Indian Institute of Technology, Kharagpur

DEPARTMENT OF AEROSPACE ENGINEERING

AERODYNAMICS AE60001

COMPUTATIONAL STUDY ON THE EFFECT OF SLIP-STREAMING

Submitted to: Dr.Sandeep Saha

Assistant Professor

Sumbittedy by:

Ajay Gunasekharan 21AE60R01

Jegadeeshwaran J 21AE60R07

Dorbala Sai Naga Bharghava 21AE60R09

August 30, 2022

Contents

List of Figures

List of Tables

Computational Study of Slip-Streaming

1 Abstract

Slip-streaming refers to one object travelling withing the other objects slip-stream(The region where wake of the fluid moves with velocity comparable to that of the body). This is a popular strategy in car racing because it enhances the likelihood of an overtaking. The trailing vehicle thus influences total down-force, which is essential because tyre traction is strongly dependent on the total normal force exerted on the tyres. The braking distance is also increased when down-force is reduced.

The curved route of a vehicle's motion can cause a considerable shift in aerodynamic performance when it enters a corner. The flow's yaw angle will change over time, and its relative velocity will increase as it moves away from the centre axis of rotation. Particularly in motor-sport, aerodynamic study of vehicles in cornering conditions is an important design element. The majority of racing cars are built to produce down-force that reduces straight-line speed in order to make big gains in the turns. Despite the importance of the cornering situation, aerodynamicists are limited in their capacity to reproduce it experimentally.

Numerical simulation is not constrained in the same way, allowing for further analysis of the situation. Cornering, on the other hand, causes a major shift in the flow field, which must be accommodated in different ways. To allow for the curved flow occuring within a non-inertial reference frame, boundary conditions must be adjusted. Furthermore, drag begins to work in a curved route, and Re varies throughout the domain. The results of this study focussed on the effect of slip-streaming on the aerodynamic parameters of the cars while turning.

2 Introduction

In an ambient fluid through which the object is moving, a slipstream is a zone behind a moving object in which a wake of fluid (usually air or water) is moving at velocities comparable to the moving object. A similar zone close to an object with a fluid moving around it is referred to as a slipstream. Because of the relative speed of the fluid in the slipstream, "slip-streaming" or "drafting" works. The phrase "slip-streaming" refers to one object travelling within another object's slipstream (most often objects moving through the air though not necessarily flying). If an object is going at the same speed as another object, the rear object will use less energy to maintain its pace than if it were moving independently. Bicyclists can employ this strategy, often known as drafting. Slip-streaming is a most commonly used strategy in automotive racing as it increases the chances of a potential overtake. By doing so the trailing vehicle also has an affect on the total down-force which is important as tyre traction is highly depended upon the total normal force acting on the tyres. Reduction of down-force also increases the braking distance.

For a variety of applications, aerodynamic performance through a corner is a critical design concern. Some examples include highly manoeuvrable aircraft and racing cars. Because of its curved line of motion, a vehicle's aerodynamic performance can change dramatically when it enters a corner. The flow is observed to pass through the body in the same curve in the body's reference frame.

Figure 1 shows the free-stream flow over a vehicle while cornering under steady state conditions. The car's angular velocity around an external point is described as constant. With increasing distance from the central axis of rotation, the flow velocity relative to the vehicle will increase. Furthermore, the flow's angle (effectively yaw) will vary along

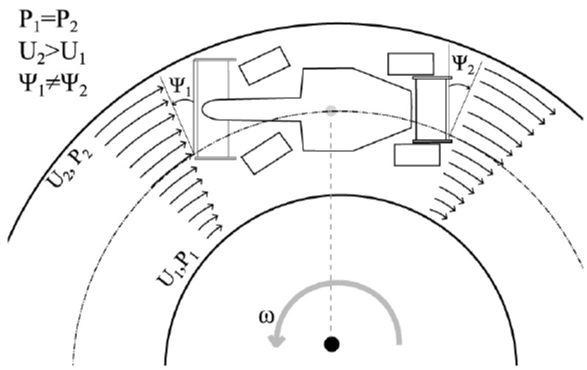


Figure 1: Free-stream flow over a vehicle while cornering

the length of the vehicle. It can be seen that the front and rear yaw angles are opposing in the situation depicted. The rear slip angle in the condition indicated is equal to the front steering angle, minus slip. With different cars and driving styles, the real-world angle of the vehicle through a corner will vary.

The aerodynamics of most racing cars are constructed in such a way that they compromise straight-line peak speed while allowing for significant improvements in the turns. The effective adhesion improves the lateral acceleration that can be sustained around a corner with the addition of down-force acting on the tyres. From the 1960s to the 1990s, racing cars' maximum lateral acceleration rose from about 1.2 times gravitational acceleration to close to four times, with just a minor contribution from improved tyre technology.

Despite the fact that aerodynamic performance is most important when cornering, designs are usually evaluated in a straight line. This is owing to the fact that the wind tunnel remains the major tool for aerodynamic research. The drawbacks of these approaches are well known in the industry. Because the true condition has yet to be attained in the public domain, numerical simulation is the preferred method for this type of analysis. While numerical simulation's capabilities are constantly improving, one known flaw is its inability to accurately forecast sensitivities to changes in conditions. Sweeps in ride height and yaw (or crosswind) will be examined in the wind tunnel rather than simulated. For the time being, aerodynamicists must rely on numerical modelling for cornering and experimental data for the straight-line, or yawed, situation.

In this numerical analysis we studied the effect of Slip-streaming on the down-force in cornering scenario. While doing the analysis velocity variation with radius is neglected for making the analysis simpler.

3 Methodology

In order to finalize the model and computational setup, we have divided the methodology of solving the problem into three parts. 2-D and 3-D analysis of a basic car design using ANSYS Fluent and 3-D analysis of replica of an actual race car model using Open foam.

