

CFD Assignment

by Christos David Zois

FILE	5363_CHRISTOS_DAVID_ZOIS_CFD_ASSIGNMENT_128450_955407285.D OCX (22)		
TIME SUBMITTED	27-JAN-2019 08:55PM (UTC+0000)	WORD COUNT	4679
SUBMISSION ID	99879531	CHARACTER COUNT	29538

Aerodynamic Vehicle Testing & Analysis

Christos-David Zois

P146231

University of Wales Trinity Saint David

School of Engineering

Motorsport Engineering

Advanced Computational Methods

M6X00408

19/01/2019

Abstract

In this assignment, a number of studies were performed, using the MIRA reference car, in order to create an optimum settings template for the use of CFD. To validate the results of these studies, the data obtained was correlated to data provided by the MIRA wind tunnel for the MIRA reference car. Finally, the effects of an aerodynamic device (diffuser) on the performance and balance of the MIRA reference car were tested.

Table of Contents

Aerodynamic Vehicle Testing & Analysis.....	i
Abstract.....	ii
1 Introduction	1
1.1 Aims.....	1
1.2 Objectives	1
2 Equipment / Apparatus	2
2.1 Computational Fluid Dynamics (CFD)	2
2.2 Computer Aided Design (CAD)	2
2.3 MIRA Reference Car	2
2.4 MIRA Wind Tunnel	3
3 Methodology & Results	3
3.1 Study Approach	3
3.2 Domain Study	4
3.2.1 Results & analysis	6
3.2.2 Blockage effects	7
3.3 Mesh Study.....	8
3.3.1 Results & Analysis.....	9
3.3.2 Density Box	10
3.3.3 Skewness	11
3.4 Turbulence Model Study.....	12
3.4.1 Results & Analysis.....	13
3.5 Prism Layer Study	14
3.5.1 Results & Analysis.....	14
3.6 Polyhedral Study	15
3.6.1 Results & Analysis.....	15
3.7 Optimum Settings Template	16
4 Initial to Final CFD Setup Comparison	18
4.1 Results & Analysis	19

4.2	Pressure Plot Comparison	20
4.3	Conclusion	20
5	Aerodynamic Device Study	21
5.1	Model	21
5.2	Setup	22
5.3	Results & Analysis	22
5.4	Conclusion	25
6	Recommended Further Work	26
7	References	27

Figure 1.	CFD and Validation processes	4
Figure 2.	Smallest domain tested (MIRA wind tunnel) (top), largest domain tested (bottom) .	5
Figure 3.	Lift Coefficient against domain size	6
Figure 4.	Drag coefficient against domain size	7
Figure 5.	Case comparison of pressures on car between small and large domain sizes	8
Figure 6.	Lift coefficient against mesh scale factor	9
Figure 7.	Drag coefficient against mesh scale factor	10
Figure 8.	2D representation of a non-skewed (left) and a skewed (right) element	11
Figure 9.	Mesh quality comparison against mesh scale factor	11
Figure 10.	Wheel blocks	12
Figure 11.	Turbulence model comparison to MIRA wind tunnel data provided	13
Figure 12.	Lift coefficient against number of prism layers	14
Figure 13.	Drag coefficient against number of prism layers	15
Figure 14.	Boundary Condition display in FLUENT	17
Figure 15.	Pressure coefficient comparison of MIRA wind tunnel data to CFD data	20
Figure 16.	MIRA car diffuser	21
Figure 17.	Velocity contours on symmetry plane	22
Figure 18.	Pressure contours on symmetry plane	23
Figure 19.	Streamlines	24
Figure 20.	Pressure contours on rear deck	24
Figure 21.	Pressure coefficients comparison between geometry with & without diffuser	25

1 Introduction

An object travelling through free stream air is generally thought to produce two types of forces. Aerodynamic drag and aerodynamic lift.

A vehicle's forward motion will set, the otherwise still air, into motion. This causes small pressure differences around the vehicle, the joint effect of which are responsible for the aerodynamic loads.

Thanks to times we live in, computers have the ability to calculate and predict these aerodynamic forces with the use of Computational Fluid Dynamics, or also known as CFD.

1.1 Aims

The aims of this assignment are

- To use computational fluid dynamics to assess aerodynamic performance.
- To use theoretical and computational methods for the formulation and solution of a vehicle aerodynamic problem.

1.2 Objectives

The objectives of this assignment are:

- To create a settings template for CFD that is valid, using the provided MIRA wind tunnel data for the MIRA reference car to achieve it.
- To alter the MIRA reference car model using a common aerodynamic device of choice. Analyse and understand the effect this device has on the aerodynamic performance of the vehicle.

2 Equipment / Apparatus

The following tools were used to complete this assignment.

2.1 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics, as the name implies, is a commonly used method to provide simulations of fluid flow problems, thanks to the processing capability that modern day computers have, to perform a myriad of calculations in a short space of time. (McBeath, 2015)

Air is a fluid, hence why we will be using CFD for the aerodynamic problems at hand. Specifically, the program used is ANSYS Workbench, which incorporates applications such as Design Modeller (Domain), ICEM Meshing (Mesh) and Fluent (Boundary Conditions & Simulation), all of which are necessary to perform the studies needed to complete this assignment.

2.2 Computer Aided Design (CAD)

As the name states, CAD, is the design process of a 3D model, aided by today's highly capable computers.

The MIRA reference car model will be modified using Solidworks (3D CAD program), to incorporate the selected aerodynamic device for analysis later in the assignment.

2.3 MIRA Reference Car

A simplified car shape, based on the proportions of the 1980s "family sized" cars, designed by G. W. Carr. The MIRA reference car has been used throughout the years, mainly for the refinement of automotive wind tunnels, but has also been used in the evaluation of ground simulation and the investigation of blockage correction techniques. (Le Good & Garry, 2004)

The MIRA reference car will be used throughout this entire assignment.

2.4 MIRA Wind Tunnel

The MIRA wind tunnel is the only full-scale aerodynamic wind tunnel in the UK. It is a closed test section, open return wind tunnel.

Test section dimensions are:

15m length x 7.9m wide x 4.4m high

Which evaluates to a cross sectional area of $35m^2$

(MIRA, 2019)

The data provided from the wind tunnel will be used to correlate and validate the findings of this assignment's studies.

3 Methodology & Results

In this section, the methodology applied to find the optimum settings for future CFD runs will be explained. The results found will be analysed and an optimum settings template will be created.

3.1 Study Approach

In order to create an optimum settings template that will be used for future CFD runs, these settings must meet some criteria.

- Altering the specific setting parameter by a small amount must not affect the results obtained.
- The limitations of the processing power available need to be taken into consideration i.e. the mesh cannot be ultra-fine in the whole domain as that would lead to a substantial number of cells in the mesh which would require a lot of processing power to solve. So, the mesh will only be refined in areas that are critical, such as the wake of the vehicle.

An initial simulation was run using the domain size of the MIRA wind tunnel. The rest of the settings were setup through prior experience and research. Settings and results of this can be found in paragraph **Error! Reference source not found.**

Following this, certain studies were conducted to try and find the settings that would meet the aforementioned criteria.

Finally, the optimum settings template will be created and a final run using these settings will be performed and analysed.

Below in Figure 1, is a representation of the CFD process and the validation approach of it. Finally, in the top right corner of the image, the MIRA reference car model used is displayed.

