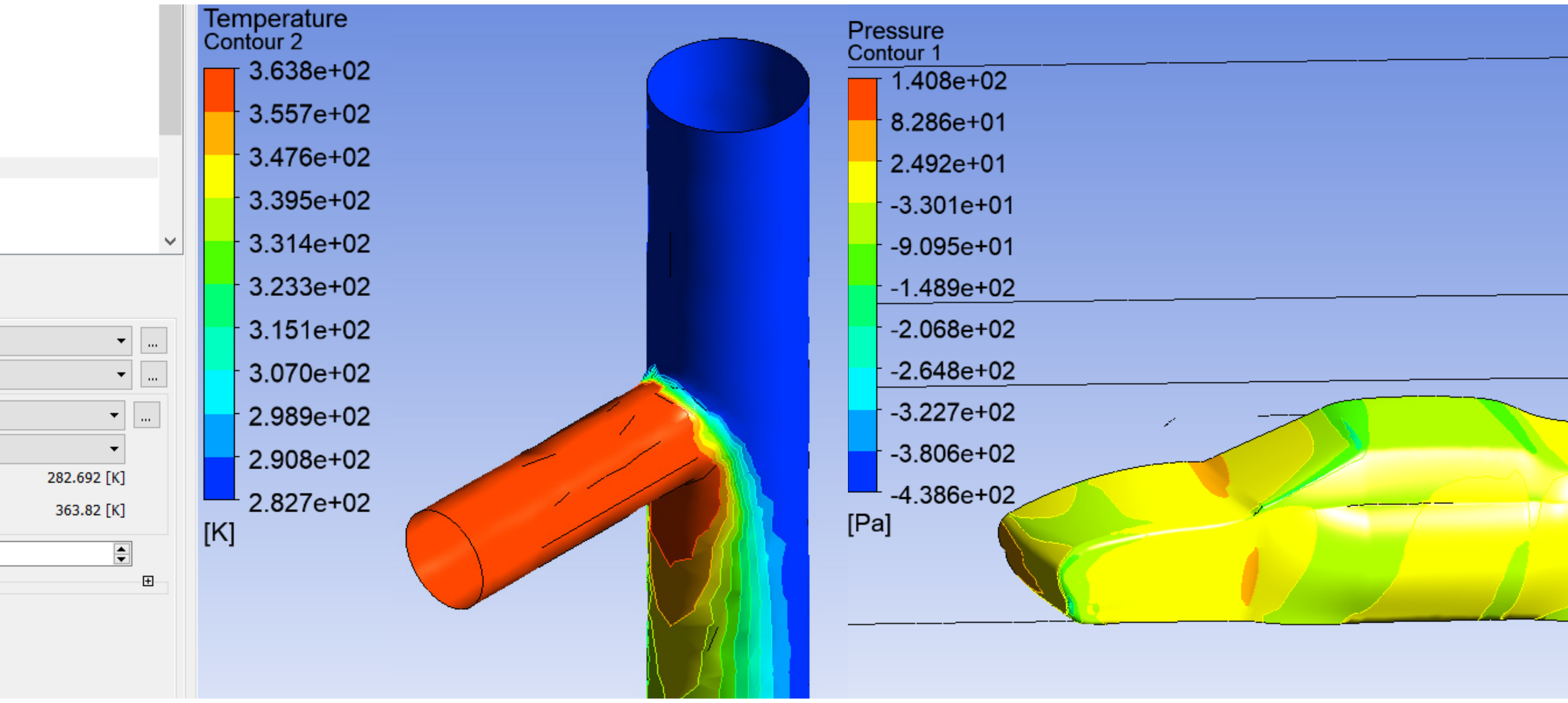
DM3601 - Computer Based Design Methods

# CFD Simulations - Assignment 2

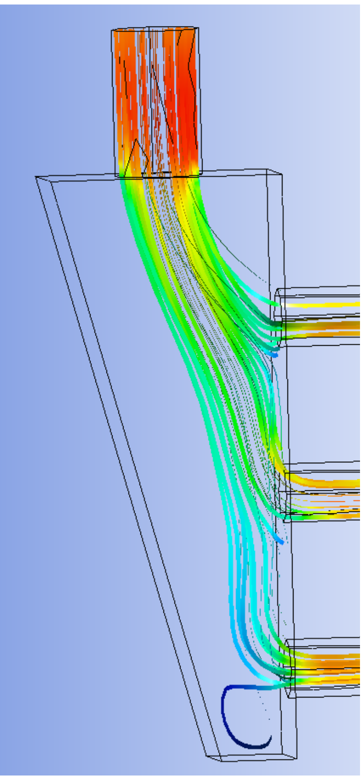


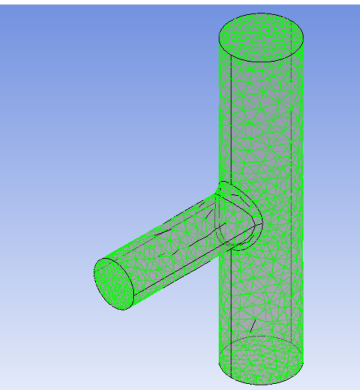

Bora Sen- 2017194

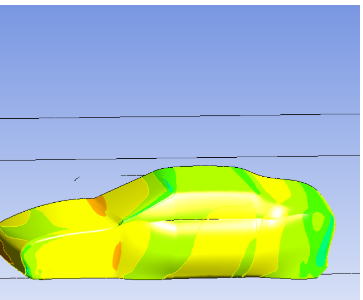
Brunel University London

# *Table of Contents*

# *Introduction*







*This report presents the simulation processes and findings of Computational Fluid Dynamics (CFD) simulation works conducted using the ANSYS application, an important tool in the practice of virtual simulations.*