3.1 2-Dimensional Analysis of basic model car using Fluent

3.1.1 Domain

General studies regarding the flow behaviour on the car involves the usage of a basic shape ”Ahmed body”, that was devised by S.Ahmed in 1984. It is designed in such a way to mimic the flow pattern over an actual car body ”High displacement flow in the front, relatively uniform flow in the medieval region and a large wake at the back”. There has been a lot of information of the aerodynamic properties over this body which could be used for validation and finalizing the mesh density for the simulation. For this study we have designed a body which resembles the shape of toy car and the dimensions that of the Ahmed body. Two cars of the same shape has been placed one behind the other separate by 2 times of their length. The flow domain over the body is as shown in the figure 2. The domain is the split into 4 regions so that more refinement can be done near the body surface for better capturing of variations of flow parameters

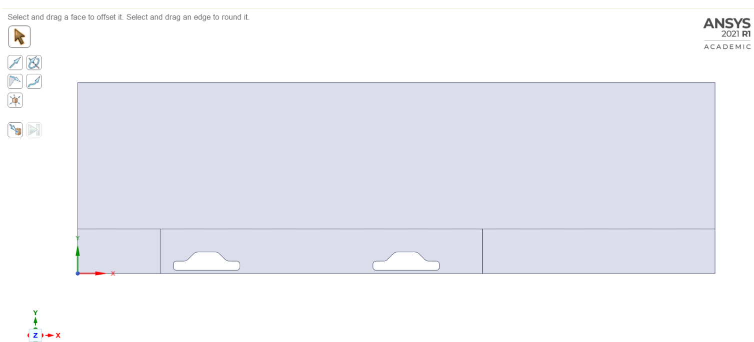


Figure 2: Flow domain over the car

3.1.2 Mesh

The flow domain has been divided into finite number of small triangular cells using mesher of mesh density(1.75L elements). It is shown in the figure 3. Hybrid mesh(both structured and unstructured) is used for meshing the domain. Structured mesh helps us in solving the governing equations quickly compared to unstructured. So wherever possible, structured mesh is used while the remaining region is divided with unstructured mesh.

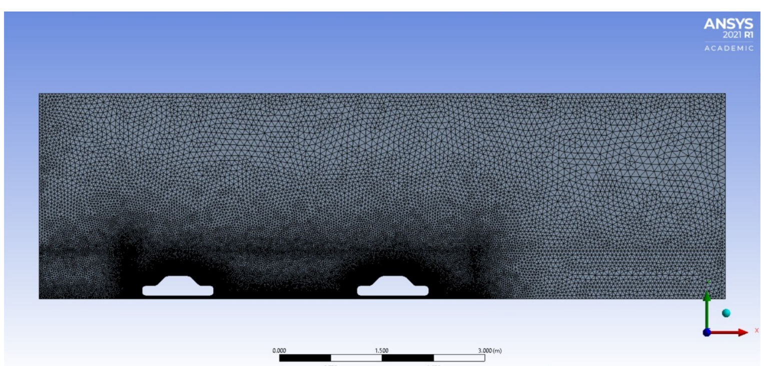


Figure 3: Meshed 2-D flow domain

For better capturing of Velocity that accounts for boundary layer, the region near the wall surfaces of the cars has been divided into even more finer elements using inflation as shown in figure 4. While giving the inflation, the first layer thickness and maximum number of layers are chosen in such a way that the y+ value lies in between range of

30 and 100.

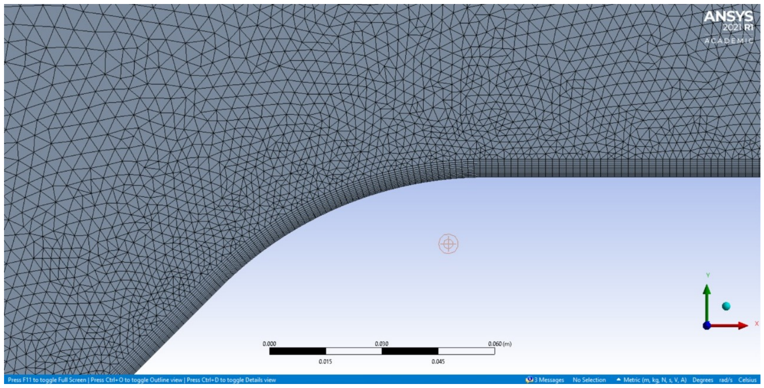


Figure 4: Inflated layer

3.1.3 Setup

Pressure based steady flow solver, K-epsilon turbulence model with Standard and non equilibrium wall functions is used for simulating the model. Inlet velocity of 40m/s in axial direction. Pressure outlet is chosen for outlet. Top and bottom surfaces of the domain are considered as slip wall i.e., no shear whereas the surfaces of the cars were given as no-slip wall conditions. Turbulent Intensity is 2.93% and Turbulent viscosity ratio 10. Pressure-velocity coupling and green gauss cell based gradient is chosen as spatial discretization.

3.2 3-Dimensional Analysis of basic model car using Fluent

3.2.1 Domain

The problem taken was to study the effect of down force while turning which implies it requires a 3D simulation. Having done the 2D simulations, we moved to 3D case of slip-streaming cars in turning situation. As it was a preliminary simulation and considering the computational power, a simple car model with an attached spoiler was modelled was made using solidworks 3D modelling software. A curved 3D domain resembling a curved path in which the cars move while turning was made. The two cars of same shape and dimensions are placed one behind the another separated by a distance which is equal to the length of the car. The domain is shown in the Figure 5.

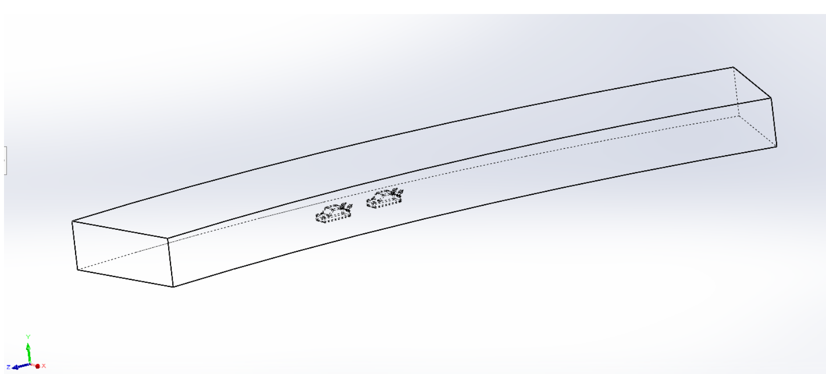


Figure 5: 3-D curved domain

3.2.2 Mesh

The domain was discretised into a structured mesh comprising mostly of hexahedral cells with cutcell method in ANSYS meshing tool. The cells near the car were very fine and the size increases with discrete lengths as we move away. The total number of nodes was around 8.5 lakhs. The meshed domain along with the given boundary conditions are shown in Figure 6.