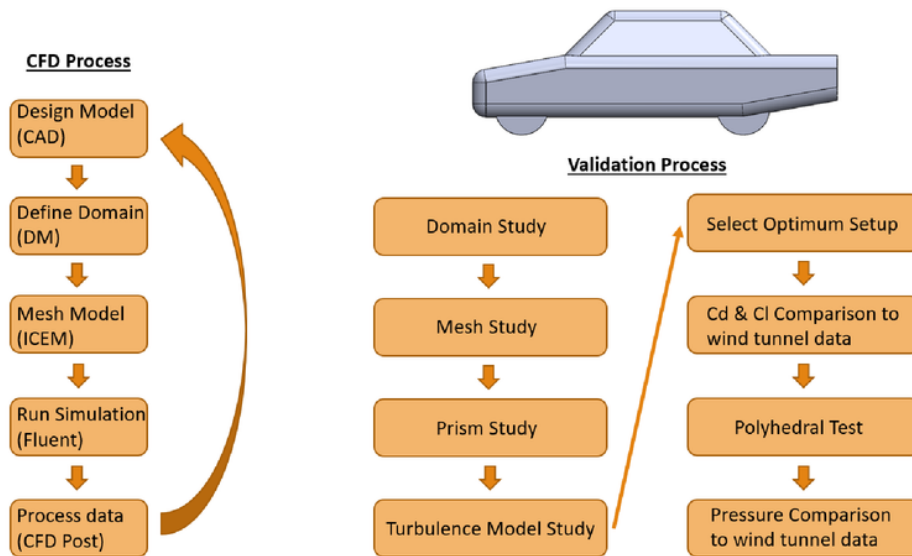


Figure 1. CFD and Validation processes

3.2 Domain Study

The first study conducted was the domain study. The initial domain size was:

15m length x 4m wide x 4.4m high

These dimensions are based on the MIRA wind tunnel proportions. The only difference is the width is halved, as only half of the model will be simulated because it is a symmetrical shape. This way, the same results can be obtained with half the computational effort.

Opening the possibilities of completing the studies in less time, or being able to run a finer, more detailed, mesh, as double the processing power is available to use.

The MIRA car was placed 4m (1 car length) behind the inlet, leaving approximately 2 car lengths behind, for the wake structure to develop.

From there, the dimensions of the domain were altered incrementally as explained below:

Initial length x 1.5

Initial width x 1.35

Initial height x 1.35

And the distance of the car to the inlet was kept at a proportion of 1 ahead to 2 behind, i.e. if there the length of the domain is 40m, the car will be placed 12m behind the inlet leaving 24m behind the vehicle for the wake to develop ($12\text{m}+24\text{m}+4\text{m}=40\text{m}$).

Below in Figure 2, the top part of the image displays the smallest domain tested which had the proportions and dimensions of the MIRA wind tunnel. The lower part of the image displays the largest domain tested. The exact dimensions can be seen in Table 1.

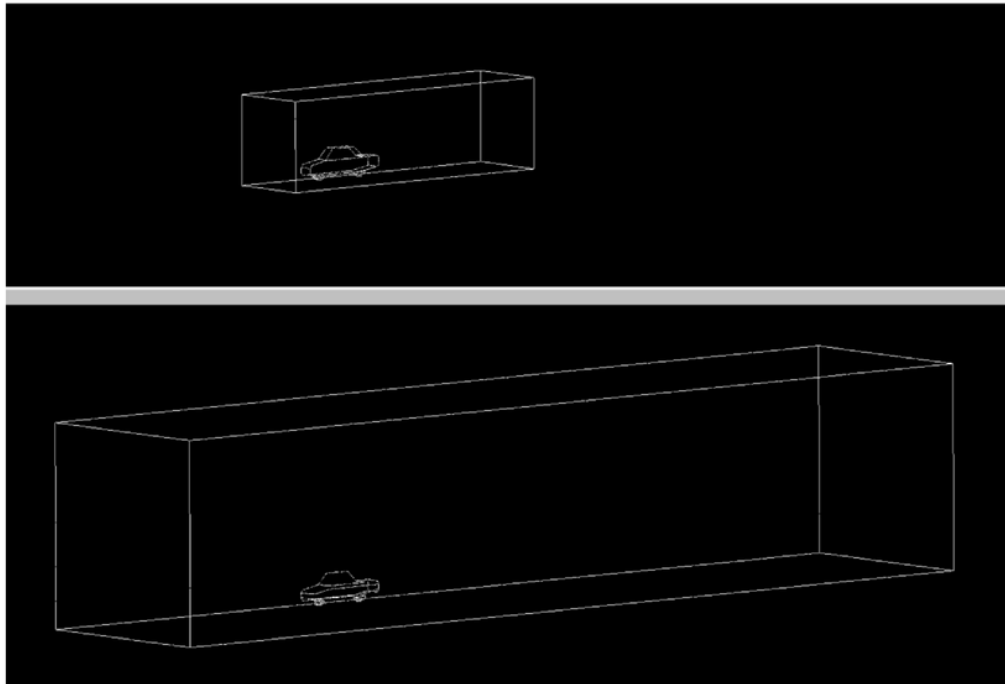


Figure 2. Smallest domain tested (MIRA wind tunnel) (top), largest domain tested (bottom)

3.2.1 Results & analysis

Table 1

	Domain Study			
No.	Size (m)	Distance to inlet (m)	Cd	Cl
3	50.6 x 10 x 11	15.5	0.260	-0.080
2	33.8 x 7.3 x 8	10	0.264	-0.083
1	22.5 x 5.4 x 6	6.2	0.275	-0.128
MIRA	15 x 4 x 4.4	4	0.301	-0.189

Note: Results of drag and lift coefficients aren't compared to the provided wind tunnel data yet, as this is a series of studies to prove a settings template. A comparison to the provided data will be performed later in this assignment, when the simulation with the optimum settings has produced data.

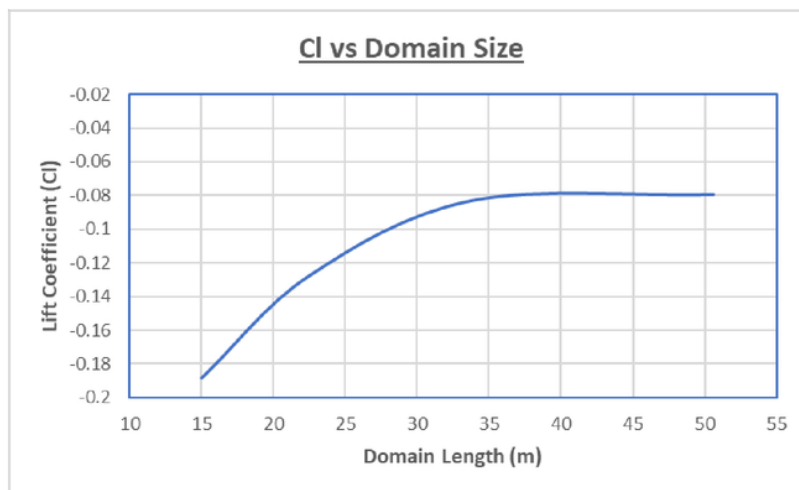


Figure 3. Lift Coefficient against domain size

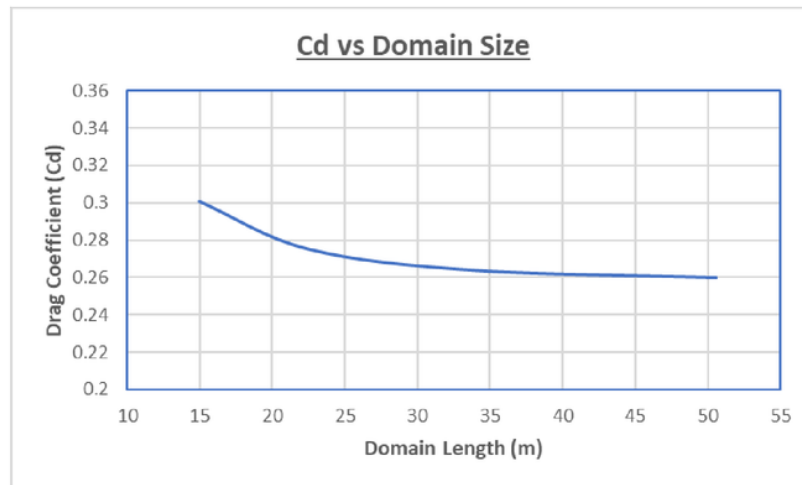


Figure 4. Drag coefficient against domain size

Looking at Figure 3 and Figure 4, it is obvious that the results are not affected after an increasing domain length of approximately 35m.

