



INVESTIGATION OF VEHICLE RIDE HEIGHT AND DIFFUSER RAMP ANGLE ON DOWNFORCE AND EFFICIENCY

Journal:	<i>Part D: Journal of Automobile Engineering</i>
Manuscript ID	JAUTO-16-0382.R1
Manuscript Type:	Original Article
Date Submitted by the Author:	26-Jan-2018
Complete List of Authors:	knight, jason; University of Portsmouth, spicak, milan; Vysoke uceni technicke v Brne Fakulta strojniho inzenyrstvi Kuzenko, Antons; University of Hertfordshire, school of Engineering Haritos, George; University of Hertfordshire, school of Engineering Ren, Guogang; University of Hertfordshire, school of Engineering
Keywords:	vehicle aerodynamics, vehicle simulation/ modelling, alternative vehicle designs, automotive components, Fuel efficiency/ economy, diffuser, downforce, ahmed
Abstract:	Diffusers are typically used in motorsport to generate negative lift (downforce). They also reduce aerodynamic drag and so significantly enhance aerodynamic efficiency. The amount of downforce generated is dependent on ride height, diffuser ramp angle and its relative length to that of the vehicle length. This paper details a numerical investigation of the effects of ride height and diffuser ramp angle in order to find an optimum downforce and efficiency for the inverted Ahmed model. A short and long diffuser with ratios of 10% and 35% respectively to that of vehicle length are studied. The short diffuser produced lower maximum downforce and efficiency at a lower ride height and lower angle when compared to the longer diffuser. The long diffuser produced highest downforce and the best efficiency with a ramp angle of 25 degrees at ride height ratio of 3.8% when compared to vehicle length. Different ride heights were found to correspond to different diffuser ramp angles to achieve optimum downforce and efficiencies.

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60

SCHOLARONE™
Manuscripts

For Peer Review

INVESTIGATION OF VEHICLE RIDE HEIGHT AND DIFFUSER RAMP ANGLE ON DOWNFORCE AND EFFICIENCY

Jason Knight¹, Milan Spicak², Antons Kuzenko³, George Haritos³, Guogang Ren³

¹University of Portsmouth, Anglesea Road, Portsmouth, PO1 3DJ, UK

²Brno University of Technology, Antonínská 548/1, 601 90 Brno, Czech Republic

³University of Hertfordshire, College Ln, Hatfield AL10 9AB, UK

¹Email: jason.knight@port.ac.uk (corresponding author)

Abstract

Diffusers are typically used in motorsport to generate negative lift (downforce). They also reduce aerodynamic drag and so significantly enhance aerodynamic efficiency. The amount of downforce generated is dependent on ride height, diffuser ramp angle and its relative length to that of the vehicle length. This paper details a numerical investigation of the effects of ride height and diffuser ramp angle in order to find an optimum downforce and efficiency for the inverted Ahmed model. A short and long diffuser with ratios of 10% and 35% respectively to that of vehicle length are studied. The short diffuser produced lower maximum downforce and efficiency at a lower ride height and lower angle when compared to the longer diffuser. The long diffuser produced highest downforce and the best efficiency with a ramp angle of 25 degrees at ride height ratio of 3.8% when compared to vehicle length. Different ride heights were found to correspond to different diffuser ramp angles to achieve optimum downforce and efficiencies.

Keywords: Downforce, Diffuser, CFD, Efficiency, Optimisation, Ahmed.

Introduction

Aerodynamic component parts are commonly used in motorsport for generating downforce which leads to more traction force around corners.¹ This can lead to achieving shorter lap times on high downforce circuits. One such component is the diffuser which is mounted underneath the car at the tail near the rear axle. In addition to increasing downforce, diffusers also reduce drag thereby significantly increasing aerodynamic efficiency of the race car.² Like other extra-components in racing cars, diffusers are also increasingly being used on production vehicles to reduce aerodynamic drag.³ Various materials including metals, metal bonded glass composites have been used to improve the aerodynamic efficiency by reducing skin friction drag on racing cars.⁴ More recently, advanced lightweight materials such as carbon fibre composites and eco-friendly natural fibre composites are starting to be used in varieties of car exterior body parts to investigate the possibility to cope with the large aerodynamic forces experienced whilst keeping the weight down and considering sustainability.⁵⁻⁷

Theoretically the main principle of generating down force with the diffuser is from the Venturi effect.^{8,9} The significance of this effect is related to the change of the cross-section area of flow underneath the car at the start and end of the diffuser. The diffuser angle has a large influence on the rate of change of those areas.¹⁰ The ride height has a large influence on the cross section area at the start of the diffuser.¹¹

Many studies have been performed using the Ahmed model in standard configuration with slanted edge uppermost.¹²⁻¹⁴ Tunay¹⁵ performed wind tunnel tests using PIV for flow

1
2
3
4
5
6
7
8 visualisation and matched this with CFD studies using LES turbulence model.¹⁶ Other LES
9 studies using the Ahmed model have been performed.¹⁷⁻¹⁹ These works have captured the flow
10 physics for use in analysing wake characteristics with aim of reducing drag.
11
12
13
14

15
16 The same Ahmed model has been inverted and used by Senior²⁰ so that the slanted surface is
17 closest to the ground to represent the diffuser of a simplified vehicle. Puglisevich²¹ has used the
18 same approach with LES turbulence model. Senior²⁰ varied the ride height and studied the effect
19 of separation in the diffuser with resulting losses in downforce. Ruhrman²² has extended this
20 work to investigate the ride height and ramp angle using a matrix approach. Humunic²³ has also
21 used this approach to study various lengths of diffuser combined with various ramp angles.
22
23
24
25
26
27
28
29

30 Other multi-parameter optimisations of automotive shapes have been studied by Han.^{24,25} They
31 investigated various upper-surface backlight angles with various boatailing angles at the side
32 combined with diffuser ramp angles. The ride heights of models with diffusers of other
33 automotive shapes have also been investigated.^{26,27} Cooper²⁷ also looked at the effect of fixed
34 and moving floors on the performance of diffuser. The more complex geometry of a Formula 3
35 car diffuser has been studied by Peddie.²⁸ Jens²⁹ has used a response surface approach for
36 optimisation of a diffuser in supersonic flow.
37
38
39
40
41
42
43
44
45