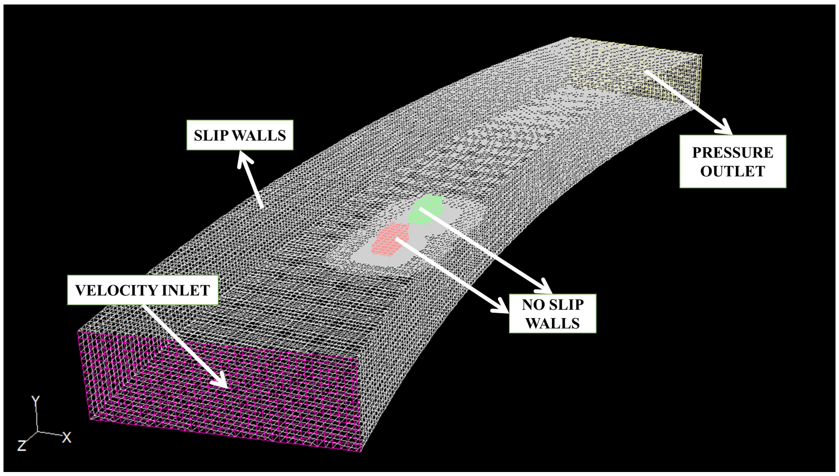


Figure 6: 3-D discretized domain with boundary conditions

Also for clear view of the mesh near the wall surfaces of the car additional mesh figure 7 have been provided.

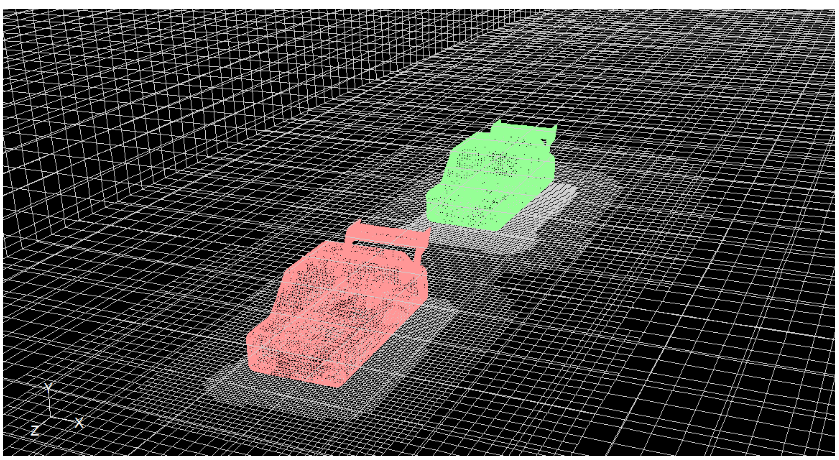


Figure 7: Zoomed view of the mesh near car surfaces

3.2.3 Setup

The side faces of the domain were considered to be slip wall such that the incident flow direction for the car varies along the length and hence resembles the turning scenario. The steady state, pressure based simulations were done using Realizable k-epsilon turbulence model with standard wall function with a freestream air velocity of 30 m/s.

3.3 3-Dimensional Analysis using Openfoam

3.3.1 Model

The vehicle under investigation is a F1 2021 concept car(Figure 8). The geometry used was downloaded from an external site. The model used is an approximation of the actual car without much intricate details. It has simplified front and rear wings to generate down-force only, which doesn't serve other purposes like generating vortices. The model was scaled 1e-3 (i.e., m to mm).

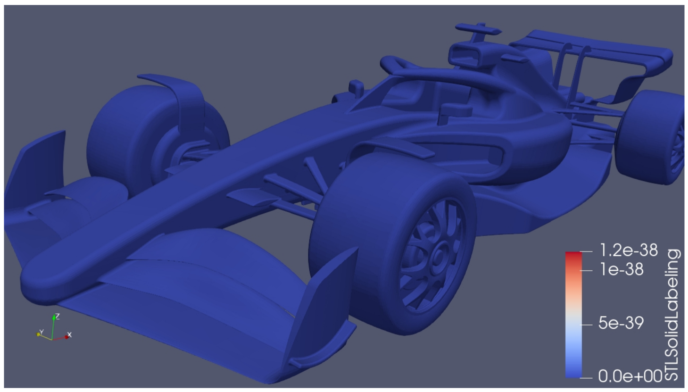


Figure 8: F1 2021 concept car

Our study is to understand the effect of slip-streaming on the rear car while turning or cornering. Experimenting under controlled settings to mimic a cornering scenario is challenging. It's a condition that can only be studied on a track or road, but the complexity of real-world conditions and the difficulty of obtaining data from an actual vehicle limit this.

The most obvious option is to design a track within a controlled environment on which an instrumented model can be propelled. Due to the substantial space requirements and inevitable expenditures, this isn't practical in the vast majority of circumstances. Additionally, some of the limits may include flow settling time between runs, instrumentation, and the time required to test variables such as riding height. So, numerical simulation is preferred for cornering aerodynamics analysis. In terms of boundary conditions and the structure of the domain there has been some variation amongst studies. In most instances the domain is defined as non inertial reference frame motion is defined by prescribing an angular velocity about a point external to the domain as shown in figure 1.

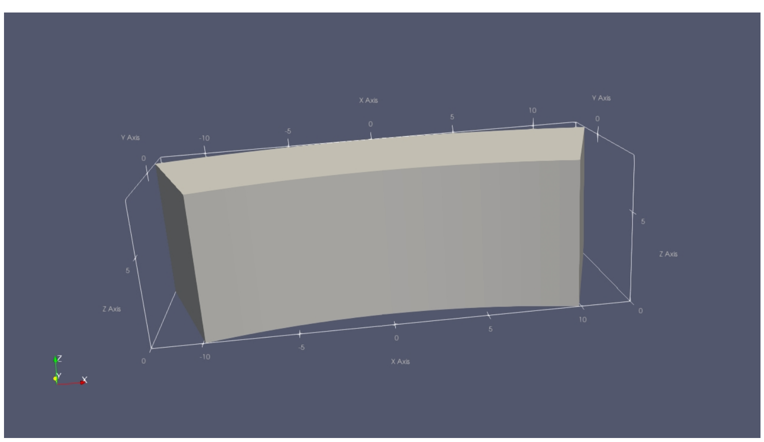


Figure 9: 3-D Domain

The domain that we are using in this study was a curved domain with uniform square shaped cross section .The dimensions of the domain are shown in the Table 1. Figure 9 shows the shape and structure of the domain that we have used in this study.

Name	Dimension
Inner radius	35.5u
Outer radius	43.5u
Height	8u
Width	8u
Angle	32.279°

Table 1: Domain dimensions

3.3.2 Mesh

SnappyHexMesh (Figure 10), which the Hex-dominant meshing algorithm type is based on, is a mesh generation tool in the OPENFOAM® open-source software. This tool generates three-dimensional unstructured or hybrid meshes consisting of hexahedra (hex) and split-hexahedra (split-hex) elements. Besides the automatic meshing tool, the SimScale platform also houses the SnappyHexMesh feature. The algorithm iteratively performs the steps detailed below to obtain the final mesh.