Furthermore, there is a significant increase in both the drag and lift coefficients when comparing the domain size of the MIRA wind tunnel run and the largest domain tested.

To try and keep computational effort to its lowest, the domain size selected for the settings template was:

35m length x 7.5m wide x 8m high, with a 10.5m distance of the car to the inlet.

These dimensions coincide with what Lanfrit suggests, which is 3 car lengths distance to the inlet, 5 car lengths behind the car, 3 car heights high and 3 car widths wide (Lanfrit, 2005).

This equates to the following dimensions:

42m length x 6.5m wide x 5.6m high, with a 12.6m distance of the car to the inlet.

Differences do exist between the two, but overall the dimensions and proportions are pretty similar.

3.2.2 Blockage effects

The obvious increase in both the drag and lift coefficients, mentioned in the paragraph above, is due to blockage effects acting in the smaller domains. The cross-sectional area of the domain at 35m long is:

7.5m wide x 8m high = $60m^2$

While the cross-sectional area of the MIRA reference car (half of the car) is $0.92m^2$.

So, the blockage ratio is:

$$\frac{0.92}{60} = 0.015 \text{ (1.5\%)}$$

A ratio which is well within the recommended allowable limits, that range from 7.5% to 10%, another reason why the aforementioned domain size was selected. (Katz, 2006) (Hucho, 1998)

Finally, a case comparison of the 2 runs was performed using the post processing feature in ANSYS Workbench. The pressure contours on the MIRA reference car were plotted and then the difference in pressures between the 2 runs can be seen below.

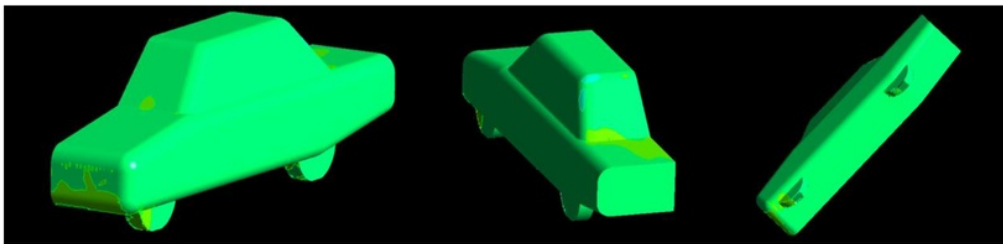


Figure 5. Case comparison of pressures on car between small and large domain sizes

The green colour suggests the pressure difference in these areas was minimal. The yellow colour portrays that there is an increase in pressure at the front of the car, ahead of the front wheel and on the rear deck, hence the increase in drag and lift when blockage occurs.

3.3 Mesh Study

The next study performed was a mesh study. The mesh used was an unstructured mesh (tetrahedrals) and not a structured mesh (hexahedrals), because of the complex shape of the body. The meshing method used was the Robust Octree method and prism layers were added once the initial meshing was complete.

The simulations from here on out kept the original settings (paragraph **Error! Reference source not found.**), apart from the domain size which was adjusted based on paragraph 3.2.

Again, Fluent Germany suggests some optimum meshing sizes for automotive external aerodynamics, and these are what were used initially:

Domain mesh: 1000mm

Density Box mesh: 100mm

Vehicle Surface mesh: 10mm

The study consisted of using the Scale Factor tool within ICEM meshing. Scaling incrementally the initial mesh from fine to coarse.

3.3.1 Results & Analysis

Table 2

Mesh Study			
Scale Factor	No. of Elements	Cd	Cl
0.85	10486055	0.263	-0.086
0.9	8913963	0.263	-0.086
0.95	7795683	0.264	-0.085
1	6641163	0.264	-0.083
1.1	5112861	0.264	-0.083
1.2	3903076	0.267	-0.082
1.3	3162319	0.279	-0.074
1.5	2132616	0.286	-0.069
2	966803	0.302	-0.057
3	348005	0.327	-0.040

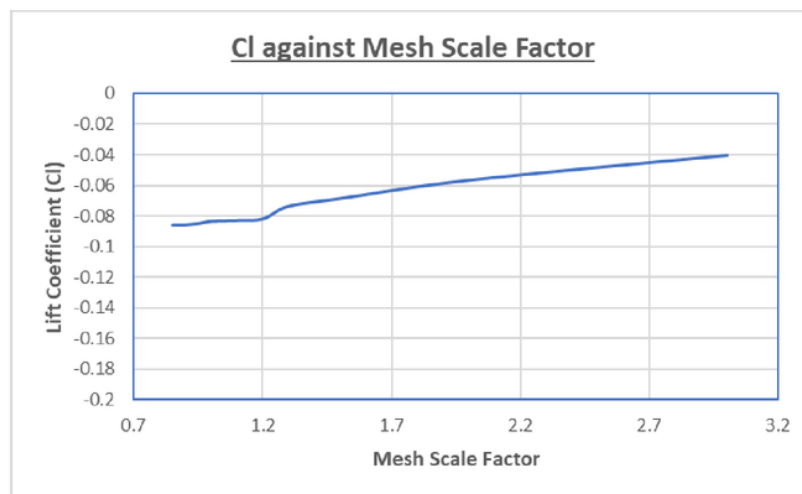


Figure 6. Lift coefficient against mesh scale factor

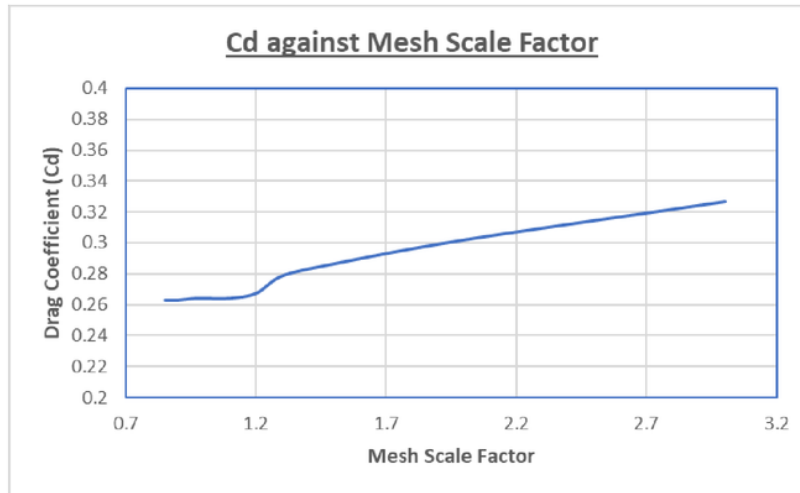


Figure 7. Drag coefficient against mesh scale factor

A scale factor of 0.9 means it will scale down the whole mesh, i.e. if the Domain mesh was 1000mm, after applying the scale it would be 900mm. If the scale was 1.2, then the Domain mesh would be scaled to 1200mm and so on.