46 In this work, we concentrate on the interaction between the parameters of ride height and ramp
47 angle. We follow the work of Senior²⁰ and use the same inverted Ahmed model for its simple
48 geometry. The slanted edge is lowermost and closest to the ground to represent a simplified
49
50
51
52
53

diffuser [as pictured in Figure 1a](#). We use the same analysis approach used by Ruhrman²² in this work for a long diffuser. However, we follow the response surface approach used by Jens²⁹ to study the short diffuser. [A schematic of our approach is shown in Figure 1b detailing the parameters used.](#)

We initially report wind tunnel setup and results which have been used to validate the CFD models without the ground. Thereafter the ground is added to the CFD model and the methodologies used are described. The CFD simulation for the long diffuser was achieved using Star CCM+ with a matrix of ride heights and diffuser ratios investigated. The short diffuser simulations have been achieved using Ansys® Fluent with the optimization tool. The relationship between the model ride height and diffuser ramp angles on generated downforce and efficiencies are presented for the long and short diffuser of 35% and 10% relative lengths respectively. Finally some comparisons are drawn and conclusions offered.

Validation of Flow around Inverted Ahmed Model

The open jet wind tunnel at the University of Hertfordshire has been used for obtaining pressure distribution along the centreline of the slanted surface side of the Ahmed body. [The pressure measurements were obtained using a scani-valve connected to a digital manometer. The results from this are compared to the Computational Fluid Dynamics simulation to demonstrate confidence in the approaches reported later.](#) The tunnel has a circular cross-section of 480mm diameter. The speed of the wind tunnel was set at 20m/s and turbulence intensity measured at 1%. These conditions are used [as inlet boundary conditions](#) in the CFD ~~models~~ [approach](#).

1
2
3
4
5
6
7
8
9
10 The Ahmed model used had a cross-sectional area of 0.029m^2 (145mm deep x 200mm wide)
11
12 resulting in a blockage ratio of 4%. According to Barnard, a less than 5% blockage in an open
13 jet tunnel is recommended to provide a realistic aerodynamic flow. A 5% blockage would
14 typically give accurate results to within 10% difference in drag measurements. Due to this
15 relatively low blockage, no correction factors were applied and the difference accepted within
16 the error estimate. Pressure tapings are located along the longitudinal centreline. The model is
17 aligned with the flow and supported by overhead struts and a tailwire as pictured in Figure 1a.
18
19

20
21
22
23
24
25
26 The CFD approach used a velocity for the inlet and a pressure for the outlet boundary
27 conditions respectively. A non-slip wall was used for the Ahmed model surface and slip wall
28 conditions used for the remaining boundaries of the domain. The Navier-Stokes equations are
29 solved using finite volume method within Star CCM+ and Fluent. In both approaches, the
30 SIMPLE algorithm is used with Central differencing schemes for all the computations.
31 Convergence is obtained for all simulations with criteria set at a minimum 0.001 in all
32 dependant variables reported in this work.
33
34
35
36
37
38
39
40
41
42
43

44 The CFD simulation has been setup to provide an ~~accurate~~ solution to within 105% of the
45 experiment. Much improved accuracy can be obtained by increasing the resolution of the mesh
46 but at the expense of computational time. The reduced accuracy enables an increased number of
47 simulations to find the optimum parameters of the diffuser for a given timeframe. Both Fluent
48
49
50
51
52
53

1
2
3
4
5
6
7
8 and Star CCM+ were used in the validation of the Ahmed model in freestream air. In all cases
9
10 the flow was assumed to be steady state and so a symmetry boundary was applied along the
11
12 longitudinal centreline. The results from Fluent and Star CCM+ were found to be within 1%
13
14 difference in pressure coefficient, C_p , so only the Fluent results are reported here.

15
16
17
18
19
20 The standard k-epsilon turbulence model has been widely used and validated in literature. In
21 addition we use the k- ω SST turbulence model in our validation. Both models are 2 equation
22 models and provide a solution to the RANS with a reasonable degree of accuracy at a
23 reasonable computational expense. The use of more accurate models such as DES and LES
24 would require much increased computational power and is beyond scope of the current work.
25
26
27
28
29

30 Figure 2 shows the velocity magnitude contour plot of the inverted Ahmed model using the k- ω
31
32 SST turbulence model. The inset in Figure 2 shows a zoomed in section at the rear of the model
33
34 showing the separated shear layers from top and bottom surfaces.

35
36
37
38 The pressure distributions from the wind tunnel measurements and the k- ω SST and the
39
40 standard k-epsilon turbulence models³⁰ are shown in Figure 3. The distribution of pressure taps
41
42 along the chord line of the upside down Ahmed body can also be seen in Figure 3 as discrete
43
44 data points. The pressure distribution has been non-dimensionalised to obtain the pressure
45
46 coefficient, C_p .

1
2
3
4
5
6
7
8 Good agreement can be seen over most of the lower surface of the model. Notable differences
9
10 are found near start of the diffuser. This inaccuracy can influence the absolute value of the
11
12 coefficient of lift (Cl) and coefficient of drag (Cd). However, in this current work with multi-
13
14 parameters, the trend is deemed more important than the absolute value and is assumed to not
15
16 alter significantly. According to the CFD results, the flow is accelerated around the starting
17
18 sharp corner of the diffuser which leads to high suction peaks. This is not evident in the
19
20 experimental measurements, albeit with the sparse measurement locations used. The k- ω SST
21
22 model has a closer agreement to experiment as can be seen in Figure 3 and is deemed to be a
23
24 more realistic value in this region. A coarser mesh with half the resolution was also run using
25
26 the k- ω SST model to check the mesh dependency. The pressure co-efficient was within 4% of
27
28 the finer mesh over most of the model. Larger differences were noted at the start of the diffuser
29
30 where the pressure co-efficient value obtained was similar to that obtained with the standard k-
31
32 epsilon turbulence model. More resolution in experiment and simulation would improve the
33
34 correlation. Nevertheless, the result serves to validate the simulation for current purposes. The
35
36 k- ω SST turbulence model is deemed more suitable for the purposes of dealing with flow in the
37
38 diffuser and is used for the remainder of simulations.

39
40
41 Both the experimental and computational approaches contain errors. A high quality experiment
42
43 using sensitive equipment and accounting for blockage compared with a high fidelity CFD
44
45 simulation would improve the accuracies and correlation between approaches. However, in this
46
47 work with a large number of simulations, we concentrate on identifying the trends and accept
48
49 the inaccuracies, which are estimated to be within 10% between approaches. The comparison of
50
51

pressure distributions are presented here merely to demonstrate confidence in the CFD approaches.

