Aerodynamic Simulations of a Generic Car Model

- Study of Underbody Diffuser Designs

Erik Nygårds- eriny141, Shreyas Maligemane- shrma734

INTRODUCTION

Aerodynamics is a key factor in increasing the performance of any transport by guiding the airflow in favourable ways to generate downforce or reduce drag. However, the aerodynamics of a road vehicle can often be quite complicated, and therefore need to be studied in order to get it right and to optimise.

A cheap and relatively quick way of doing this is to utilise a computational fluid dynamics (CFD) tool, such as ANSYS CFX. To reduce the complexity, a bluff body widely known as the Ahmed body [1] was considered as it's geometry and flow characteristics is similar to a typical car with a rear slant. Combining this simple geometry with CFD analysis means a wide range of geometric modifications can be tested quickly. These modifications can then be optimized by varying geometrical parameters.
A good way of adding grip to a car, especially at high speeds, is to add downforce. One of the best ways to generate large amounts of downforce, while adding little drag, is to utilise a diffuser on the underbody. Formula 1 cars, as of 2018 regulations, generated around 65% of their total downforce with the floor and diffuser, while adding only 15% of the total drag [2]. It is therefore the logical first area of improvement when aiming to generate downforce. Thus, a diffuser was added to the Ahmed body to improve the performance.

The aim of this study was to investigate the impact on aero-performance by different diffuser designs. The diffuser was studied with side skirts, 0-, 1-, and 2-strake configurations, and with sharp or rounded leading edge. Before altering the design, the bluff body was mesh verified, and validated using the SST turbulence model in comparison with previously validated large eddy simulation (LES) data. A thorough investigation on the geometry of the diffuser was carried out in order to primarily increase the generated downforce, e.g. decrease C_L, but also to minimize the drag. The parameters varied and studied were the diffuser angle, ride height, strake positions, and the leading edge radius for the rounded diffuser.

METHODS

The domain and geometry used can be seen in figure 1. The turbulence model used was SST, using high resolution discretization scheme with first-order turbulence numerics in order to reduce convergence issues. The inlet Reynolds number (Re) was set to 30000, using the square root of the frontal area as the characteristic length. Symmetry was used to reduce the computational time. Free-slip boundary condition was applied to the walls far from the body, to avoid having to use inflation layers there, as they are too far away from the body to impact it.

The mesh was constructed with unstructured tetrahedrons, with prism inflation layers. The mesh was verified using the method specified by Celik et al. [4], with the total drag and lift coefficients, C_D and C_L, as variables. The result of which can be seen in table 1.

The medium mesh was deemed good enough as the difference between the medium and fine mesh was less than 5% for all indicators. Since the SST turbulence model resolves the viscous sublayer, the

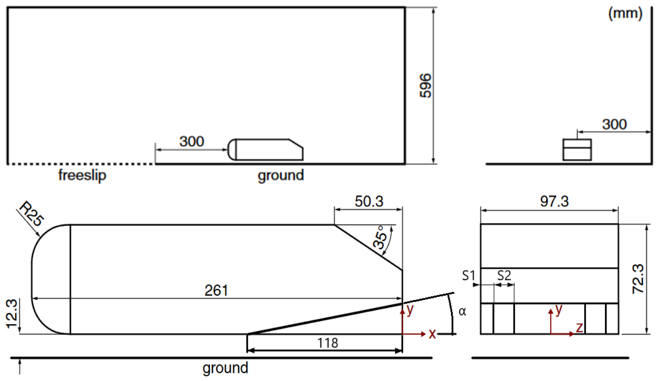


Fig. 1 . The domain used for the simulations in accordance with [3] and modified with a diffuser.

Tab. 1. The results of the mesh verification, in accordance with [4].

	#nodes			e_a^{32} [%]	e_{ext}^{32} [%]	GCI^{32} [%]	ea^{21} [%]	e_{ext}^{21} [%]	GCI^{21} [%]
N1	1244378	$r_{21} = 1.34$	C_D	3.19	2.94	3.57	1.47	1.24	1.57
N2	512183	$r_{32} = 1.33$	C_L	133.49	4.62	5.52	4.48	0.13	0.16
N3	217218								

inflation layers were made small enough to give a $y^+ < 1$ at all no-slip boundaries.