*In the initial section of this report, I will introduce the processes and outcomes of the four simulations that I have undertaken.*

*Following this, the document will detail the simulations conducted on the simplified version of the vehicle model developed in Term 1, including the results obtained and the design improvements I have personally implemented.*

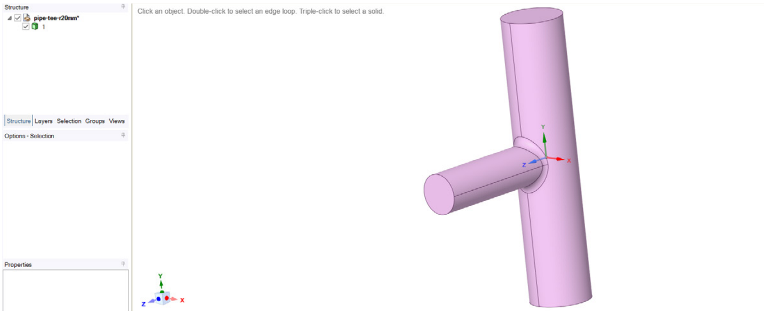
*This report aims to provide a comprehensive overview of our simulation work, showcasing both the methodologies applied and the subsequent enhancements made to the vehicle design.*

*Virtual simulations replace expensive and time-consuming physical prototypes by enabling the investigation of a product's performance in a digital setting. This method helps us modify and optimize designs more effectively by speeding up the development cycle and conserving resources.*

# ***Simulation 1*** - Mixing Tee (T-shape pipe)

## 1.1 Geometry

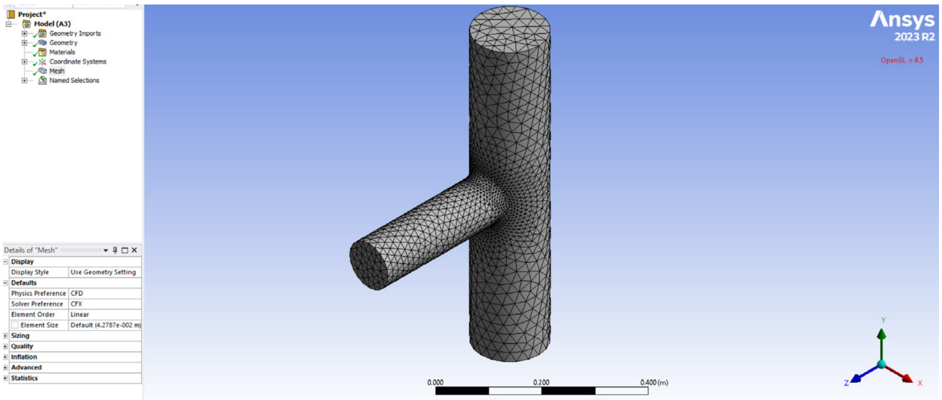
*We can observe the geometry of our T-shaped pipe using Spaceclaim.*



## 1.2 Meshing

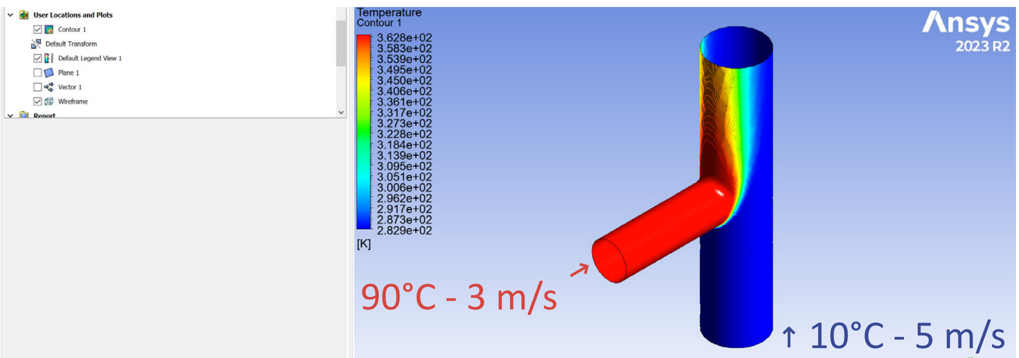
*Meshing is a technique where the complete body is divided into smaller parts, each governed by its own separate equation.*