Computational Methodologies used for Inverted Ahmed Model in ground effect

The diffuser geometries being investigated were of 0.65 and 0.9 diffuser ratios, meaning that diffuser starts at 65% and 90% respectively of the length of the model. Its size is therefore 35% and 10% respectively of the length of the model. We name the 35% relative length diffuser as the long diffuser and the 10% relative length diffuser as the short diffuser.

The ground was added to the CFD models used in the validation. The boundary condition applied at the ground was set to slip wall conditions to model the road or moving floor. A polyhedral mesh was used for the long diffuser. The mesh base size used was 0.13 m with the model minimum and target mesh sizes being 5% and 10% of the base size, respectively. With this mesh a good mix of accuracy and computing time was achieved to facilitate a large number of simulations within a given timeframe. However, for low ride height values the mesh had to be refined to produce sensible values and achieve convergence, which resulted in longer computing times. The long diffuser allowed for experimentation with different diffuser ramp angles up to a maximum of 38.5° which would result in a sharp trailing edge of the Ahmed model. The angles investigated were chosen to be 35° , 30° , 25° , 20° , 15° , 10° and 5° . The ride heights were chosen to be 50, 40, 30, 20, 10 and 5mm. The lowest ride height being 5 mm off the ground was added to study the area where the ground effect has the most significant

1
2
3
4
5
6
7
8 influence and to ensure a maximum point was reached in the results presented later. All 6 ride
9 heights were simulated with all 7 angles, resulting in 42 simulation runs.

10
11
12
13
14 Fluent has been used to investigate the short diffuser using a similar approach to that used when
15 investigating the long diffuser. However, the mesh has been defined as the default tetrahedrons
16 with refining box of influence and inflation of prismatic layers on the walls. In addition, a
17 different analytical approach to finding the optimum values has been used, namely, the Ansys
18 Response Surface Optimization tool.³¹ For this purpose the two input parameters of ride height
19 and diffuser ramp angle have been defined in the Design Modeller. Four output parameters have
20 been monitored which were, Number of Mesh elements and Mesh Skewness for purpose of
21 validity of the solution and Coefficient of Lift and Drag for optimization.³¹ Based on a reduced
22 number of simulations compared to the long diffuser case, a response surface has been created,
23 which determines the probability of occurrence of the desired solutions. From this assumption
24 the optimization has been performed for maximising downforce coefficient, $-C_l$ and maximising
25 downforce efficiency, $-C_l/C_d$. Results from the optimization have been verified, which has
26 created additional simulation points for a new response surface. This closed loop procedure has
27 been repeated until the solution has been satisfied.³¹ In all cases convergence was obtained. A
28 typical mesh of the short diffuser is shown in Figure 4.

43 44 45 46 **Results**

47 Figure 5 shows the change in downforce, $-C_l$, for various ramp angles and ride heights of the
48 long diffuser. As the long diffuser model approaches the ground it experiences an increase in
49
50
51
52
53

1
2
3
4
5
6
7
8 downforce for all ramp angles except 35°. The 35° diffuser shows very little change over with
9
10 all ride heights above 10mm. All other diffusers show a clear optimum or maximum point is
11
12 reached. All diffuser ramp angles show a reduction in downforce at the closest point to the
13
14 ground as can be seen in Figure 5. At 20 mm ride height the maximum downforce coefficient is
15
16 1.43 and was produced by 25° diffuser ramp angle, whereas at 10 mm ride height the maximum
17
18 downforce is 1.41 but was produced by the 20° ramp angle (see Figure 5). 5° and 10° diffuser
19
20 ramp angles produced the least downforce in general, although, interestingly, at 10 mm ride
21
22 height, the 10° diffuser ramp angle produced more downforce than 30° and 35° diffuser ramp
23
24 angle arrangements. At this ride height the 10° diffuser is near its optimum, whereas the 30°
25
26 diffuser is too close to the ground to work efficiently and the 35° diffuser does not work
27
28 efficiently at any angle. At the lowest ride height tested of 5mm, the optimum diffuser ramp
29
30 angle is shown to be 10° in Figure 5. At this ride height none of the diffusers are working
31
32 efficiently, but the 10° diffuser is shown to give most downforce.
33
34
35

36 The peaks of maximum downforce were found at 10mm ride height for diffusers with 20°, 15°,
37
38 10° and 5° ramp angles. 20mm ground clearance was optimum for 25° diffuser and 30mm
39
40 ground clearance for 30° diffuser. This shows the optimum downforce is a combined function of
41
42 ride height and ramp angle. Therefore, the 30° ramp angle diffuser is best for ride height of
43
44 30mm and above, the 25° ramp angle diffuser for a ride height of 20mm and the 20° ramp angle
45
46 diffuser is best for a ride height of 10mm. It should be noted more accurate results could be
47
48 obtained refining the simulations or using polynomial functions. Nevertheless, a clear
49
50 dependency between ramp angle and ride height on downforce generation is shown.
51
52
53
54
55
56
57
58
59
60

1
2
3
4
5
6
7
8
9
10 Aerodynamic efficiency is a measure of downforce generation with respect to drag. Drag is an
11 undesirable effect on the vehicle travelling through air and downforce is generally a desirable
12 outcome of a design. As can be seen from Figure 6, almost every ride height studied has its own
13 corresponding optimum diffuser angle that results in the highest efficiency. For example, at 30
14 mm ride height the most efficient ramp angle can be deduced to be around 27° , whereas for 20
15 mm ride height the most efficient angle is around 24° and even lower at 10 mm ride height the
16 most efficient angle is around 19° . Between 10-30mm ride heights, the maximum efficiencies
17 are within 1% of each other albeit with different ramp angles. Obtaining this data was only
18 possible due to the matrix approach taken, where all ride heights were simulated with all ramp
19 angles. Otherwise, if initially a single feature, ride height or angle, was found first and the
20 second parameter adjusted to find the optimum, the real optimum may be overlooked. In this
21 case, the most efficient setup is at 20 mm off the ground with 25° diffuser ramp angle where
22 aerodynamic efficiency has maximum value of 3.03. The 20mm ride height corresponds to a
23 ratio of 3.8% when compared with length of model.
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41