Looking at Table 2, the row with a scale factor of 1 was the initial run using the settings mentioned above.

Observing Figure 6 and Figure 7, it is clear that the mesh can be scaled up to a factor of 1.1 without affecting the results. It was decided that the following mesh settings will be used from now on in the assignment:

Domain mesh: 1100mm

Density Box mesh: 110mm

Vehicle Surface mesh: 11mm

3.3.2 Density Box

To capture the complex flow around the vehicle i.e. its wake, it is generally suggested the mesh be refined in these areas. A density box was created in all the simulations during this assignment. Its dimension remained unaltered throughout.

Density box dimensions:

10.5m length x 1.6m wide x 2.1m high

Extending half a car length in front of the model, a whole car length behind, half a cars width to the side and half a cars height above. (Piechna, et al., 2009)

3.3.3 Skewness

Once the mesh is generated its quality has to be checked, to determine if it will produce valid or not results.

One of the main mesh quality metrics is skewness. Skewed cells can affect the outcome of a simulation, hence producing misleading results. Below in Figure 8, a tetrahedral element is displayed in 2D. On the left is a non-skewed element, where as on the right is what a skewed element would look like.

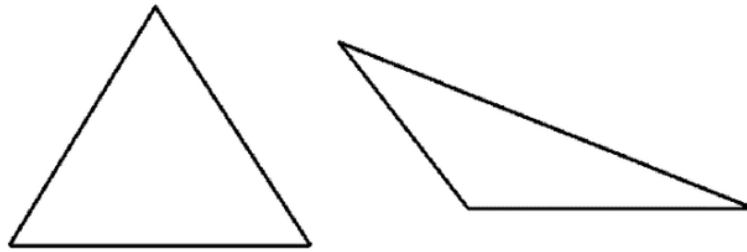


Figure 8. 2D representation of a non-skewed (left) and a skewed (right) element

In ICEM meshing there is a feature to check mesh skewness. Below in Figure 9, is an example of how mesh quality is affected by the mesh scale factor (indicated in the bubbles on the left).

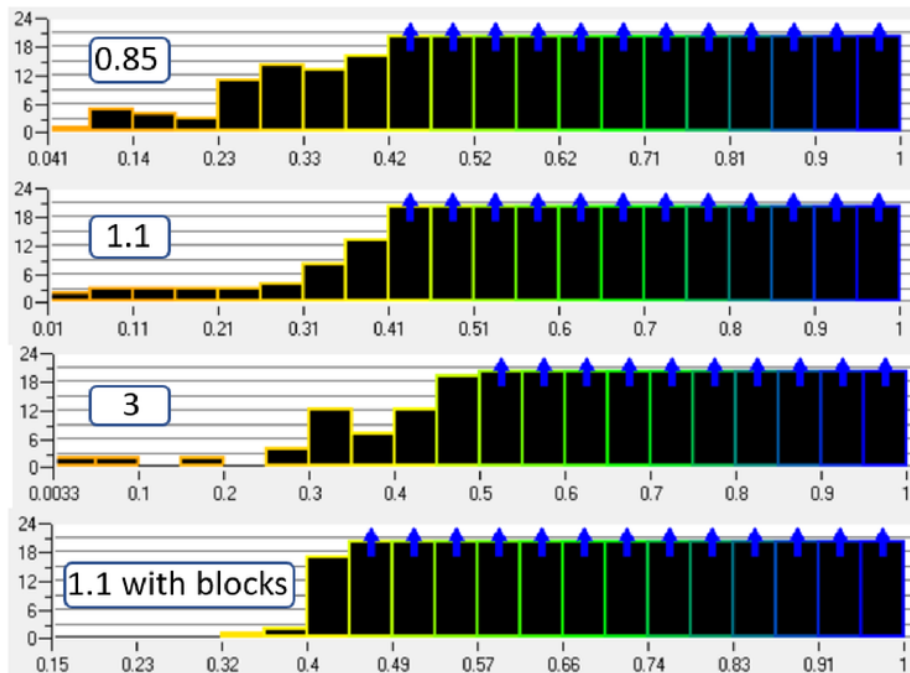


Figure 9. Mesh quality comparison against mesh scale factor

Note: The x axes on the above image, are similar but not identical.

The finest (0.85), the coarsest (3) and the optimum (1.1) mesh scale factors were chosen as part of the example above. Orange indicates skewed cells, while the blue, good quality ones. It seems like the finest mesh has the greatest number of skewed cells, but that is probably due to the increased number of cells compare to the rest of the meshes. In the final mesh examined, skewed elements were almost non-existent. That is because blocks were added to the wheels of the MIRA reference car, as shown in Figure 10 below. This prevented the creation of skewed elements between the floor and the wheels.

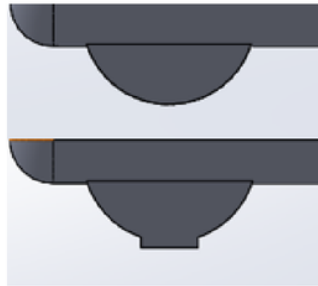


Figure 10. Wheel blocks

3.4 Turbulence Model Study

A fundamental part of CFD are the fundamental equations behind the calculations performed during a simulation of fluid flow, taking into account the conservation of Mass, Momentum and Energy in all 3 dimensions. The flow around a car is pretty complex, where substantial parts of the flow are turbulent. To make it even more complex, the effects of viscosity, the development of a boundary layer and flow separation all need to be considered too. This is done thanks to the Navier-Stokes equations that deal with fluid momentum. Turbulence Models are what help compile and solve all these equations during a simulation. (McBeath, 2015)

Depending on the situation, there usually is an appropriate turbulence model for it. For our case, Lanfrit suggests using the K-epsilon Realizable Model, which was used for our initial run (Lanfrit, 2005). A number of different turbulence models were tested. Near wall treatment in the form of Non-Equilibrium wall functions were used for the two K-epsilon models tested. The rest of the models were tested as default.

3.4.1 Results & Analysis

Table 3

MIRA windtunnel data provided	
Cd	Cl
0.296	-0.085

Table 3 shows the wind tunnel data provided by MIRA for the MIRA reference car. This data is compared to the results of this turbulence model study in Table 4 and Figure 11.