*In meshing, the focus is on how the points, or nodes, connect with one another. If the nodes are interconnected, they will transfer data among themselves, ultimately enabling the aggregation of results across the entire body. This process is fundamental in computational simulations, as it allows for the detailed analysis and understanding of the body's behavior under various conditions.*




## 1.3 CFD Post

### *Temperature Contour Plot*



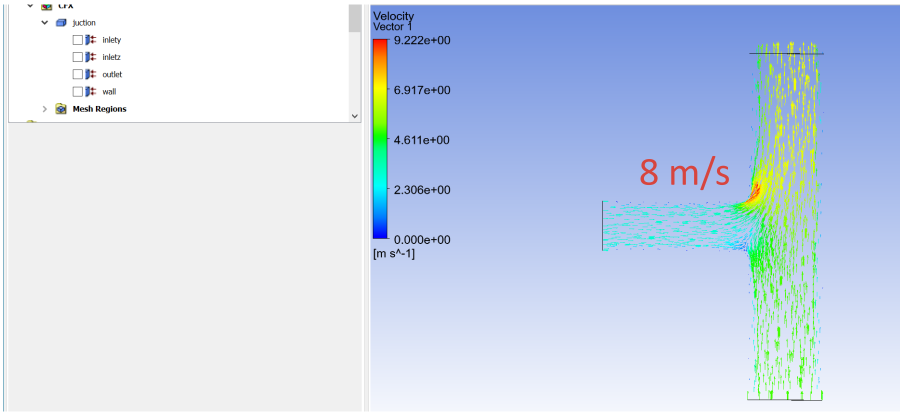

*The pipe has three inlets, with the top and the left face serving as the points of entry. There is a notable temperature difference between two of these inlets: the temperature at the Y inlet is set at 10 degrees Celsius, whereas the Z inlet has a much higher temperature of 90 degrees Celsius. This significant temperature variance is vividly illustrated in the temperature contour plot. The Z inlet is represented by a red color, denoting a high temperature, in contrast to the Y inlet, which is cooler at 10 degrees Celsius.*

# S1. Results

*Water enters through the Y inlet at a velocity of 5 meters per second, while hot water from the Z inlet enters at a slower pace of 3 meters per second. When these two streams meet, the higher velocity of the cold water prevents it from mixing thoroughly with the hot water, propelling it towards the outlet. It is observed that if the velocity at the Y inlet were reduced to 1 meter per second or less, allowing it to be equal to or less than the velocity at the Z inlet, the mixing would occur more uniformly. This would result in a more even color distribution across the contour plot, particularly noticeable on the right side of the pipe.*

*However, due to the higher velocity of the inlet water, it sweeps the hot water along, leading to the effects of hot water being predominantly observed on the ,upper side on Y axis, where the color distribution is apparent. This phenomenon underscores the influence of inlet velocity on the temperature distribution and mixing efficiency within the pipe system.*

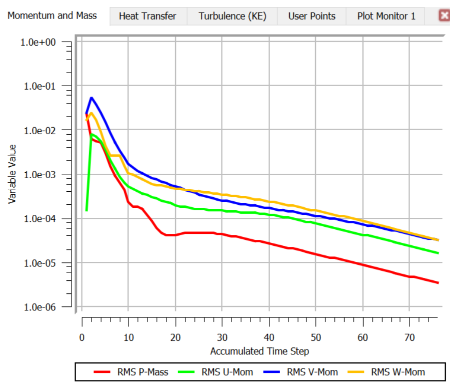
## *Velocity Vector Plot*



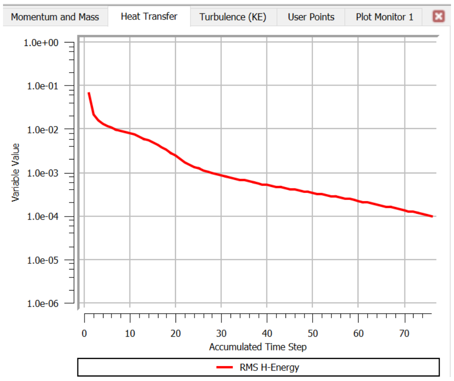

*The Velocity Vector Plot analysis reveals velocity patterns within the system, depicted by arrows indicating flow direction. Water entering from the below (Y axis) at 5 m/s collides with the stream from the left (Z axis) at 3 m/s, resulting in a discernible color gradient in the visualization. This gradient illustrates the velocity differences, with the faster flow marked by a distinct color.*

*The convergence of these streams on the left side is highlighted in red, indicating a higher velocity area where the combined flow reaches approximately 8 m/s, accounting for minor losses due to friction and particle collisions.*