The basic snappyHexMesh methodology consists of 4 main steps:

- In the first step, the Castellated mesh step, the reference base mesh is created (with or without refinements) around the object.
- Next, the castellated mesh is snapped onto the object's surface in the Snapping step.
- If specified, mesh layers are generated on selected surfaces and adjusted to the main mesh in the Layer addition step.
- Finally, in the Mesh quality assessment step, the final mesh is checked for illegal/bad cells and further iterations are performed until the quality standards are attained.

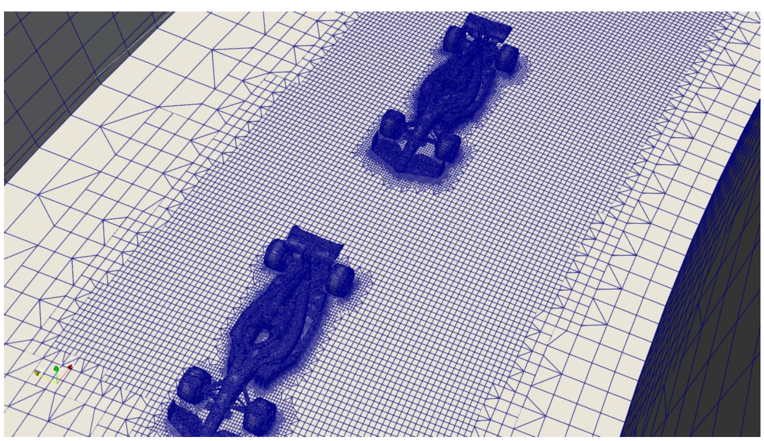


Figure 10: snappyHexMesh

3.3.3 Solver

The simulations were done using using Simplefoam. SimpleFoam is a steady-state solver for incompressible, turbulent flow,that uses SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. In the newer releases it also includes an option to use the SIMPLEC (Semi-Implicit Method for Pressure Linked Equations Consistent) algorithm. The steady state simulations were done using k-ω SST (Turbulent Kinetic Energy- Vorticity Shear Stress Transport) turbulence model with the cars one behind the other separated by a car's length apart with a free stream air velocity of 20 m/s. The parameters focused in this study were the coefficients of lift and drag of the leading and the trailing car for three different positions of the trailing car with respect to the leading car.

3.3.4 Boundary Conditions

Inlet

Velocity inlet condition with a velocity of 20m/s normal to the inlet patch and resolved into x and y components using the known angle(θ). Inlet turbulent intensity was given to be 24% and ν_t =0. ω = 1.78

Outlet

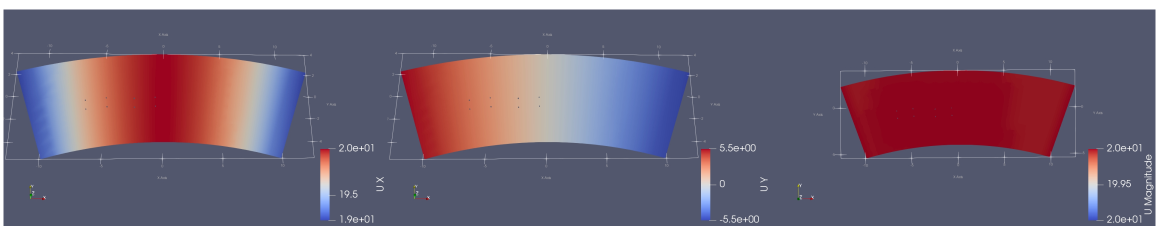
The outlet was given as zeroGradient.

$$\frac{\partial}{\partial n}\phi = 0$$

Turbulent intensity was given to be 24% and ν_t =0. ω = 1.78

Road/Lower wall

For each cell of the patch the velocity has been given with respect to its position and angle(θ_c) so as to maintain the same resultant velocity i.e., tangential.This is to account for the fact that the cars are stationary frame of reference.



(a) Ux (b) Uy (c) U

Figure 11: a) Velocity of road along X-axis b) Velocity of road along Y-axis and c) Resultant Velocity

Front & Back walls

Slip wall condition is given. For each cell of the patch the velocity has been given with respect to its position and angle(θ_c) so as to maintain the same resultant velocity i.e., tangential.This is to account for the fact that the cars are stationary frame of reference.

Upper wall

Upper wall is considered to be a symmetryPlane as it represents the free atmosphere.

Cars

There are two cars,for both the cars all the surfaces are considered as no-slip walls.

4 Results & Discussion

4.1 2-Dimensional Analysis of basic model car using Fluent

The cars are designed in such a way that they themselves can produce significant amount of down-force without the need of spoilers. However the same designs with the addition of spoiler(Inverted S1223 airfoil) have also been analyzed to get a clear picture of flow behavior. An accuracy of 1e-5 is used for convergence of the residuals and the following residuals are obtained.

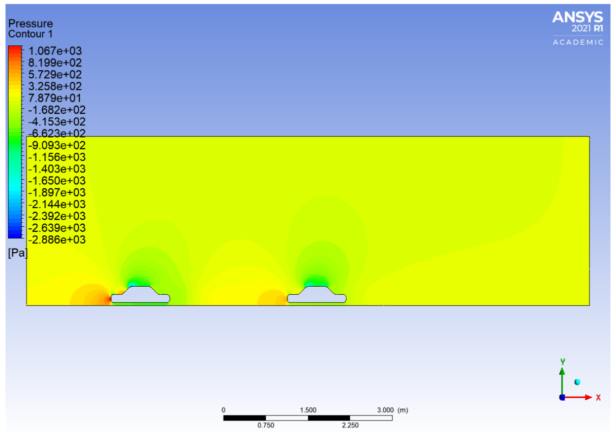


Figure 12: Pressure distribution without spoiler

Figure 12 indicates the Pressure distribution over the leading and trailing cars. As we can observe, a high pressure zone is formed in the front portion of the leading car(stagnation region). The same can be observed in case of the trailing car too but the intensity of the stagnation region has been considerably reduced. This is due to the interaction of the wake of the leading car with the stagnation region of the rear car. The size of the expansion fans has also got reduced for the rear car when compared to that of the leading car.