To validate the model, a comparison between the total-, pressure-, and friction-drag, and lift components for each part of the car was made between the model and the given LES data. The comparison can be seen in figure 2.

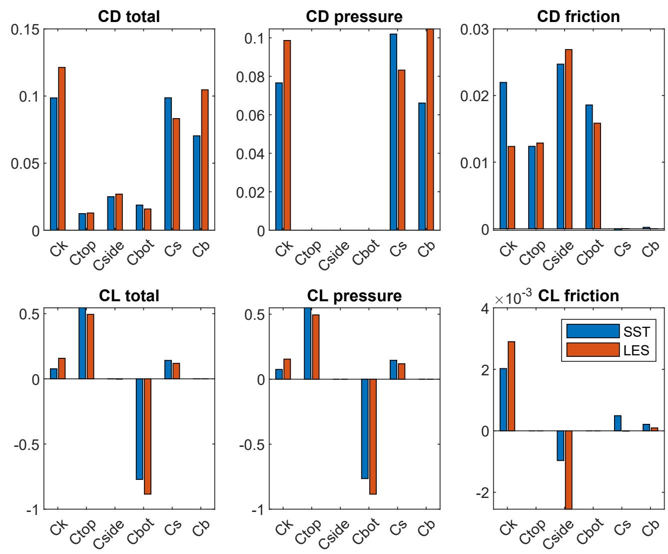
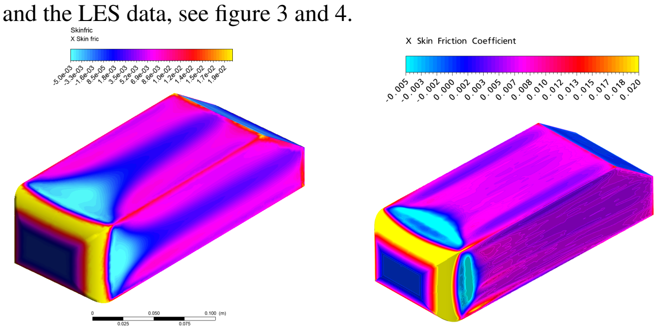


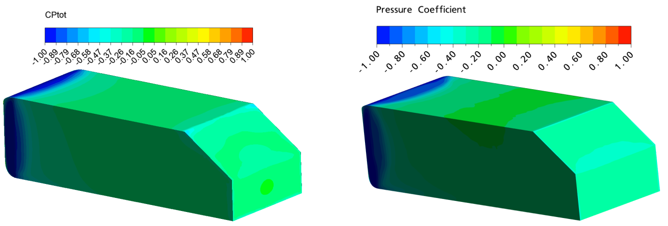
Fig. 2 . Comparison of the forces at different locations of the body between the LES data and the SST-model. Where Ck is the coefficient at the frontal surfaces, Ctop the top surface, Cside the sides, Cbot the underside, Cs the rear slope, and Cb the rear.

It is clear that there are some small differences between the two data sets. The model follow the trends of the LES data quite nicely, however there are some outliers. For example the friction drag at the front, which shows quite a bit higher values than the LES data, however as all other areas are quite close it was deemed okay. The results of this validation was deemed to be good enough. To further validate the model, a qualitative comparison between the surface skin friction and pressure coefficient was made between the SST-model and the LES data, see figure 3 and 4.



(a) RANS - SST (b) LES

Fig. 3 . Skin friction coefficient(C_f) contour comparison



(a) RANS - SST (b) LES

Fig. 4 . Pressure coefficient(C_P) contour comparison

In general the results look quite similar, however the SST-model predicts a larger region of separation along the centerline at the top and sides of the car. Another notable differences is that the LES data predicts slightly stronger vortices near the floor edge compared to the SST-model. At the slant the models look quite similar, however the SST model predicts a higher pressure coefficient, especially at the back below the slant. This explains the lower pressure drag predicted by the SST model at the back, see figure 2. However, as all other structures look very similar, the model was considered accurate enough, and thus fully validated.