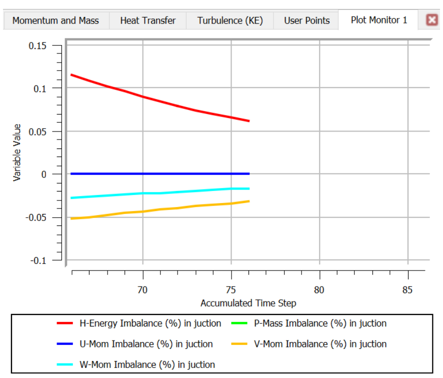
## RMS Residuals

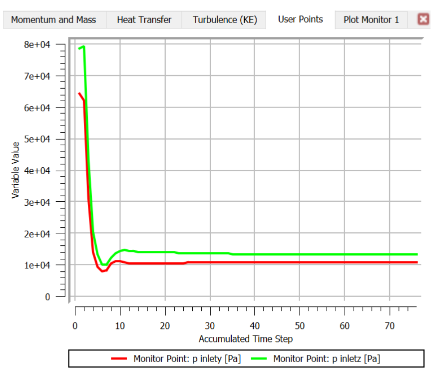



## Heat Transfer

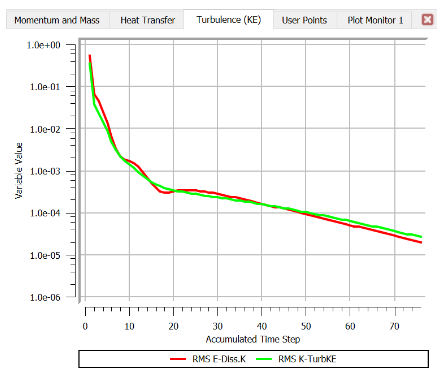



## Imbalances



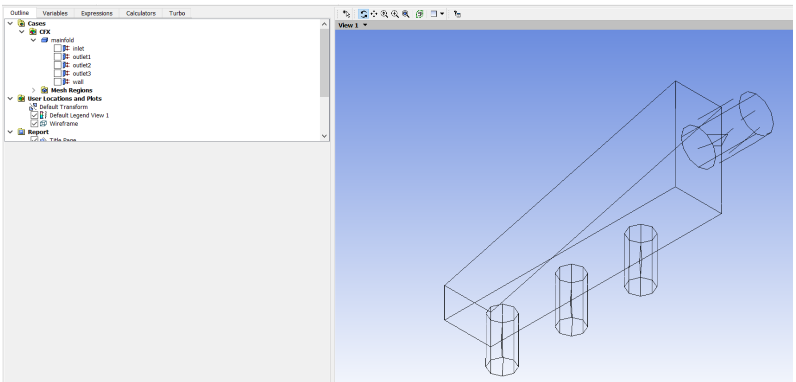

## Turbulance



# Simulation 2 - Manifold

## 2.1 Geometry

*We can observe the geometry of our Manifold using Spaceclaim.*

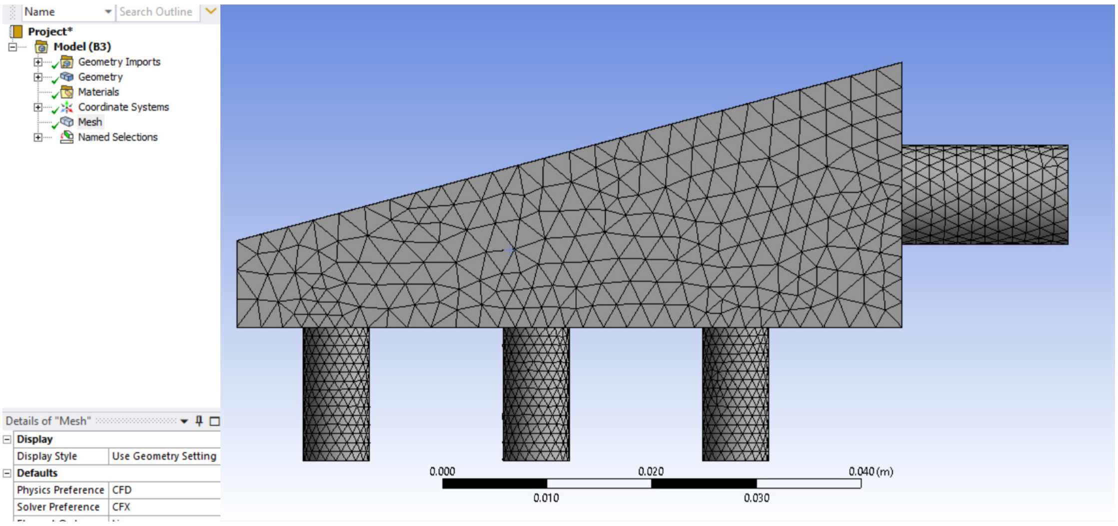


*In this phase of the study, we progressed to more advanced meshing techniques, which involved adjusting the mesh elements from their default settings to specific values.*

*Additionally, we employed the quality of the mesh as an orthogonal property, enhancing the precision of our simulations.*