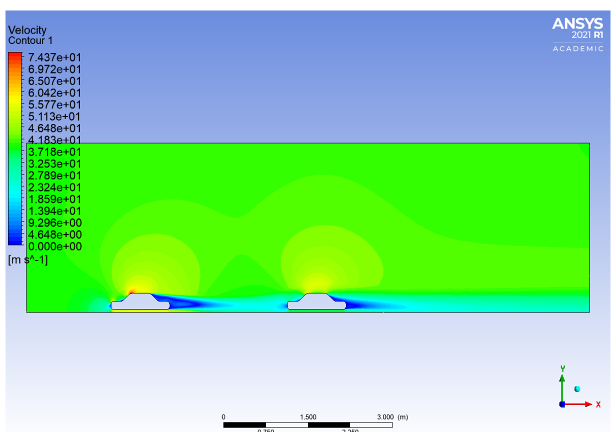


Figure 13: Velocity distribution without spoiler

Figure 13 shows us the velocity distribution. In this contour we can clearly see the interaction of the leading car wake with stagnation region of the rear car. Also we can see high velocity regions are created at the bottom of the leading car compared to that of the rear car, owing to the fact that it experiences lesser free stream flow. This can explain us the reason for the reduction of down-force for the rear car.

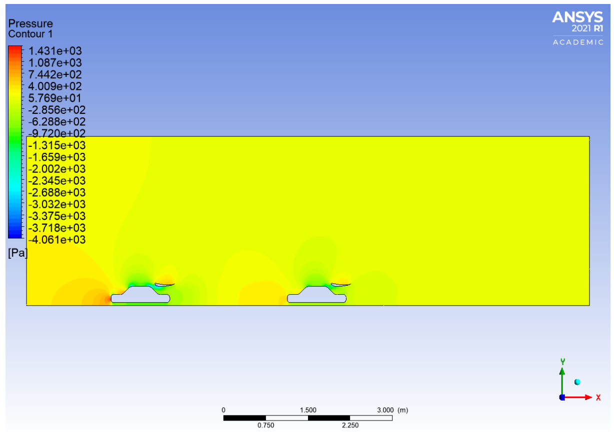


Figure 14: Pressure distribution with spoiler

Figure 14 explains us the variation of pressure over the cars when a spoiler is added to both. Now we can observe that the strength of high pressure region has been increased with that of the cars without spoilers. This explains us that an extra down-force has been created with the addition of spoiler. So the purpose of the addition of spoiler is served. But due to the excess pressure, the drag force acting on the body was increased. However this could be useful in case of turnings or cornering by reducing the braking distance. Also we can observe that all these effects have been reduced for the rear car though a spoiler is being attached.

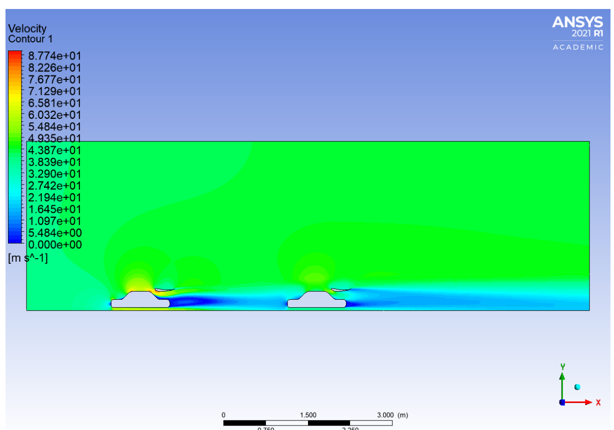


Figure 15: Velocity distribution with spoiler

Figure 15 refers to the velocity distribution of the flow over the cars. We can observe that the interaction of stagnation region of rear car and wake of the leading car clearly in this contour. This is because, the wake of the leading car is being pulled by the spoiler thus leading to increment of the wake region lengthwise. But the strength of the is reducing when it comes to rear car though the length of the region is increased. This reduction in strength of wake explains us the reduction in drag of the rear car.

The values of the aerodynamic coefficients for both the cars in both cases i.e., with and without spoilers has been tabulated(Table 2) for comparison.

Case	C_d	C_l
Leading car	0.3213	-0.3099
Trailing car	0.3091	0.8135
Leading car with spoiler	0.4447	-2.0167
Trailing car with spoiler	0.4063	-0.7426

Table 2: Aerodynamic Coefficients

4.2 3-Dimensional Analysis of basic model car using Fluent

Figure 16 shows the pressure field over the cars as seen from the top of the domain. By seeing the figure one can note the reduction in pressure in the upstream and down stream of the rear car compared to the leading car. Since the distance between the cars is small, we can see that the pressure field of the rear car affects the pressure field around the leading car hence affecting its aerodynamic characteristics. The difference in pressure between the upstream and downstream contributes to the drag experienced by the vehicle. This means that the drag is reduced in both the front and the rear car.

Figure 16: Top view of Pressure field over the cars

Figure 17 shows the top view of the velocity field in a plane, where we clearly see that the rear car experiences less velocity airflow from the wake of the leading car. This explains the fact that both down-force and drag experienced by the rear car is less than that experienced by the leading car. We can also see the acceleration of the flow in the corners of the vehicle.

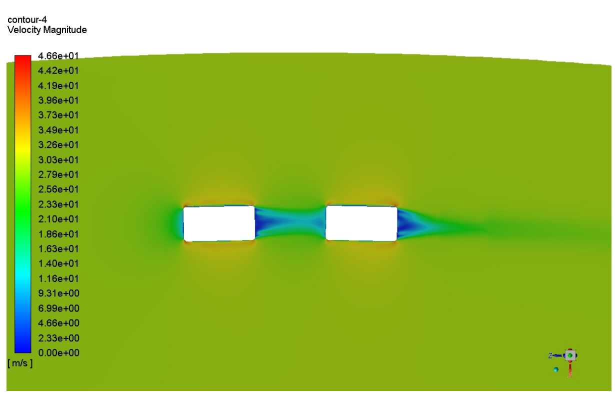


Figure 17: Top view of Velocity field over the cars

Figure 18 shows the pressure field as seen from a side, at a cross sectional plane. The contour shows the pressure difference between the upstream and downstream car for the leading car to be high and it also shows a high pressure region near the windscreen part of the car followed by a low pressure region. Pressure on the bottom surface is nearly the same compared to the top except near the windscreen. This combined with the pressure difference from the spoiler gives a down force of 278 N corresponding to a down-force coefficient of 0.28. The rear are experiences a complete loss of down-force in this case and has a value of 0.3 N.