The diffuser design implemented had a fixed start location, at $x = -118[mm]$, see figure 1. Due to the high C-pillar slant angle of 35°, the diffuser angle, α, could not exceed 17° without removing the rear face of the body. Thus, simulations were run with $1° \leq \alpha \leq 17°$, and with ride heights ranging from 5 to 15 mm. Both a sharp and rounded leading edge (LE) was studied, with the radius being $2 \leq r \leq 50$ $[mm]$. Road and racing cars usually have inboard strakes to handle the airjets shooting in from the tires squishing the air into the ground [5]. Therefore the effects of strakes on a diffuser without the tires was studied to see how they influence a diffuser in perfect conditions. To do this the sharp LE diffuser was simulated with 0-, 1- and 2-strake designs, and the rounded LE diffuser with 0- and 1-strake designs. All simulations had skirts, extending to the same level as the floor. The strakes spacing was studied, with the outboard-most strake positioned between 5-20 mm, and the inboard strake 10-40 mm, from the side of the body. However the strakes were never closer than 5 mm.

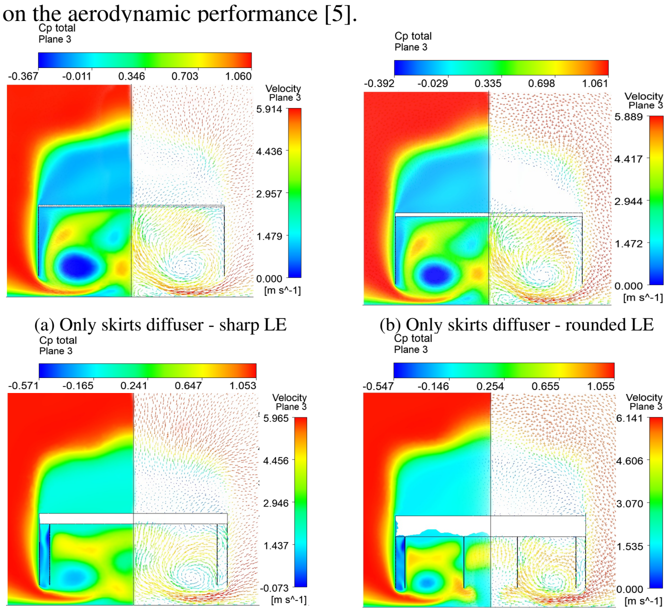
A design of experiments study was set up. With design points generated using Optimal Space-Filling and Latin Hypercube Sampling Design. A response surface was then generated using Kriging or Genetic Aggregation, depending on lowest error to verification points. The response surfaces had a Root Mean Square Error (RMSE) of less than 9% to verification points. The design was then optimised for lowest C_L, with low drag. The optimal designs were then simulated for verification and presented.

RESULTS & DISCUSSION

In the validation there were bigger differences between the SST model and the LES when comparing skin friction rather than pressure coefficient. The skin friction differences show that the SST model over predicts the top surface separation, see figure 3. This could be due to how the time averaging is done between the two models. As LES is a transient method, which also resolves more of the turbulence, the intricate structures of the flow are captured more accurately. Thus when the time averaging is done afterwards, the results are significantly more accurate. The SST model does however capture the pressure variations on the top and sides of the model quite closely to the LES model, see figure 4.

The pressure difference between the free stream air and the diffuser leads to the formation of two counter rotating longitudinal vortices on the edges along the diffuser. These vortices help in capturing the high energy air entering into the diffuser and maintains the low pressure in the center of the diffuser [6]. The vortices also keeps the airflow attached at high diffuser angles [7]. However, these rotational vortices are more effective if they do not burst and stay attached to the skirts and strakes of the diffuser. A diffuser with no skirts not only leads to pressure losses, but also has reduced downforce in yaw conditions [8], as the vortices can freely move over the diffuser. In contrast, having skirts sealed to the ground is inefficient. As the flow is stalled due to the reduction of inflow from the vortex pairs, and the effective downforce is lesser than a no-skirts diffuser [8]. Figure 5 shows how side skirts are effective in keeping vortices and the flow bounded.