Table 4

Turbulence Study			Difference from wind tunnel data	
Turbulence Model	Cd	Cl	Cd	Cl
Ke-Realizable	0.264	-0.083	0.032	-0.001
K-KI-w	0.441	0.194	-0.145	-0.278
Ke-RNG	0.273	-0.012	0.024	-0.073
Kw-SST	0.290	-0.023	0.006	-0.061
Spalart-Almaras	0.308	-0.034	-0.011	-0.051
Transition-SST	0.293	-0.025	0.003	-0.059

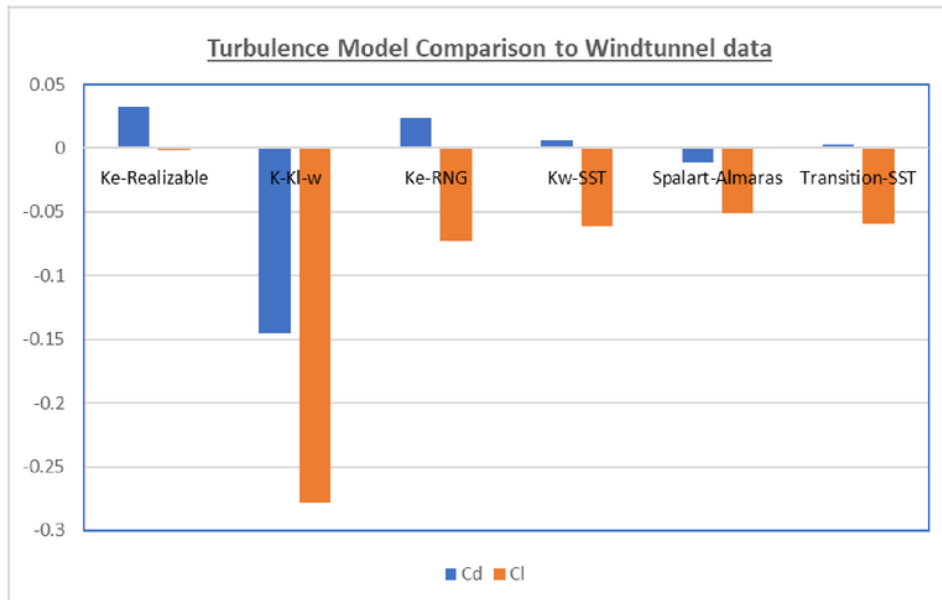


Figure 11. Turbulence model comparison to MIRA wind tunnel data provided

Observing Figure 11, most turbulence models produced similar results to the provided data, apart from the K-KI-w model. The model that came closest overall to the wind tunnel data and predicted the lift coefficient to a negligible difference was the one used initially, the K-epsilon Realizable.

3.5 Prism Layer Study

Prism layers are layered elements that are extruded from surfaces that are affected by viscosity, surfaces on which a boundary layer is formed. These elements help capture the boundary layer better than what several tetrahedrals could.

Fluent Germany suggests the creation of 5 prism layers with a growth rate of 20% between them. Prism layers were added to the entire car and to the domains static floor. These settings were used for the initial run. 1-8 prism layers were tested in this study. (Lanfrut, 2005)

3.5.1 Results & Analysis

Table 5

Prism Layer Study		
No. of Prisms	Cd	Cl
1	0.277	-0.128
2	0.281	-0.131
3	0.273	-0.112
4	0.273	-0.120
5	0.264	-0.083
6	0.264	-0.081
7	0.264	-0.083
8	0.264	-0.082

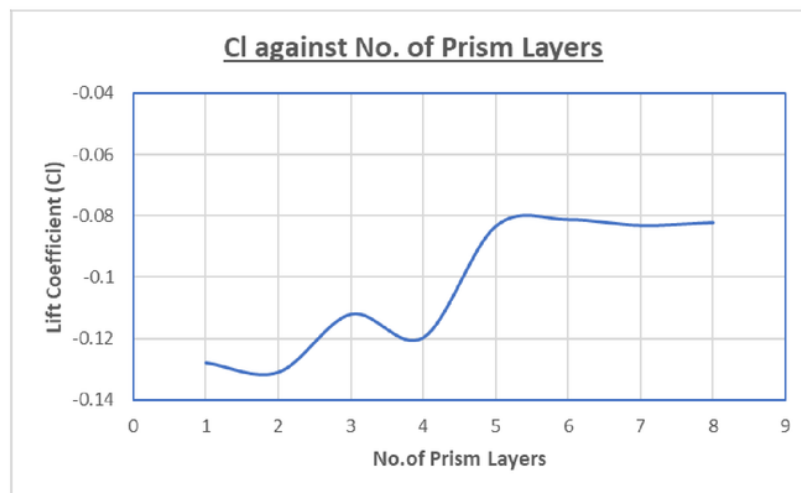


Figure 12. Lift coefficient against number of prism layers

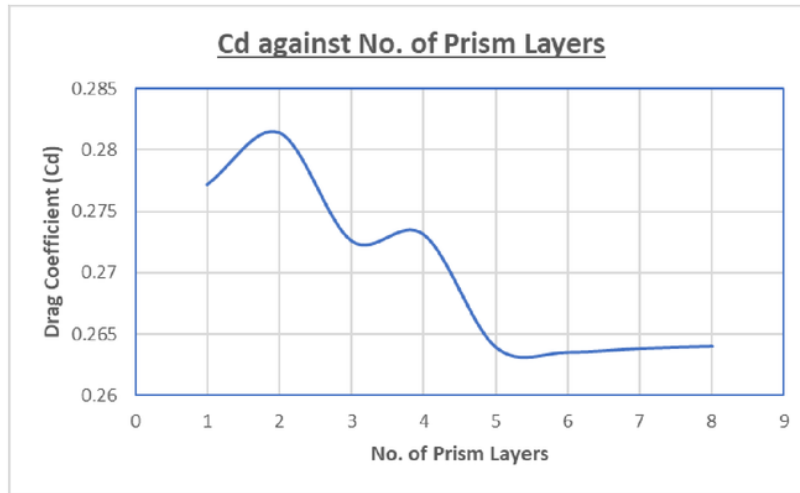


Figure 13. Drag coefficient against number of prism layers

Inspecting the graphs in Figure 12 and Figure 13, it is noticeable that the coefficients of lift and drag stop varying when using 5 or more prism layers. So, 5 prism layers will be used as the optimum setting, which is what Lanfrit suggested initially.

3.6 Polyhedral Study

A polyhedral mesh consists of combining 5 tetrahedral elements into 1, reducing the computational power required to solve the fluid flow problem, but can lead to inaccuracies. A polyhedral mesh was created and compared to the optimum parameter setup based on the studies performed so far.

3.6.1 Results & Analysis

Table 6

Polyhedral Study				
	Cd	Cl	Delta Cd (%)	Delta Cl (%)
Polyhedral	0.265	-0.081	-0.6	3.0
Optimum setup	0.264	-0.083		

Studying Table 6, using the polyhedral mesh predicted the coefficients of lift and drag with a 3% and a 0.6% difference respectively, to those of the optimum setup. It solved the problem in approximately a third of the original time. So, for the almost negligible difference in producing results, it is probably worth using a polyhedral mesh if you are pushed for time. What needs to be considered though is that the MIRA reference car is a pretty simple car model compare to a race car for example, around which the flow will be significantly more complex, that could lead to the inaccuracies using a polyhedral mesh increase. It all comes down to finding that balance between accuracy and time-efficient processing.

3.7 Optimum Settings Template

From the studies performed above, a settings template with the goal of attaining optimum results for future simulations was created.