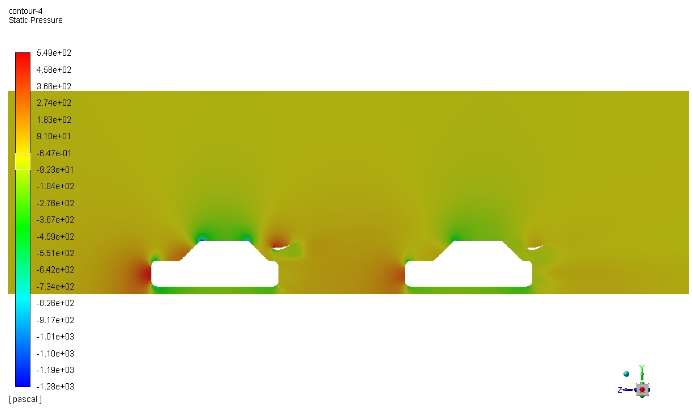


Figure 18: Side view of Pressure field over the cars

The velocity field in the side view is shown in Figure 19. It clearly shows the wake of the cars and how it differs for the both cars. We can see the acceleration of the flow clearly on the top surface of the leading car whereas in the rear car the acceleration is not that much ultimately because the airflow seen by the rear car is from the wake of the leading car. Also we can note that the wake region is big for the rear car compared to the leading car. This is because the energy of the airflow is not sufficient to attach to the surface because it had already lost energy while passing over the first car. Hence the obvious early flow separation compared to the leading car and bigger extent of wake the region. But it is important to note that the difference in pressure contributes to the force experienced. Here, even though the rear car produces bigger wake region, the pressure difference between the upstream and downstream or the top and bottom surface is less than that of the front car and hence the drag and down-force follows this trend.

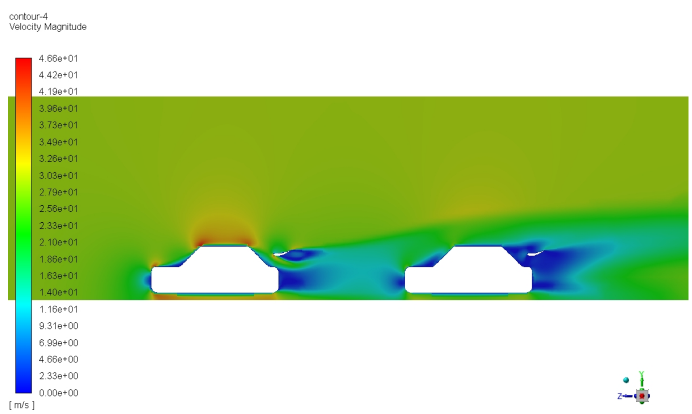


Figure 19: Side view of Velocity field over the cars

4.2.1 Partial Slip-streaming for basic model car

Figure 20 shows the pressure field for the partial slip streaming case where the rear car is 0.5W away from the axis of the leading car where W denotes the width of the car. In this case both the cars have higher down-force compared to the full slip-streaming case. The leading car has a down-force coefficient of 0.33 and the rear car has a down-force coefficient of 0.37 which means the rear car experiences higher down-force than the leading car. This is evident from the fact that, the pressure on the top surface near the windshield of the rear car is more than that for the leading car.

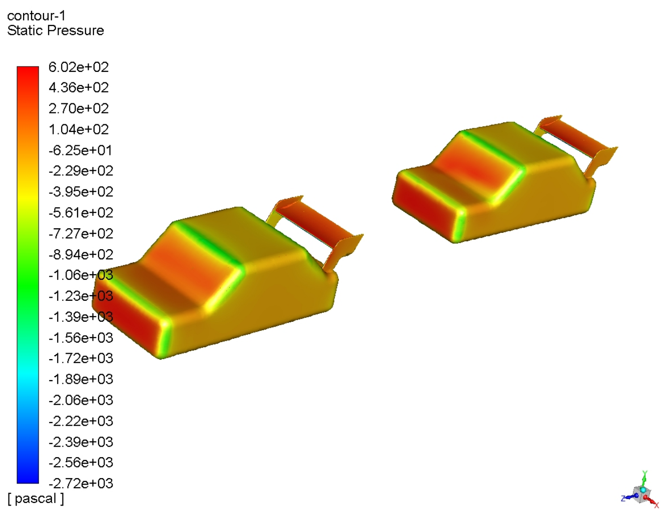


Figure 20: Partial Slip-streaming for basic model car

4.3 3-Dimensional Analysis using OpenFOAM

After doing the 3-D analysis over the basic model car an actual replica of race car model was taken from the internet for simulating various scenarios of slip-streaming. Unfortunately, the model that was taken for studying has certain design faults which causes it to have a high pressure stagnation point under the car which diminishes the ground effect and down-force generated by the front and rear wings. And sometimes even producing positive lifts. Fortunately, we are much more interested in the behaviour and the affects of slip-streaming under different scenarios than the actual value of Lift-coefficients. And the coefficients of lift and drag are just for understanding and comparing the values and does not signify the actual values of an F1 car, which has a much more complicated geometry and completely different values of Cl and Cd. Such complications are neglected here for the sake of simplicity.

4.3.1 Full Slip-Streaming

When a car is fully slip-streaming another car it is expected to experience less drag and decrease in down-force. The figure 21 shows the simulation of such a case. Here, the cars exhibit a similar tread where the front car experiences clean laminar air and has has higher down-force and drag-force and the slip-streaming car has a lower drag-force and even lesser down-force. And the stagnation pressure of the slip-streaming car relatively lower than that of leading car but, the down-force generated by the front and rear wings are not sufficient enough to counteract it resulting in a higher Cl value than that of the leading car.