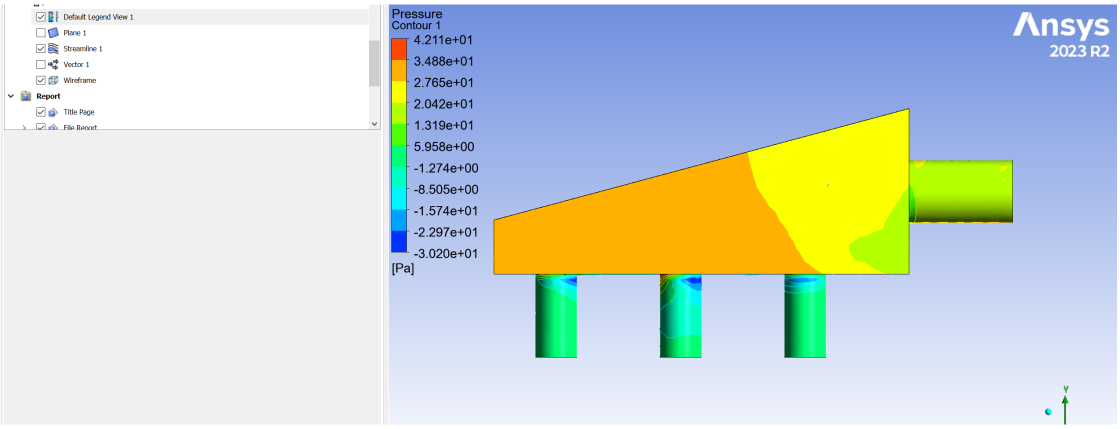
*This step marks a significant advancement in our meshing approach, allowing for more detailed and accurate modeling.*

## 2.2 Meshing




## 2.3 CFD Post

## *Pressure Contour Plot*

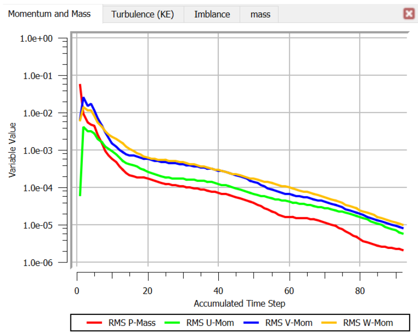



*The visual analysis reveals that air enters the system from the right side, known as the inlet, with high velocity. This observation aligns with the principle discussed in previous classes that a high velocity is associated with lower pressure. Consequently, the absence of red coloring on the inlet side can be attributed to the high velocity and corresponding low pressure in this area. Red coloring is used to denote regions of high pressure, which are not present at the inlet due to these conditions.*

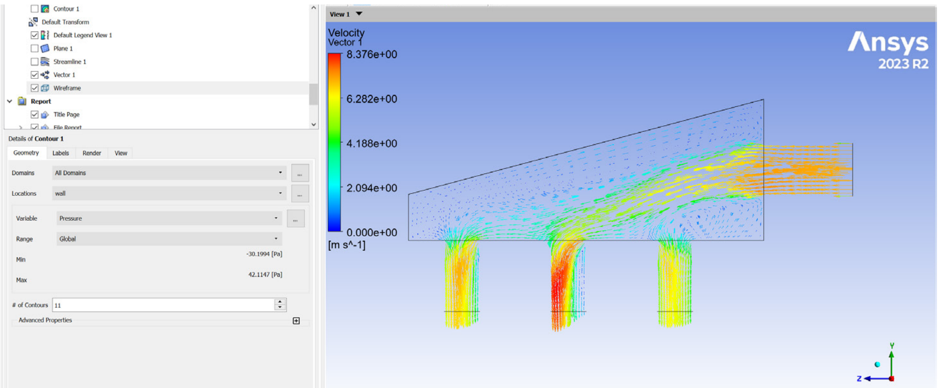
# S2. Results

*Upon entry from the inlet, the air strikes the body of the object. However, it does not immediately exit through the outlets due to the lack of pressure to facilitate its movement. This results in the air not dispersing rapidly across the entire body. When the air makes contact with the body surface, its velocity significantly reduces to almost zero because of the interaction with the wall, leading to an increase in pressure at the point of contact.*