Domain Dimensions (Design Modeller):

35m length x 7.5m wide x 8m high, with a 10.5m distance of the car to the inlet

Mesh Setup (ICEM):

Domain mesh: 1100mm

Density Box mesh: 110mm

Vehicle Surface mesh: 11mm

Density Box Dimensions:

10.5m length x 1.6m wide x 2.1m high

Number of Prism layers:

5

Boundary Conditions (Fluent):

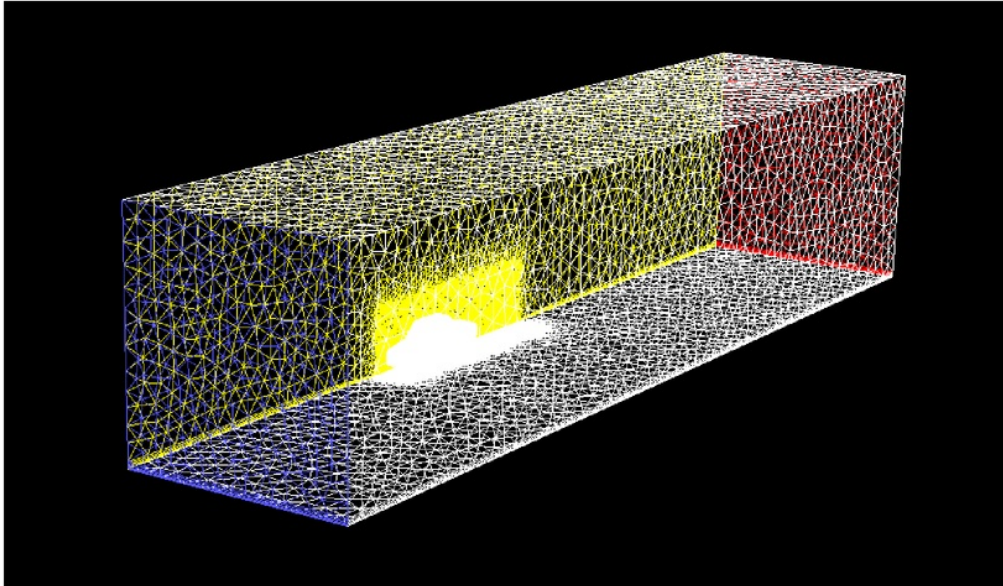


Figure 14. Boundary Condition display in FLUENT

In Figure 14 above, the blue grid resembles the velocity inlet (20 m/s), the red grid the pressure outlet, the yellow grid the symmetry plane and the white grid the walls (car, floor, side, top). The density box and the prism layers on the floor are also visible.

Turbulence Model:

K-epsilon Realizable, Non-equilibrium wall functions

Reference Values:

Frontal Area of MIRA car (half): 0.92 m^2

Density: 1.225 kg/m^3

Velocity: 20 m/s

Viscosity: $1.7894 \times 10^{-5} \text{ kg/ms}$

Solution Methods:

Pressure Velocity Coupling: Coupled

Gradient: Least Squares Cell Based

Pressure: Standard

Momentum: Second Order Upwind

Turbulent Kinetic Energy: Second Order Upwind

Turbulent Dissipation Rate: Second Order Upwind

Solution Controls:

Flow Courant Number: 50

Explicit Relaxation Factors: Momentum 0.25, Pressure 0.25

Under-Relaxation Factors:

Density: 1

Body Forces: 1

Turbulent Kinetic Energy: 0.8

Turbulent Dissipation Rate: 0.8

Turbulent Viscosity: 1

Initialization: Standard

The settings mentioned above were used to complete the final simulation of the original MIRA reference car model.

4 Initial to Final CFD Setup Comparison

Differences between initial and final runs:

Domain size changed from this,

15m length x 4m wide x 4.4m high, with a 4 m distance of the car to the inlet,

to this,

35m length x 7.5m wide x 8m high, with a 10.5m distance of the car to the inlet.

Mesh setup changed from this,

Domain mesh: 1000mm

Density Box mesh: 100mm

Vehicle Surface mesh: 10mm

to this,

Domain mesh: 1100mm

Density Box mesh: 110mm

Vehicle Surface mesh: 11mm

Apart from those 2 changes the rest of the settings remained the same.

4.1 Results & Analysis

Table 7

	Cd	Cl	Delta Cd	Delta Cl
MIRA	0.296	-0.085		
CFD Initial	0.301	-0.189	0.004	0.104
CFD Final	0.264	-0.083	0.032	0.001

In Table 7 above, the initial and final CFD runs for the MIRA car were compared to the MIRA wind tunnel data provided. The initial CFD run produced very accurate results for the drag coefficient compare to the provided data, but deviated by 0.1 regarding the lift coefficient, which is a significant amount when referring to coefficients on a common family car. The final CFD run deviated by 0.032 Cd compare to the wind tunnel data, but produced an almost identical Cl.

So, overall, the Final CFD setup was more accurate at producing results, but the domain size did not match the size of the MIRA wind tunnel, as did the initial CFD setup. Perhaps the initial CFD setup could produce closer results to the wind tunnel data using some wind tunnel correction equations with the data acquired.

4.2 Pressure Plot Comparison

The final CFD run and wind tunnel data were used to plot and compare the pressure along the length of the MIRA reference car, above and below.



Figure 15. Pressure coefficient comparison of MIRA wind tunnel data to CFD data

The blue colour represents the pressure on the upper side of the car and the orange the pressure on the lower side. The dotted lines represent the MIRA wind tunnel data, while the solid lines the CFD data.

There are some slight deviations at points, but overall the trends and values seem pretty similar. The differences observed could be due to the lower number of pressure taps in a wind tunnel test compare to a CFD test.

4.3 Conclusion

To conclude, the final and optimum CFD setup, based on the studies performed above, produced very similar results and trends compare to the MIRA wind tunnel data provided.

5 Aerodynamic Device Study

In this part of the assignment, an aerodynamic device was added to the model. The device investigated was a diffuser. A diffuser is thought to be one of the most efficient aerodynamic devices when working correctly. Its role is to allow the high velocity air travelling under the car to slow down again, thanks to the increasing cross-sectional area. This allows the kinetic energy to be recovered as efficiently as possible. This permits the increase of static pressure back up to ambient and consequently the decrease of velocity, to try and match that of the free stream. Furthermore, on average, the static pressure in the diffuser is lower than ambient, hence generating some downforce as a secondary effect. (McBeath, 2015)

5.1 Model

The diffuser added to the MIRA reference car can be seen in Figure 16 below.

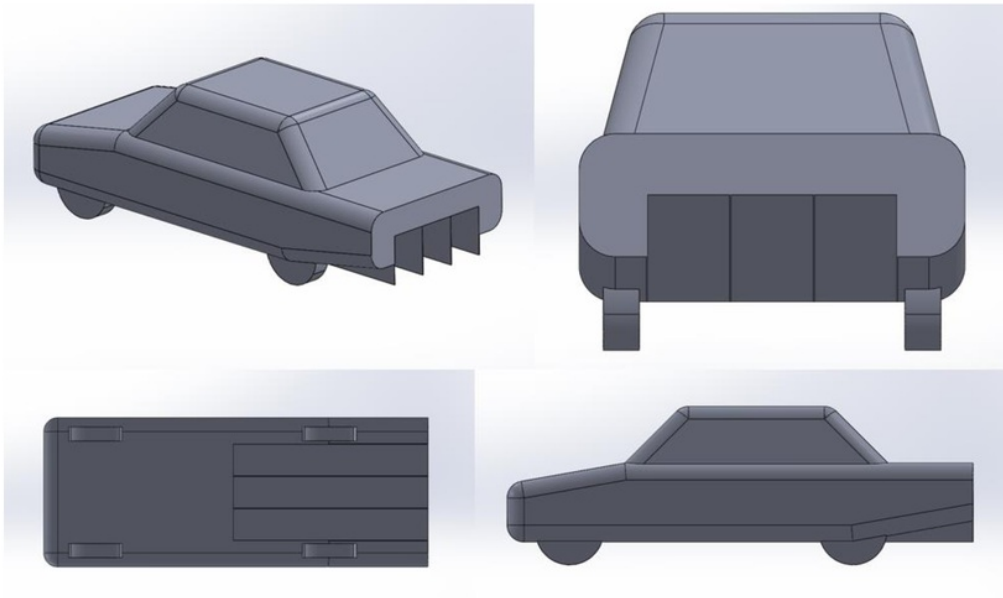


Figure 16. MIRA car diffuser

It's a radiused diffuser, meaning that the transition from the flat floor to the steepest angle is smooth and does not occur immediately after the kick point. The kick point is halfway along the length of the car. Two strakes were added to help organise the flow through the diffuser and help keep the flow attached throughout, as it is a pretty aggressive diffuser.