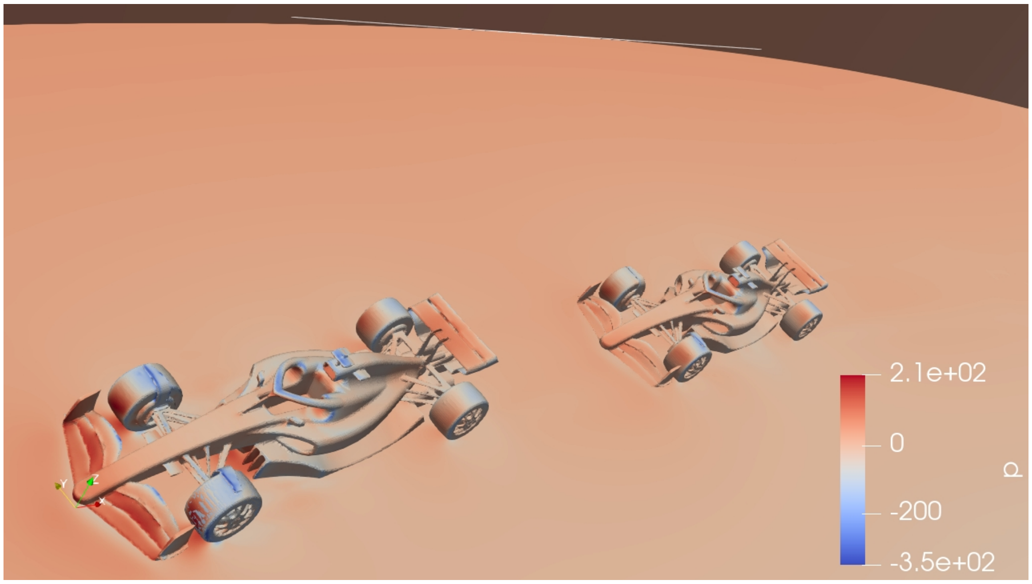
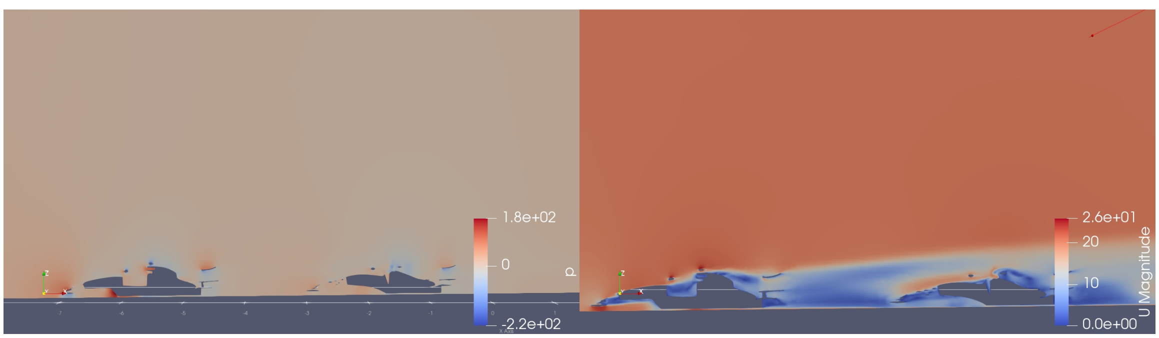


Figure 21: Pressure Plot of trailing car experiencing slip-streaming

Figure 22 it can be seen that the cars models doesn't experience any ground effect. And the pressure is high at the bottom for both the cars and corroborates the previous statement.



(a) Pressure (b) Velocity

Figure 22: Slice view

4.3.2 Partial Slip-Streaming from the Outside

The simulation for partial slip-streaming showed different results compared to fully slip-streaming. The results were as expected with the slip-streaming car with higher Cd value than the fully slip-streaming case and higher down-force.

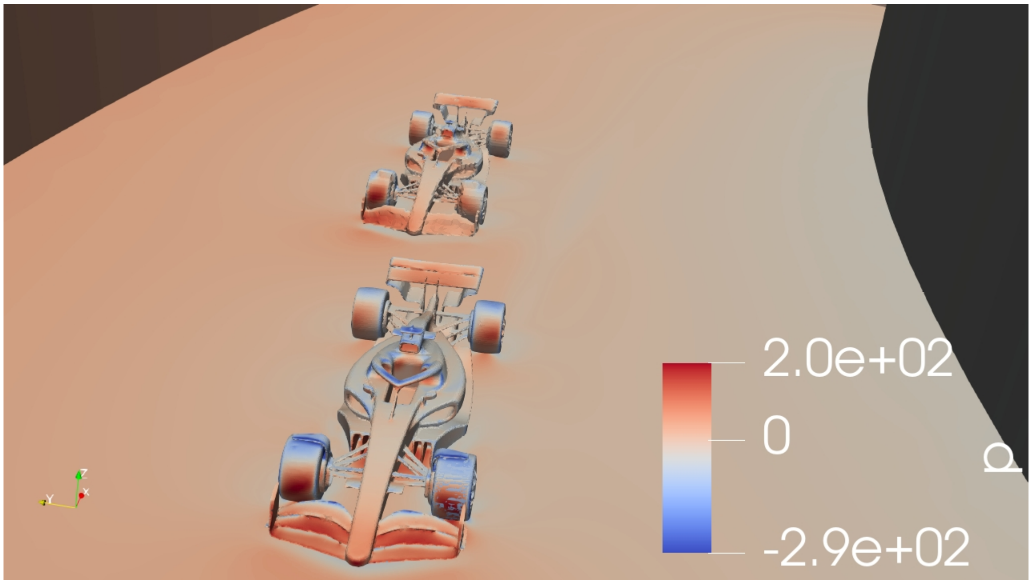
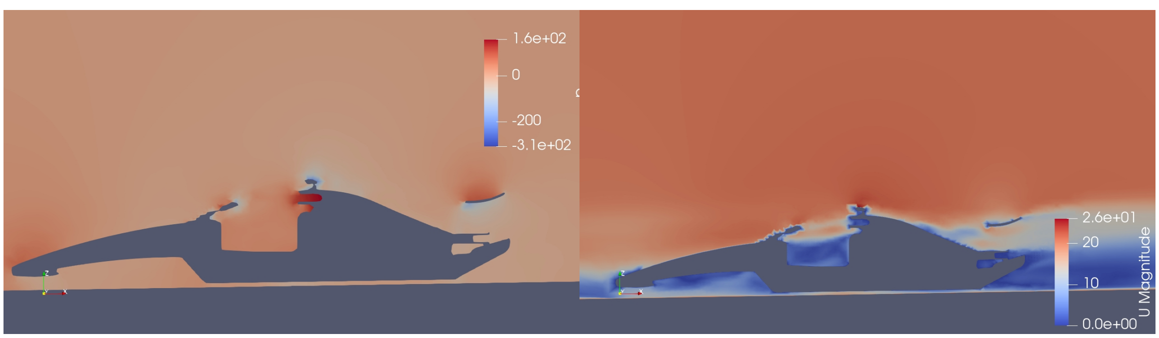


Figure 23: Pressure of trailing car experiencing Partially slip-streaming on from outside

From the figure 23 it can been seen that the pressure at the front wing is increased for the part which is not under the effects of slip-stream which causes a higher down-force as well as increase in drag-force.



(a) Pressure (b) Velocity

Figure 24: Slice view of trailing car

From figure 24 pressure plot it can be seen that the pressure at the at the front and rear wings are a bit higher than the lower part of the car. And this corroborates for the higher down-force.

4.3.3 Partial Slip-Streaming from the Inside

Similar to the previous case, the down-force increased and the drag-coefficient of the car were increased. The pressure of parts of wing not under slip-stream creates higher down-force as seen in figure 25. However, the increase in down-force was not as the previous case.