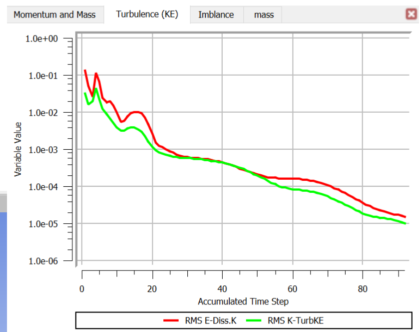
## RMS Residuals



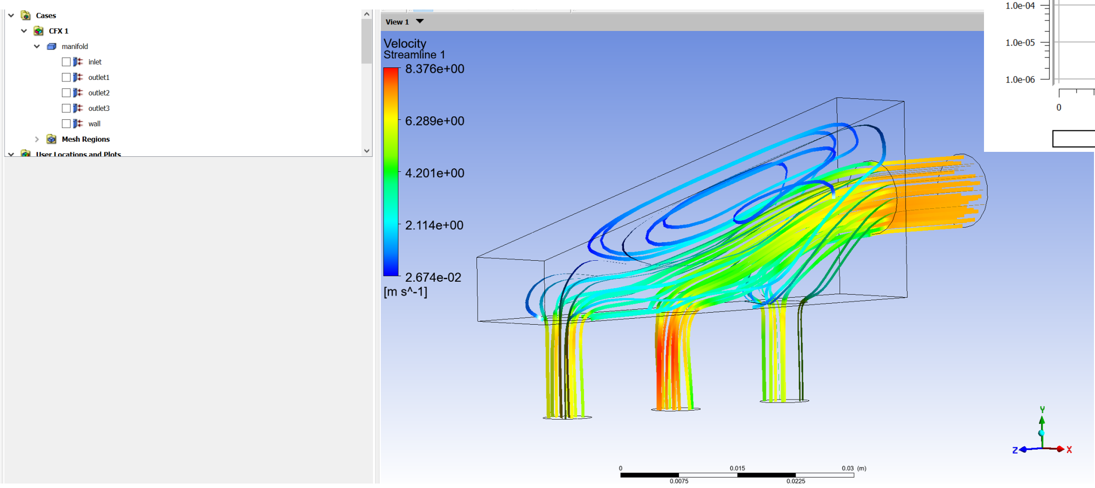

## *Velocity Vector Plot*




## Turbulence



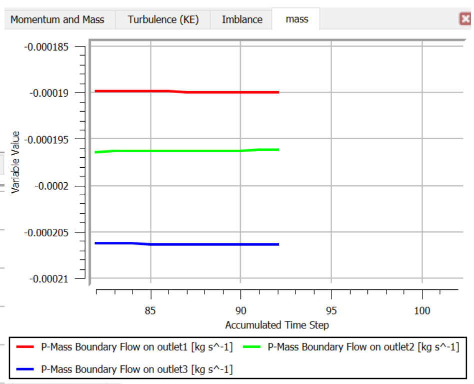

## *Velocity Streamlines*



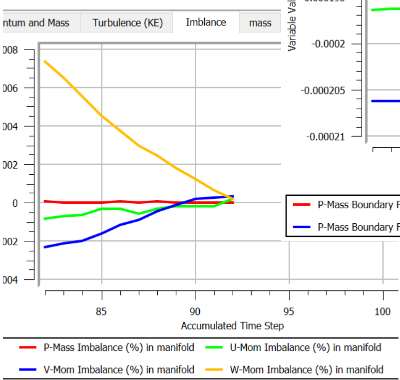

*The streamline form serves as a vector to examine velocity within the system. Our focus on analyzing velocity is reflected in the visual representation, where high velocity is indicated by a red color emanating from the inlet, as previously discussed.*

*On 3 outlets, the velocity decreases for two of them due to particles striking against the walls, with the maximum velocity recorded at 8.095 m/s. This reduction in velocity is attributed to wall friction and the intercollation of particles, factors that significantly influence the flow dynamics within the system.*