42 The results from the short length diffuser optimisation study are shown in the Figure 7. The
43 contour plot of dependence of the diffuser parameters on the down force (-Cl) as well as
44 downforce efficiency (-Cl/Cd) are presented. The optimum ride height and diffuser angle for
45 downforce generation has been determined as 12mm and 10° respectively. This corresponds to a
46 maximum coefficient of lift of -0.625. This is also very close to the most aerodynamically
47
48
49
50
51
52
53

1
2
3
4
5
6
7
8 efficient point as well being at the same ramp angle but slightly higher ride height of 13mm,
9 which corresponds to a ratio of 2.5% when compared with length of model. This suggests the
10 drag produced has a smaller effect on efficiency with respect to downforce in short diffusers.
11
12
13
14
15

16 The downforce efficiencies for both long and short diffusers are shown in Figure 8 for varying
17 ramp angles at 10 and 20mm ride heights. These ride heights are in the region of optimal
18 efficiency and allow a direct comparison between the short and long diffusers to be made. The
19 trend in downforce is very similar to the trend downforce efficiency so only the downforce
20 efficiency is reported here. As can be seen from Figure 8, the trends in downforce efficiencies
21 do vary similarly with changes in ride height and ramp angle for the both the long and short
22 diffusers. The comparison clearly shows the much larger downforce efficiency of the long
23 diffuser as expected due to increased surface area. The long diffuser achieves these higher
24 efficiencies at higher angles when compared to the short diffuser. Conversely, the short diffuser
25 has lower maximum efficiencies at lower angles when compared to the long diffuser. The
26 profile of efficiencies in the short diffuser can also be seen in Figure 8 to be flatter, which
27 suggests lower sensitivities for shorter diffusers.
28
29
30
31
32
33
34
35
36
37
38
39
40
41

42 **Conclusions**

43
44 The CFD setup of an inversed Ahmed model in free stream was validated by a physical
45 experiment in an open jet wind tunnel to close agreement on most of the lower surface. Some
46 discrepancies were noted near the start of the diffuser which requires better resolution in
47 experiment and numerical approaches to improve accuracy.
48
49
50
51
52
53
54
55
56
57
58
59
60

Downforce values and efficiency were obtained for multiple ride heights with different diffuser ramp angles for a long and a short diffuser. Both diffusers show similar trends to changes in ride height and ramp angle but at different absolute values.

The most efficient arrangements for the long diffuser ratio were found to be in close proximity over a relatively large variation in ride height. A~~within~~ 1% difference in efficiency was noted using 10 to 30 mm ride heights. However, the higher the ride height, the higher the ramp angle needed to maintain optimal efficiency. At 10 mm ride height a 19° diffuser ramp angle was optimum, whereas at ~~20 mm and 30mm~~ ride heights, ~~24° and 27°~~ ramp angles ~~respectively~~ were found to be the most efficient. ~~Thence, the higher the ride height, the higher the ramp angle needed to maintain optimal efficiency.~~ The most efficient arrangement overall for the Ahmed Model with long diffuser of 35% relative length diffuser was found to have a downforce/drag ratio of 3.03 at 20 mm ride height with 25° diffuser ramp angle in 20 m/s air speed. The 20mm ride height corresponds to a ratio of 3.8% when compared with length of model.

The short diffuser model produced much lower values of downforce and efficiency values as expected from reduced area of the diffuser. The short diffuser was found to be more sensitive in the production of highest downforce ~~and best~~ efficiency. For the same 1% difference in maximum efficiency a much smaller range of ride heights was observed. At 11 mm ride height a 9° diffuser ramp angle was optimum, whereas at 14mm ride height, 11° ramp angle was found

1
2
3
4
5
6
7
8 to be the most efficient. at a ramp angle of 10 degrees. The most downforce was obtained with
9 ride height ratio of 12mm which corresponds to a ratio of 2.1% when compared with length of
10 model. The most efficient production of downforce in the short diffuser was again at 10 degrees
11 but at a slightly increased ride height ratio of 2.5%.
12
13
14
15
16
17

18 The optimum downforce and efficiency for long diffusers was found to be more sensitive, larger
19 and occurring at higher ride heights and angles, when compared to the short diffuser. The short
20 diffuser was found to be more sensitive. Different ride heights correspond to different diffuser
21 angles for optimum downforce and efficiency in both the long and short diffusers.
22
23
24
25
26
27

28 **Acknowledgments**

29 The authors would like to acknowledge the use of the University of Hertfordshire wind tunnel,
30 Star CCM+ and ANSYS FLUENT CFD software used in this work.
31
32
33
34
35

36 **References**

- 37
38
39 1. Katz J. *New Directions in Race Car Aerodynamics: Designing for Speed*. 2nd ed.
40 2008.
41
42 2. Zhang X et al. Ground Effect Aerodynamics of Race Cars. *Applied Mechanics*
43 *Reviews* 2006; 59(1): 33.
44
45 3. Bansal R and Sharma RB. Drag Reduction of Passenger Car Using Add-On Devices.
46 *Journal of Aerodynamics* 2014: 1-13.
47
48 4. Loftus D, Found MS and Yates JR. The performance of aluminium to carbon fibre
49 composite bonded joints in motorsport applications. *Sports Engineering* 2002; 2(4):
50 201-257.
51
52
53
54
55
56
57
58
59
60