5.2 Setup

The simulation was set up identically to the final CFD simulation of the validation process (paragraph 3.7), aside from updating the model.

5.3 Results & Analysis

Table 8

	DRAG (N)	DOWNFORCE (N)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	Drag Cd	Lift Cl
Final CFD run	118.91	37.53	0.32	40.63	15.25	22.28	0.264	-0.083
MIRA Diffuser	144.37	90.53	0.63	29.49	26.70	63.83	0.320	-0.201

Observing Table 8, it is clear that the diffuser is in effect. Although the drag increased by approximately 21%, the downforce generated increased by 53N (141%), leading to an overall increase in aerodynamic efficiency of 97%. Furthermore, the aerodynamic balance shifted rearwards, from approximately 40-60 (40% front 60% rear) to 30-70.

Note: In the following figures, the top car is the original MIRA reference car and the lower one is the modified car.

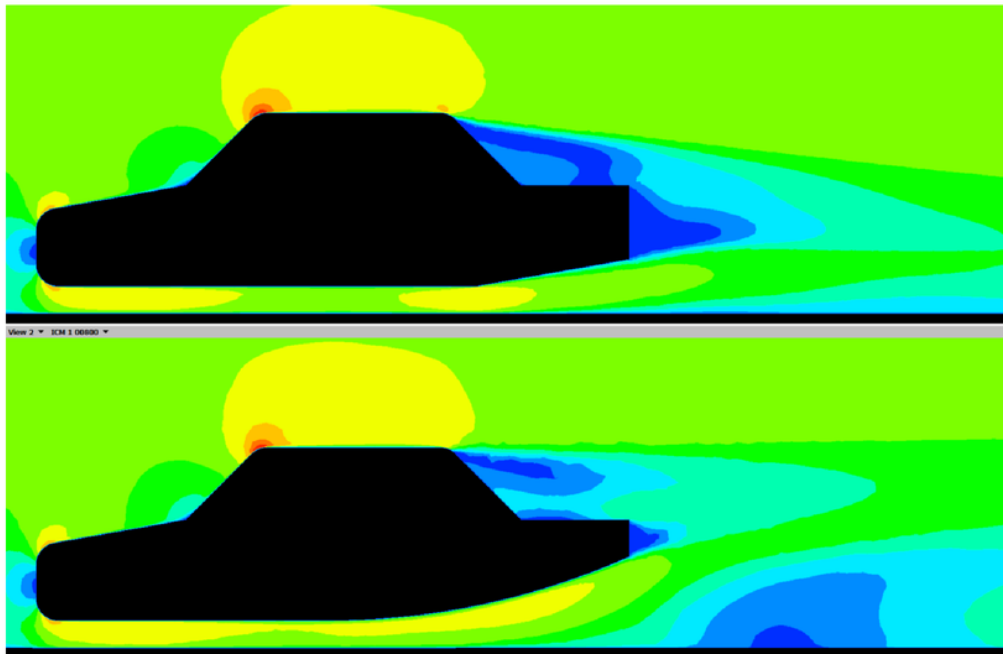


Figure 17. Velocity contours on symmetry plane

In Figure 17 above, looking at the underside of the cars, it is clear that the diffuser helps maintain that high speed flowing air under the vehicle. The flow remained attached throughout the length of the diffuser. Furthermore, by returning the static underbody pressure back to ambient gradually thanks to the diffuser, the low speed wake behind the car has decreased. Contradicting this is the fact that the drag increased. This might be caused by that low speed bubble created behind the car on the ground acting as part of the cars wake.

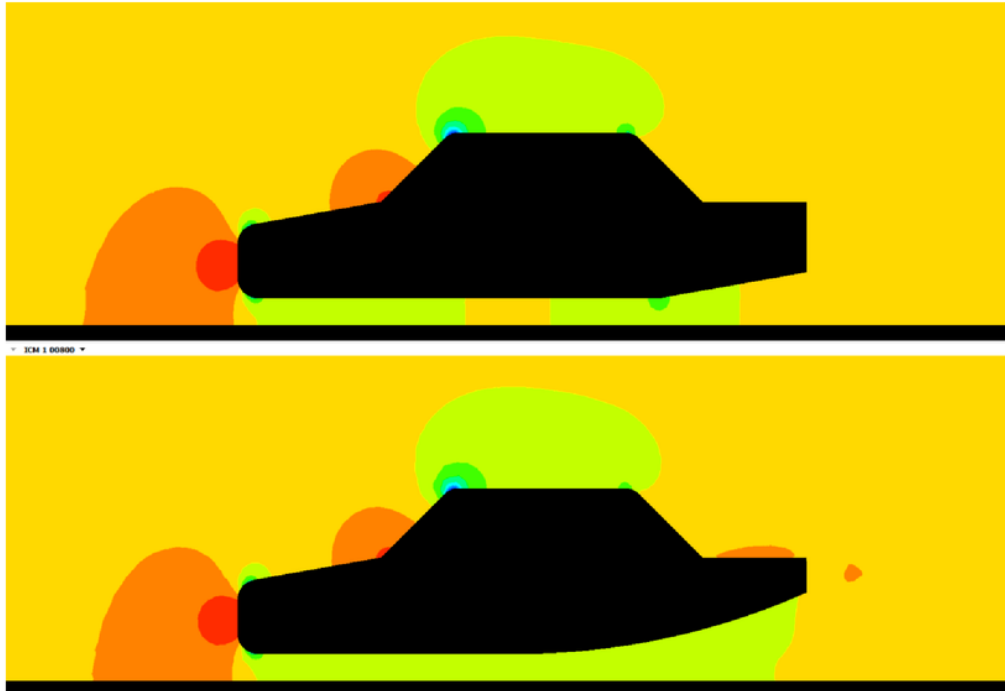


Figure 18. Pressure contours on symmetry plane

Inspecting Figure 18, it is obvious that the diffuser helped spread the low pressure under the car over a large area, hence generating more downforce.

There also seems to be a high-pressure bubble on the rear deck of the modified geometry, possibly due to some recirculation. Figure 19 below, negates the previous statement, because the streamlines clearly show a significant decrease in recirculation on the rear deck of the geometry with the diffuser. Figure 20 however, indicates that the bubble created was very small and happened to be intersected by the symmetry plane in Figure 18, so the overall pressure contours on the rear deck of the cars seem very similar.