## Mass



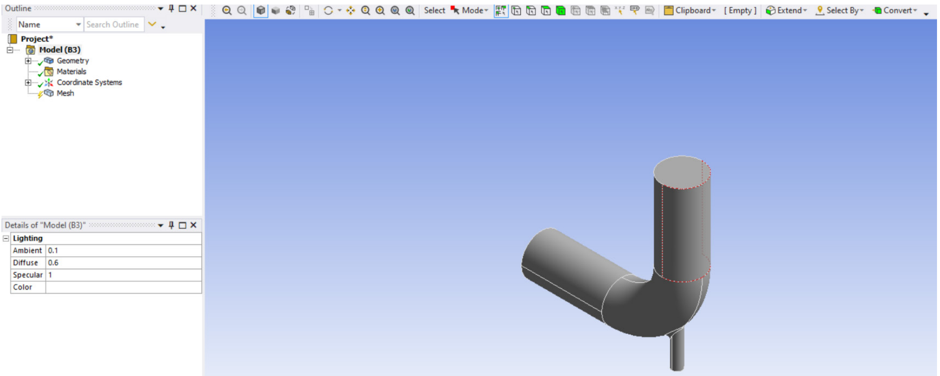

## Imbalances



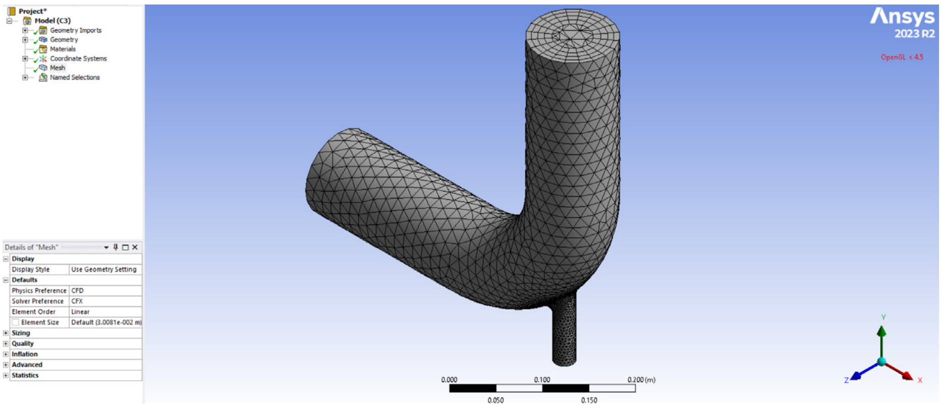
# *Simulation 3* - Mixing Elbow

## 3.1 Geometry

*We can observe the geometry of our Mixing Elbow using Spaceclaim.*

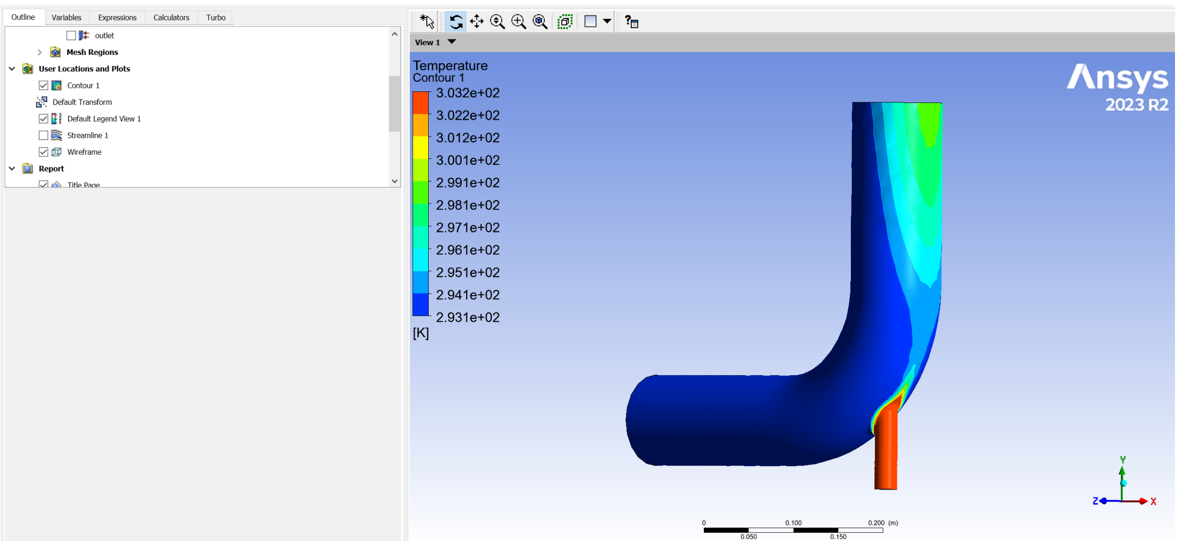



## 3.2 Meshing




## 3.3 CFD Post

*Temperature Contour Plot*



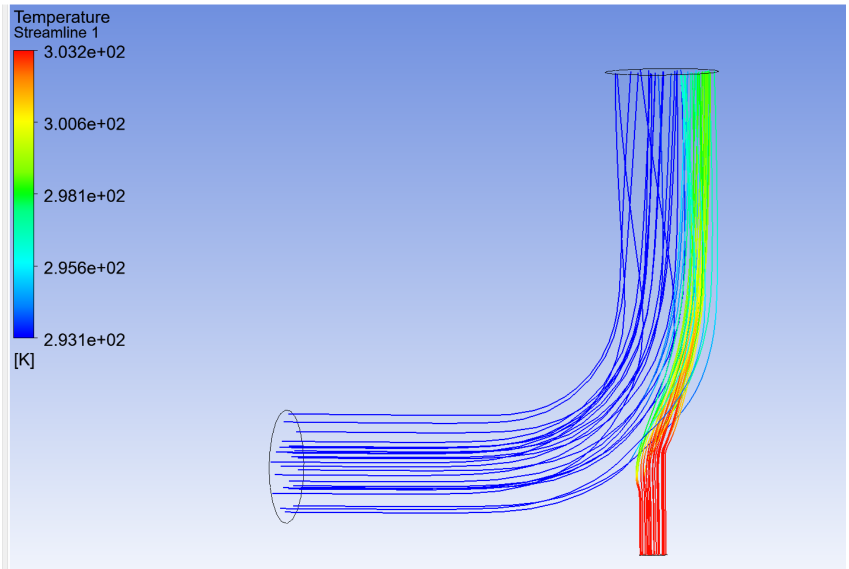

*From the results, it is seen that the small inlet exhibits a completely red coloring, indicating high temperature. Given that our analysis focuses on temperature, this suggests that the small inlet has a higher temperature compared to the large inlet. Consequently, water entering from the large inlet attempts to mix with the hotter water, but this mixing is not uniform across the entire outlet.*

*This uneven distribution is visually apparent, with the left side of the pipe remaining cold and the right side becoming hot due to the influence of the high temperature from the small inlet. Additionally, it is noted that the minimum temperature observed at the body of the pipe is 20°C, highlighting the temperature variation within the system.*