5. Graham T. Surface coatings offer opportunities for protection in harsh environments. In: *Transactions of the IMF*; 89, Issue 2 (March 2011), pp. 71-72.
6. Jawaid M and Abdul Khalil HPS. Cellulosic/synthetic fibre reinforced polymer hybrid composites: A review. *Carbohydrate Polymers* 2011; 86: 1– 18.
7. Pil L, Bensadoun P, Parisent J and Verpoest I. Why are designers fascinated by flax and hemp fibre composites. *Composites: Part A* 2016; 83: 193-205.
8. Hucho WH. *Aerodynamics of road vehicles*. 4th ed. Warrendale, Pennsylvania: SAE International. 1998.
9. Barnard RH. *Road vehicle aerodynamics*. 3rd ed. Mechaero Publishing, 2010.
10. Humnic A et al. Study of aerodynamics for a simplified car model with the underbody shaped as a Venturi nozzle. *International Journal of Vehicle Design* 2012; 58(1): 15-32.
11. White FM. *Fluid Mechanics*. New York: McGraw-Hill Print, 1986.
12. Baxendale, A. J., Graysmith, J. L., Howell, J. P., and Haynes, T. Comparisons between CFD and experimental results for the Ahmed reference model. In Proceedings of the Royal Aeronautical Society Conference on Vehicle Aerodynamics, London, UK, 18–19 July 1994, pp. 30.1–30.11.
13. Han T. Computational analysis of three-dimensional turbulent flow around a bluff body in ground proximity. *AIAA Journal* 1989; 27 (9): 1213-1219.
14. Chee Chok SC, Parameswaran RS, Sun MR and Gleason M. Numerical investigation of the effects of base slant on the wake pattern and drag of three-dimensional bluff bodies with a rear blunt end. *Journal of Wind Engineering and Industrial Aerodynamics* 1994; 51, 269-285.
15. Tunay T, Sahin B and Akilli H. Experimental and numerical studies of the flow around the Ahmed body. *Wind and Structures* 2013; 17: 515-535.
16. Yakhot V, Orszag SA, Thangam S et al. Development of turbulence models for shear flows by a double expansion technique. *Phys. Fluids A* 1992; 4(7): 1510–1520.

- 1
2
3
4
5
6
7
8 17. Serre E, Minguez M, Pasquetti R, et al. On simulating the turbulent flow around
9 the Ahmed body: A French–German collaborative evaluation of LES and DES.
10 *Computers & Fluids* 2013; 78: 10-23.
11
12 18. Minguez M, Pasquetti R and Serre E. High-order large-eddy simulation of flow over
13 the “Ahmed body” car model. *Physics of Fluids* 2008; 20, 095101.
14
15 19. Howard RJA and Pourquie M. Large eddy simulation of an Ahmed reference model.
16 *Journal of Turbulence* 2002; 3, N12.
17
18 20. Senior AE and Zhang X. The Force and Pressure of a Diffuser-Equipped Bluff Body
19 in Ground Effect. *Journal of Fluids Engineering* 2000; 123(1): 105-111.
20
21 21. Puglisevich LS and Page G. Large Eddy Simulation of the Flow Around a Diffuser-
22 Equipped Bluff Body in Ground Effect. In: *ASME International Mechanical*
23 *Engineering Congress and Exposition*, 2011. ASME.
24
25 22. Ruhrmann A and Zhang X. Influence of Diffuser Angle on a Bluff Body in Ground
26 Effect. *Journal of Fluids Engineering* 2003; 125(2): 332-338.
27
28 23. Humnic A and Humnic G. Computational Study of Flow in the Underbody
29 Diffuser for a Simplified Car Model. SAE Technical Paper 2010-01-0119, 2010.
30
31 24. Han T. Computational analysis of three-dimensional turbulent flow around a bluff
32 body in ground proximity. *AIAA Journal* 1989; 27(9): 1213-1219.
33
34 25. Han T, Hammond DC and Sagi CJ. Optimization of bluff body for minimum drag
35 in ground proximity. *AIAA Journal* 1992; 30 (4): 882-889.
36
37 26. Desai S, Lo C and George A. A Computational Study of Idealized Bluff Bodies,
38 Wheels, and Vortex Structures in Ground Effect. SAE Technical Paper 2008-01-0327,
39 2008.
40
41 27. Cooper K, Bertenyi T, Dutil G, Syms J et al. The Aerodynamic Performance of
42 Automotive Underbody Diffusers. SAE Technical Paper 980030, 1998.
43
44 28. Peddie KM and Gonzalez LF. CFD Analysis of the Diffuser of a Formula 3
45 Racecar. *Orbit: University of Sydney Undergraduate Research Journal* 2009; 1(1).
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60

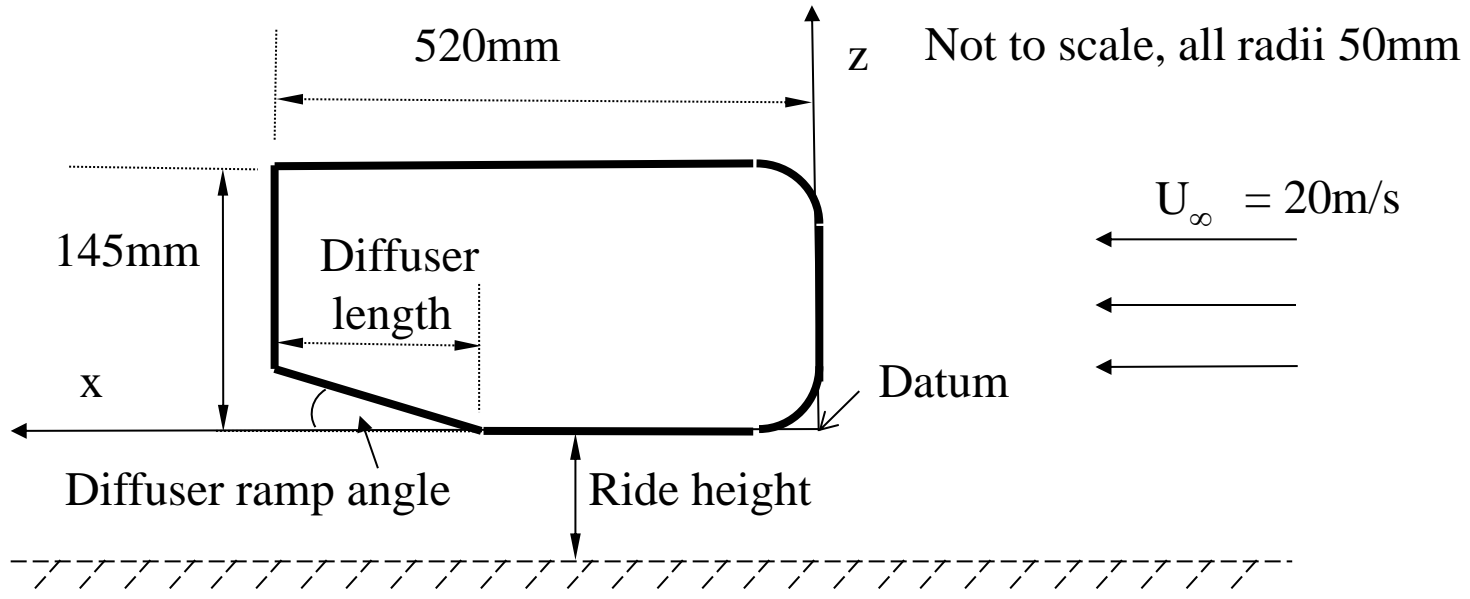
- 1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60
29. Jens I, Madsen, Shyy W and Raphael TH. Response Surface Techniques for Diffuser Shape Optimization. *AIAA Journal* 2000; 38(9): 1512-1518.
30. Jones WP and Launder BE. The prediction of relaminarization with a two-equation model of turbulence. *Int. J. Heat Mass Transfer* 1972; 15: 301–314.
31. ANSYS Inc. *ANSYS 14.5 Help*. ANSYS, Inc. Theory Release 8. 2012.