The vortices seen in figure 5 for rounded and sharp LE carries more energy and stays undisturbed when compared to the others. This contributes to achieve maximum downforce by allowing the diffusers to have a higher entry to exit area ratio, with the rounded LE having a ratio of 1 : 3.26. In the presence of strakes the vortices carry less energy and are not as efficient as the no-strake diffusers. Although in the presence of tyres and yaw condition the strakes can have a positive impact on the aerodynamic performance [5].



(a) Only skirts diffuser - sharp LE (b) Only skirts diffuser - rounded LE

(a) 1-strake diffuser - sharp LE (b) 2-strake diffuser - sharp LE

Fig. 5 . $C_{p,tot}$ and velocity vectors at X = 0 [mm]

Comparing the C_L of the different optimised diffuser designs, it is clear a 0-strake diffuser is preferred, see figure 6. The reason for this is that the strakes prevent the skirt vortices from attaching properly to the skirts, and instead attach at the more inboard strakes. This limits the effective diffuser volume, since the fluid within the strake gaps contain very low energy, see figure 5, as low $C_{p,tot}$ values indicate low energy flow. As can be seen in figure 6, both single strake diffuser designs show a rather different pattern in both C_L and C_D variation with ride height. The optimal points, which can be seen in table 2, indicate that the more strakes the diffuser has, the lower the ride height and angle for max downforce. This could be why the single strakes variations look different, however this is not supported by the 2-strake being similar to the 0-strakes. The reason the straked diffusers have higher performance at lower ride heights is likely due to the counter-rotating vortices, see figure 5, being smaller the more strakes are added. The smaller the vortices are the less likely they are to burst due to the boundary layer of the road [7]. Allowing the diffuser to be efficient at lower ride heights. Furthermore it follows that the diffuser angle is reduced with the ride height, as per figure 7, for maximum performance.

As per figure 7, the downforce is very low for max ramp angles at small ride heights. The reason behind this is the reduced height would not provide necessary space for the vortices to travel and they might burst. Also, at lower ride heights the flow is extremely unsteady, which RANS cannot capture well [8]. This limitation might explain why there is no sharp, cliff-like, drop off in performance with lowered ride heights, which has been shown to occur in experimental cases [6] [7].

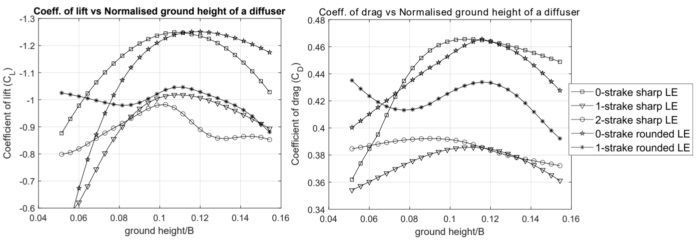


Fig. 6 . The force coefficients, C_L left, C_D right, for the different diffuser designs against ride height normalised by the breadth of the body.

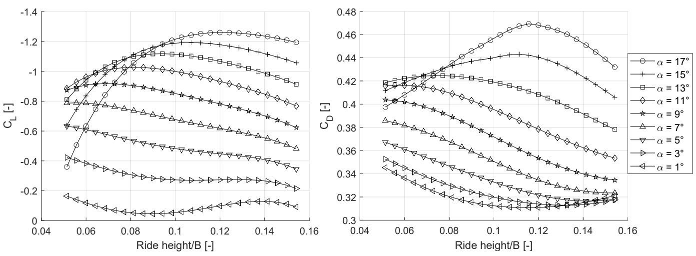


Fig. 7 . C_L (left) and C_D (right), varying with ride height normalised by the breadth of the body, for different diffuser angles, α, for an optimised rounded LE no strakes diffuser design.

Comparing the flow behaviour at the surface of the no-strake sharp LE diffuser to experimental findings of a diffuser in the max downforce generating region [8] shows good agreement, see figure 8. With both results showing a central separation region, or bubble, with separation regions downstream at the skirts.