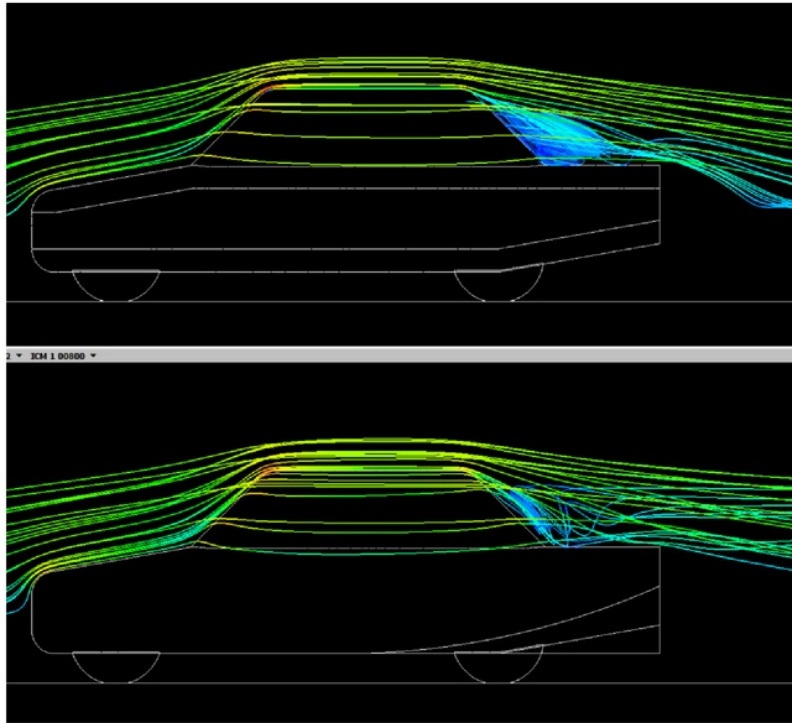


Figure 19. Streamlines

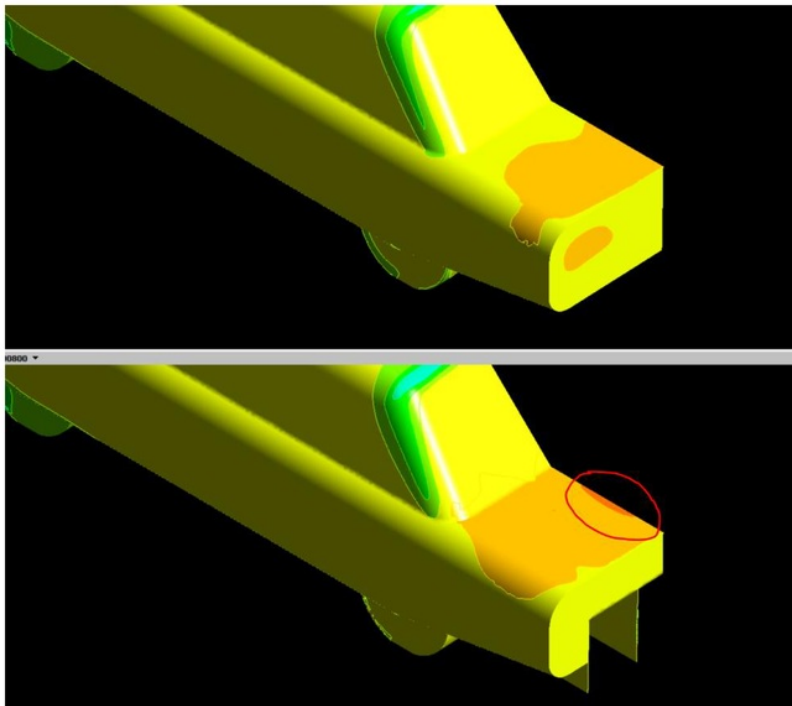


Figure 20. Pressure contours on rear deck

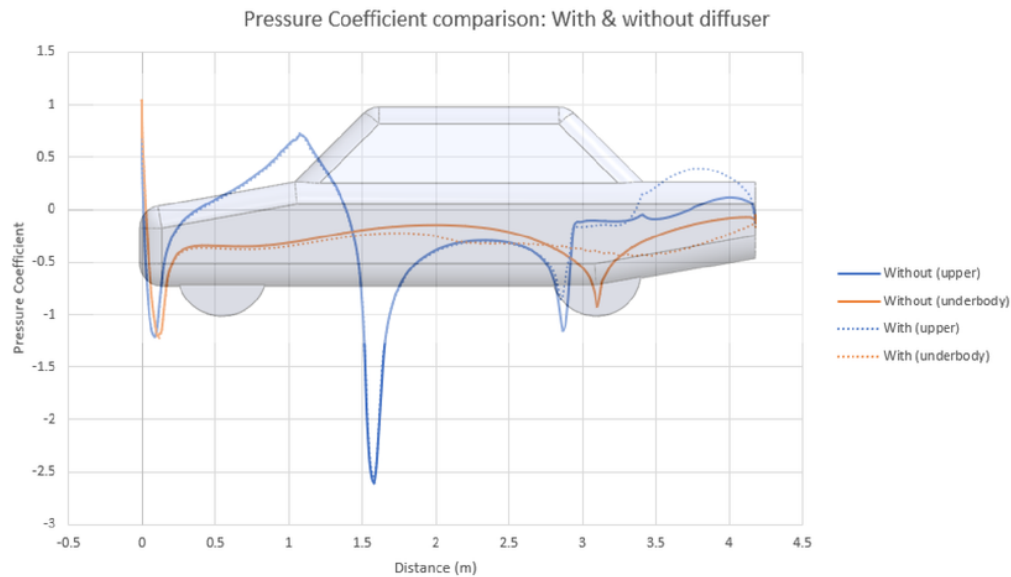


Figure 21. Pressure coefficients comparison between geometry with & without diffuser

Finally, observing the plot in Figure 21 above, as previously stated, the overall pressure under the car with the diffuser was lower. The low-pressure spike that can be seen in the underbody without the new diffuser (solid orange), is caused by the kick point of the original diffuser. Returning to the rear deck, the pressure, after all, did increase (blue dotted line) on the geometry with the new diffuser. The plot above displayed what was expected, the pressures ahead of the new diffuser remained almost identical and varied after its kick point all the way to the rear of the car.

5.4 Conclusion

To conclude, the diffuser did increase the overall aerodynamic performance of the car (C_L , C_d , Efficiency), and it affected its aerodynamic balance by moving the centre of pressure 10% further rearwards.

6 Recommended Further Work

If the time frame for this project was larger, some further work to complement the results could be performed, such as:

- Individual studies for surface mesh, density box mesh and domain mesh.
- Meshing method study.
- A more in-depth study of turbulence models.
- Testing of a variety of aerodynamic devices.
- Vehicle attitude study for each aerodynamic device.

7 References

- Hucho, W. H., 1998. *Aerodynamics of Road Vehicles, Fourth Edition*, Warrendale: SAE International.
- Katz, J., 2006. Tools of the trade. In: *Race car aerodynamics*. Massachusetts: Bentley Publishers, pp. 61-108.
- Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, Darmstadt: Fluent Germany.
- Le Good, G. M. & Garry, K. P., 2004. *On the Use of Reference Models in Automotive Aerodynamics*, s.l.: SAE International.
- McBeath, S., 2015. Computational Fluid Dynamics (CFD). In: *Competition car aerodynamics*. Dorchester: Veloce Publishing, pp. 46-58.
- McBeath, S., 2015. Underbody aerodynamics. In: *Competition car aerodynamics*. Dorchester: Veloce Publishing, pp. 146-183.
- Merriam-Webster, n.d. *merriam-webster*. [Online]
Available at: <http://www.merriam-webster.com/dictionary/evolve>
- MIRA, 2019. *Full scale aerodynamic wind tunnel*. [Online]
Available at: [https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-\(fswt\)](https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt))
- Piechna, J., Rudniak, L. & Piechna, A., 2009. *CFD Analysis of the Cental Engine Generic Sports Car Aerodynamics*, Munich: EASC.