017.