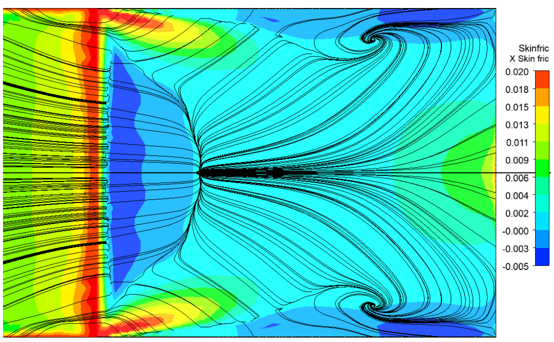
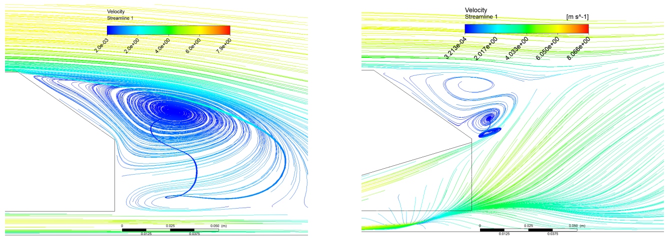


Fig. 8 . Skin friction and streamlines on the optimised no strake sharp LE diffuser surface.

The static pressure and velocity of the air at the exit of diffuser is almost equal to the air present at the rear of the bluff body. The velocity gained at the entry of diffuser is eventually lost due to the expanding geometry creating upwash. This helps in reducing the size of wake formed behind the bluff body. This can be seen in figure 9. At the same time, the diffuser adds more drag due to the large amounts of downforce it generates.



(a) Ahmed body (b) No strake diffuser rounded LE body

Fig. 9 . Backward direction velocity contoured stream lines

Tab. 2. The parameters for the optimal points, and their resulting lift and drag.

Sharp LE	Ride height/B [-] (Ride height [mm])	Angle α [°]	Strake 1 pos. [mm]	Strake 2 pos. [mm]	C_L	C_D
2-strake	0.102 (9.924)	12.609	5.062	35.070	-0.958	0.407
1-strake	0.108 (10.5)	15	5.525	-	-0.996	0.421
0-strake	0.116 (11.3)	17	-		-1.241	0.465
Rounded LE				Radius [mm]		
1-strake	0.108 (10.5)	15	5.525	2.248	-1.05	0.432
0-strake	0.111 (10.796)	16.647	-	2.263	-1.304	0.472
Base model	0.126 (12.3)	-			-0.009	0.324

Conclusively, an optimal diffuser design in terms of generated downforce, without influence from tires, is a zero strake, high angle diffuser with either rounded or sharp LE. As there seems to be only small differences between sharp and rounded LE. Adding significant downforce, while only increasing the drag by around 47%, see table 2. However, diffusers can be designed in many more ways than explored in this report, for example by adding passive flow-control methods [7]. Which can result in even greater amounts of downforce. Strakes should also not be ruled out, as they can be beneficial when tires are present near the diffuser [5], however this requires further investigation.

REFERENCES

[1] Ahmed, S, Ramm, G, and Faltin, G. *Some Salient Features Of The Time Averaged Ground Vehicle Wake*. SAE Technical Paper 840300, 1984.

[2] Vyssion and jin9128. The ground effect. *F1 Technical*, 2018.

[3] Venning J, Lo Jacono D, Burton D, Thompson M and Sheridan J. The effect of aspect ratio on the wake of the ahmed body. *Experiments in Fluids*, vol. 56, no. 6, pp. 1–11, 2015.

[4] I Celik, U Ghia, Patrick J Roache, CJ Freitas, Hugh Coleman, Peter E Raad. Procedure for estimation and reporting of uncertainty due to discretization in cfd applications. *ASME*, vol. 130, 2008.

[5] Eric Hazen and Paul Lucas. Diffusers - how they work. *Verus Engineering*, 2018.

[6] Ruhrmann A, Zhang X. Influence of diffuser angle on a bluff body in ground effect. *Journal of Fluids Engineering*, vol. 125, 2003.

[7] Ehirim O, Knowles K, Saddington A and Finnis M. Aerodynamics of a convex bump on ground-effect diffuser. *ASME Journal of Fluids Engineering*, vol. 140, no. 9, 2018.

[8] Ehirim O, Knowles K and Saddington A. A review of ground-effect diffuser aerodynamics. *ASME Journal of Fluids Engineering*, vol. 141, no. 2, 2019.