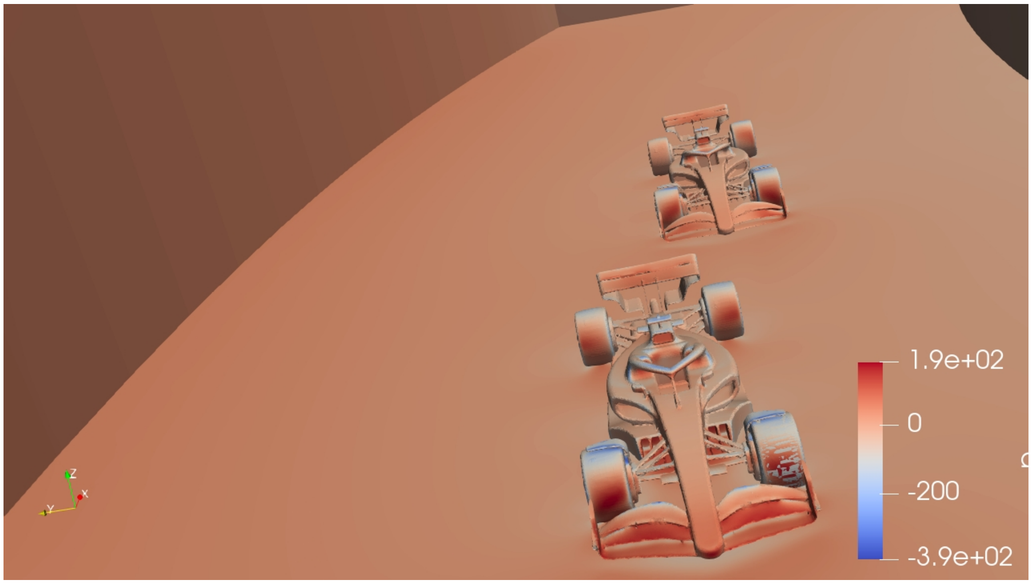
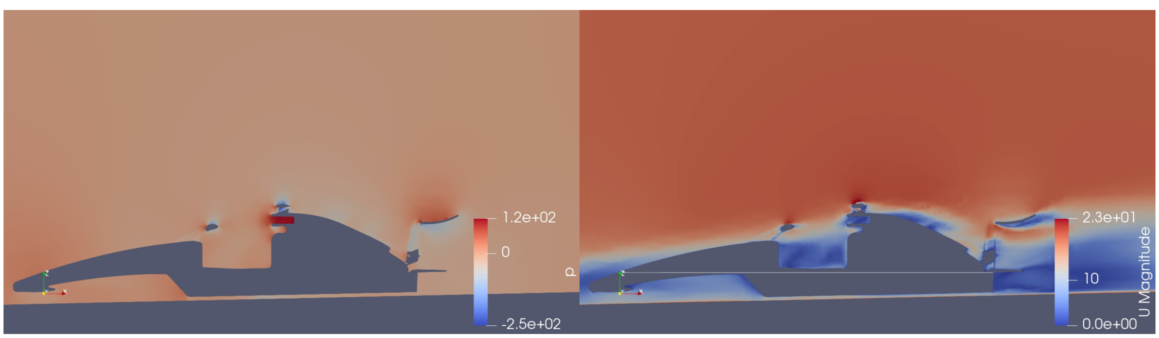


Figure 25: Pressure Plot of trailing car experiencing partially slip-streaming from the inside

It can be seen from figure 26 that the pressure at the bottom is less than that at the top but when comparing it with previous case it is relatively stronger.



(a) Pressure

(b) Velocity

Figure 26: Slice view of trailing car

Listed below are the lift and drag co-efficient of different scenarios.

Case	C_d	C_l
Leading car	0.195	0.0122
Full Slip-Streaming	0.126	0.172
Partial Slip-Streaming Outside	0.163	-0.0379
Partial Slip-Streaming Inside	0.135	-0.00839

Table 3: Coefficients

The values from table 3 represent the lift and drag coefficients from different scenarios, the values also corroborate the above statements. The values for overtaking from outside the value of Down-force is higher because the effects of lower stagnation point was much less compared to that of the inside case.

5 Conclusion

The effect of slip-streaming was numerically studied for different scenarios while turning. It was found that when the trailing was fully in slip-stream of the leading car the down-force was reduced drastically. In case of partial slip-streaming, a part of the car experienced clean air, and was able to produce some amount of down-force. However, this doesn't depict the actual scenarios and cannot be used to justify the real-life overtaking scenarios. Furthermore, to study close to actual scenarios, a better model which is a replica of an actual car, different spacing between cars and different velocities has to be analysed by comparing it with experimental values from wind tunnel.

6 Things learnt

Ajay Gunasekharan (21AE60R01)

- Simulating using OpenFOAM
- Creating cluster
- Mesh Generation
- Solving using cluster
- Different Turbulence models (RAS & LES)

Jegadeeshwaran J (21AE60R07)

- 3-D Designing using Solid works
- Mesh Generation
- 3 dimensional simulation using ANSYS Fluent with various turbulence models

Dorbala Sai Naga Bharghava (21AE60R09)

- Designing using Solid works
- Mesh Generation
- 2 dimensional simulation using ANSYS Fluent with various turbulence models
- Documentation

7 References

1. Gan,E.C.J.,Fong,M.,& Ng,Y.L.(2020).CFD Analysis of Slipstreaming and Side Drafting Techniques Concerning Aerodynamic Drag in NASCAR Racing. CFD Letters, 12(7), 1–16.

2. Džijan, I., Pašić, A., Buljac, A. et al. Aerodynamic characteristics of two slipstreaming race cars. J Mech Sci Technol 35, 179–186 (2021).

3. I. Džijan, A. Pašić, A. Buljac and H. Kozmar, Aerodynamic forces acting on a race car for various ground clearances and rake angles, J. App. Fluid Mech., 12 (2019) 361-368.

4. Cheng, S. Y., and S. Mansor. "Rear-roof spoiler effect on the aerodynamic drag performance of a simplified hatchback model." In Journal of Physics: Conference Series, vol. 822, no. 1, p. 012008. IOP Publishing, 2017.

5. Chin, K. Y., Cheng, S. Y., and Mansor, S. "Yaw Angle Effect on the Aerodynamic Performance of Hatchback Vehicle Fitted with Combo-Type Spoiler." Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 44 (2018): 1-11.

6. Dominy, R. G. (1990). The Influence of Slipstreaming on the Performance of a Grand Prix Racing Car. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 204(1), 35–40.