Figure Captions

- Figure 1 Inverted Ahmed model located in open jet wind tunnel [a\) picture b\) schematic](#).
- Figure 2 Contour Velocity plot using is k- ω SST turbulence model.
- Figure 3 Pressure distribution along the longitudinal centreline of Ahmed body.
- Figure 4 Detailed mesh example.
- Figure 5 Line plot of downforce (-Cl) against ride height for various ramp angles of long diffuser.
- Figure 6. Line plot efficiency ratio (-Cl/Cd) against ramp angle for various ride heights of long diffuser.
- Figure 7 Contour plot of downforce and efficiency optimization of short diffuser.
- Figure 8 Line plot of downforce efficiency (-Cl/Cd) for long and short diffusers at 10mm and 20mm ride heights.



1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41



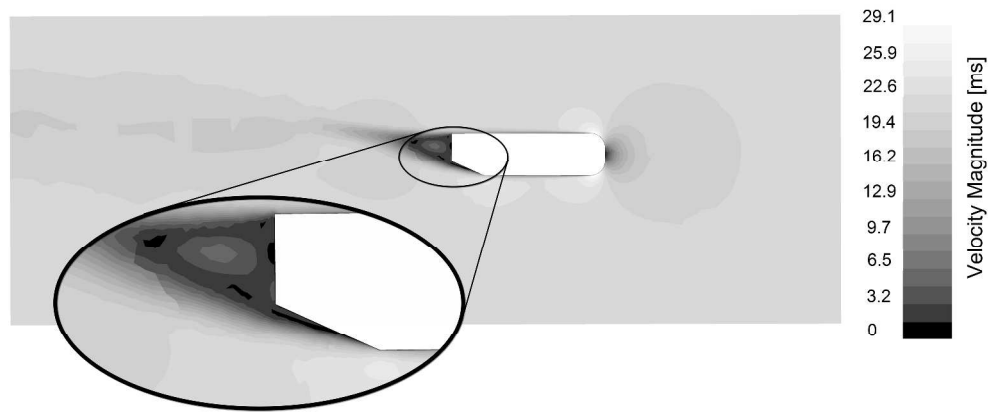


Figure 2 Contour Velocity plot using is k- ω SST turbulence model.

2016x867mm (96 x 96 DPI)

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60

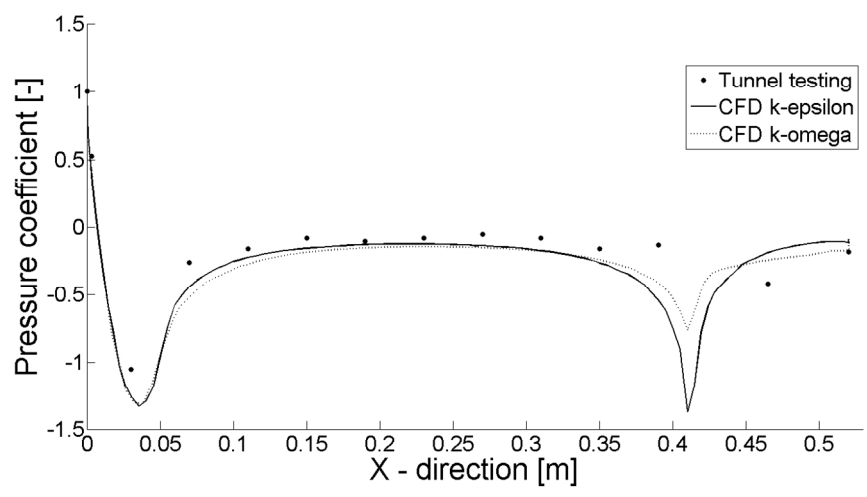


Figure 3 Pressure distribution along the longitudinal centreline of Ahmed body.

423x211mm (96 x 96 DPI)

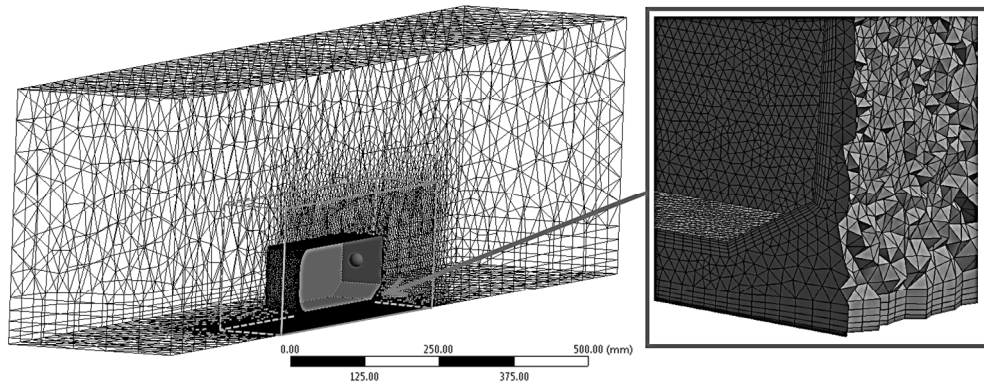
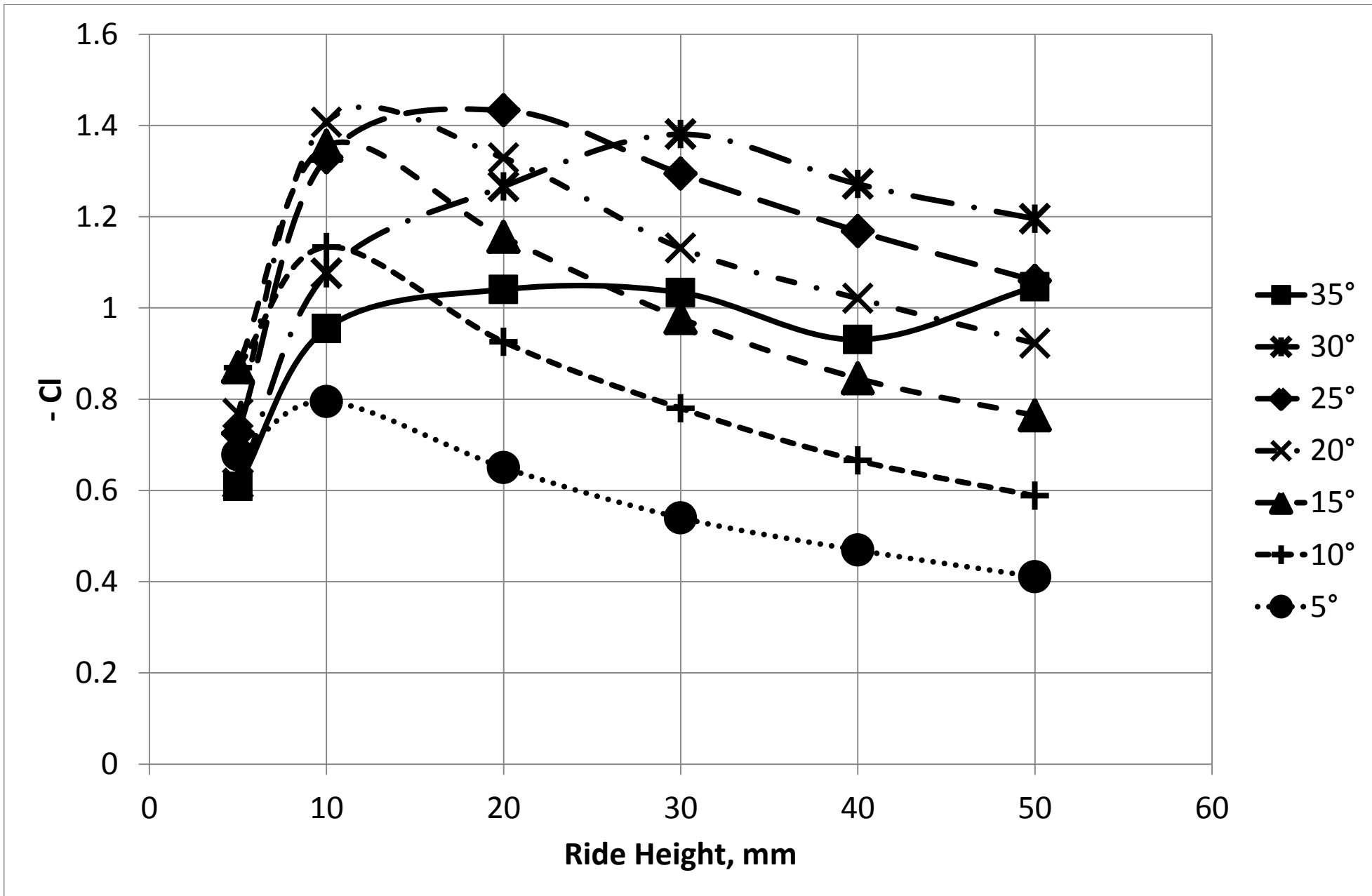


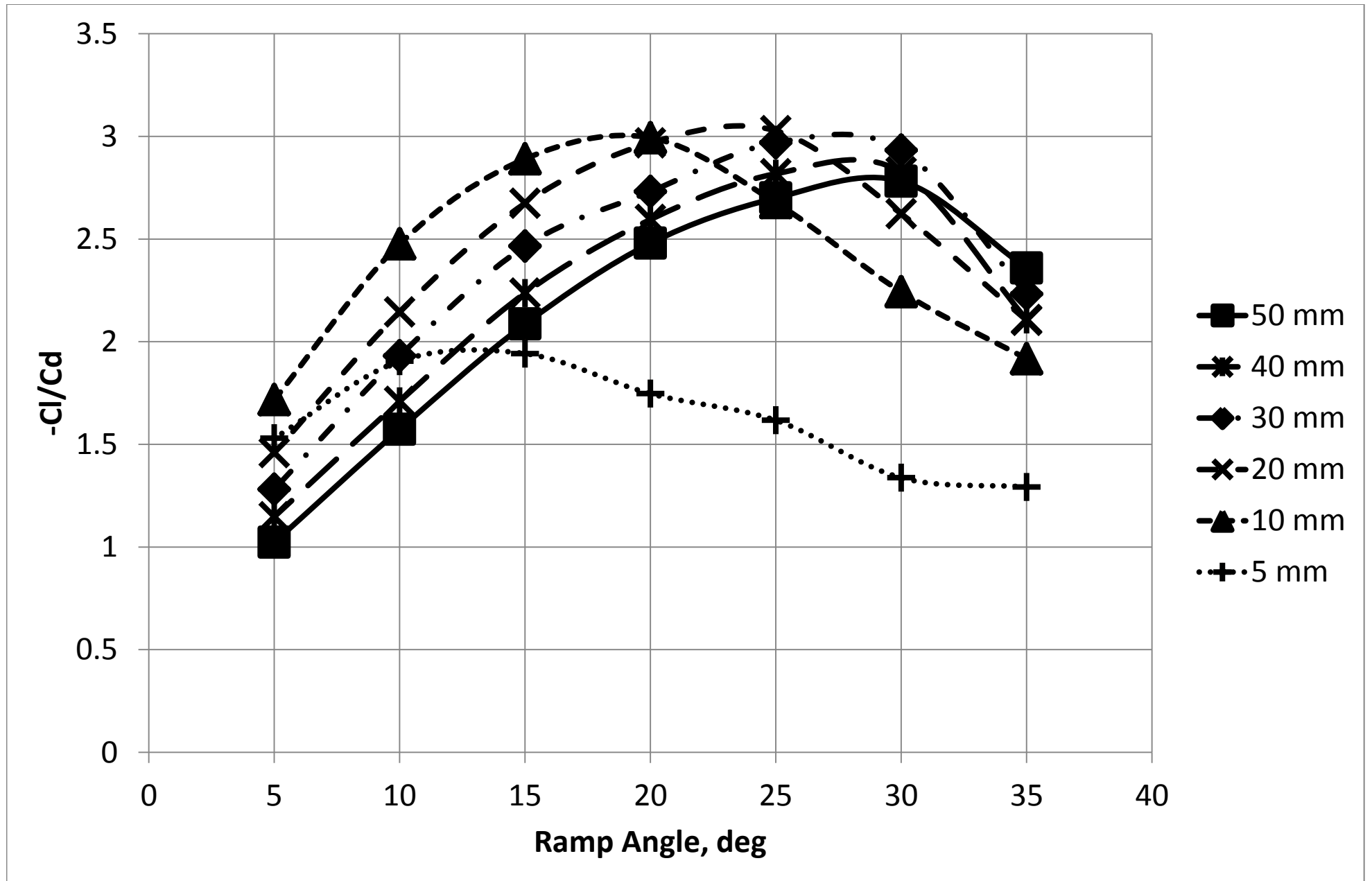
Figure 4 Detailed mesh example.

399x153mm (96 x 96 DPI)

Peer Review

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46





1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60

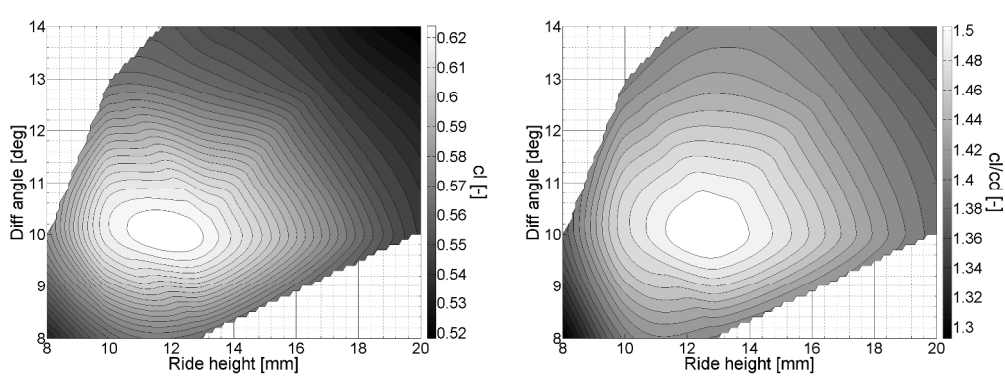
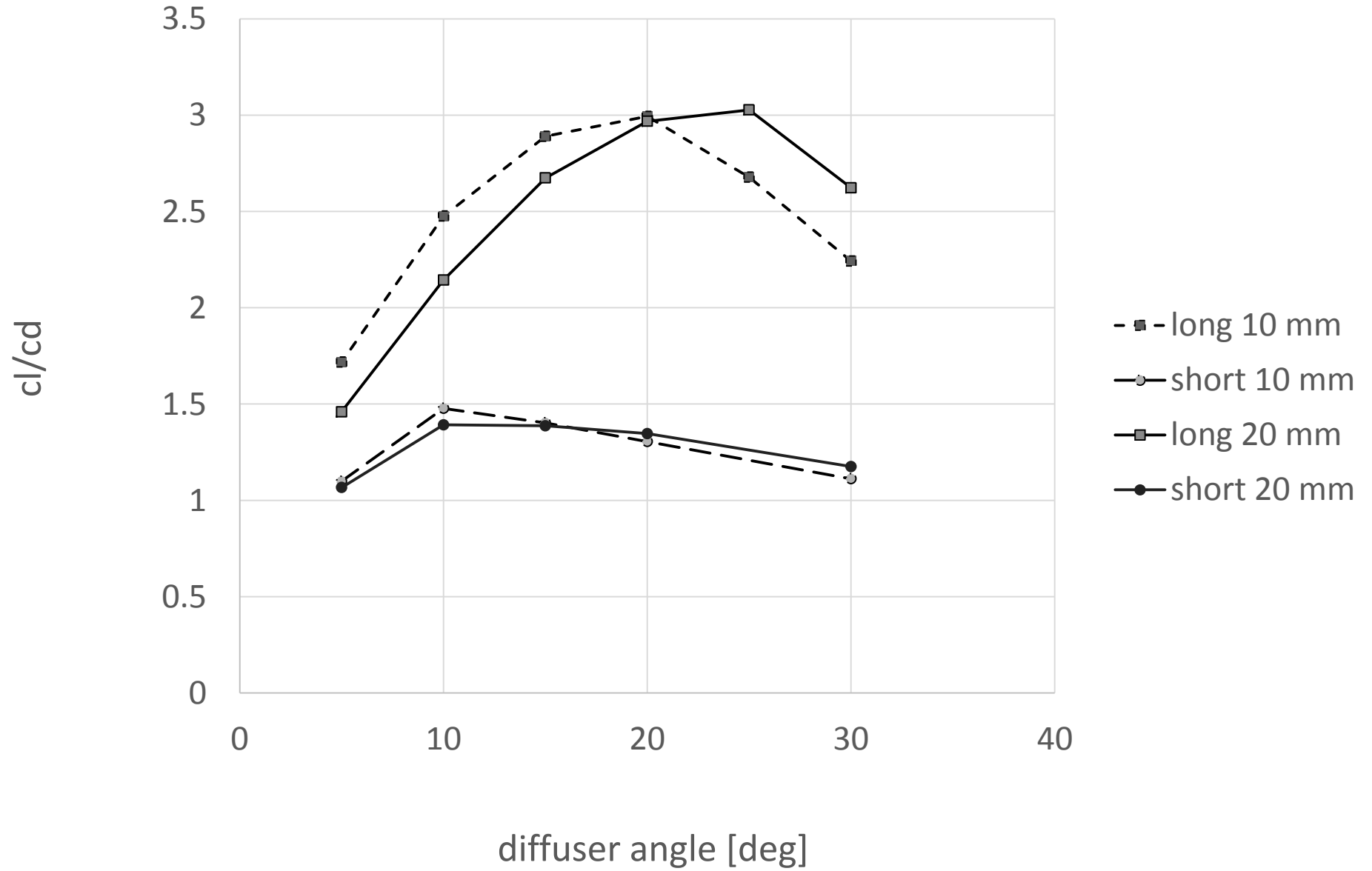


Figure 7 Contour plot of downforce and efficiency optimization of short diffuser.

830x317mm (96 x 96 DPI)

Peer Review



1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46