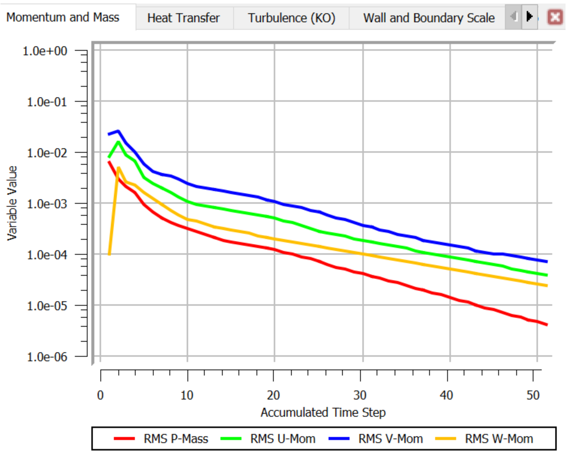
## *Velocity Streamlines*

*Analyzing the data in streamline form, which offers a distinct visual depiction of the flow dynamics, might also help to clarify the findings. It is evident from the streamline visualization that the majority of the streamlines are blue, meaning that most of the system is not experiencing a large temperature change. The majority of temperature changes are found on the right side, indicating that this is the location of the strongest thermal interactions.*

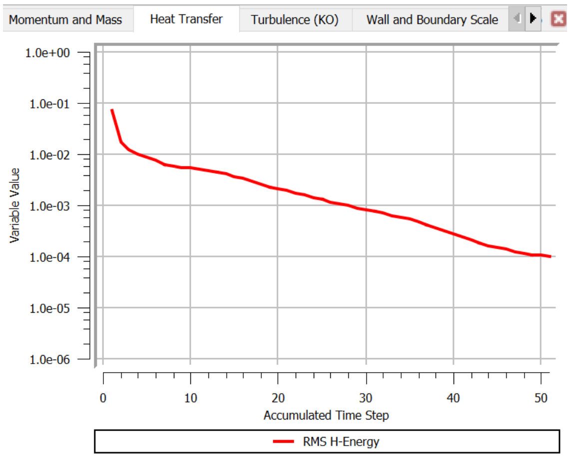
*This observation can be expanded upon to deduce that the velocity at the large inlet is higher compared to the small inlet, resulting in a greater mass of fluid entering the system from the large inlet. This influx of fluid strikes the particles from the small inlet, pushing them towards the walls. This dynamic explains why proper mixing of hot and cold streams does not occur; the forceful movement of the larger mass of cooler water from the large inlet prevents the smaller volume of hotter water from the small inlet from evenly distributing throughout the system.*



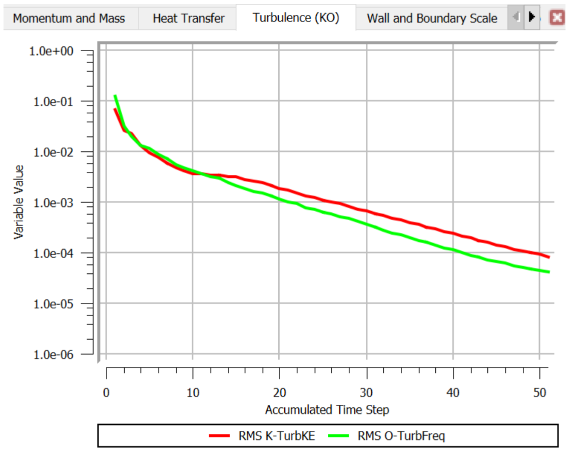

RMS Residuals




Heat Transfer

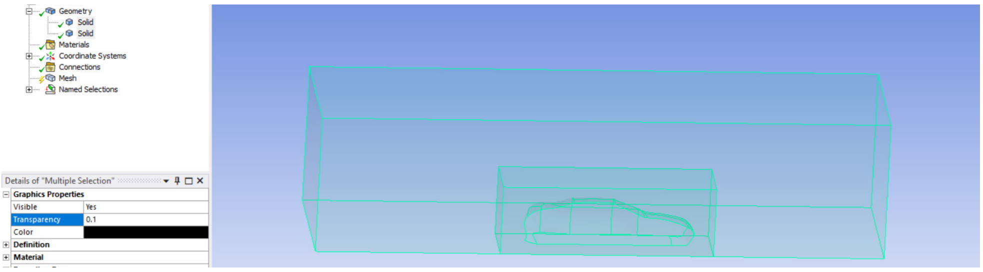



Turbulance

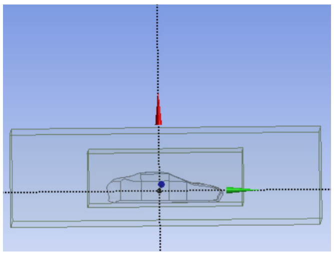


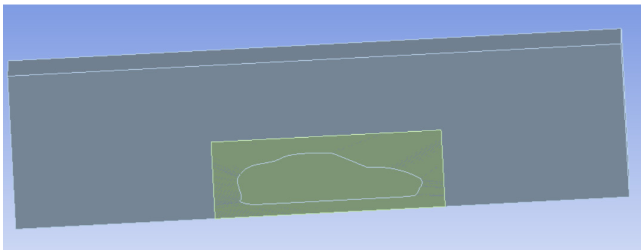
# ***Simulation 4*** - Simple Car Body Simulation

## 4.1 Geometry



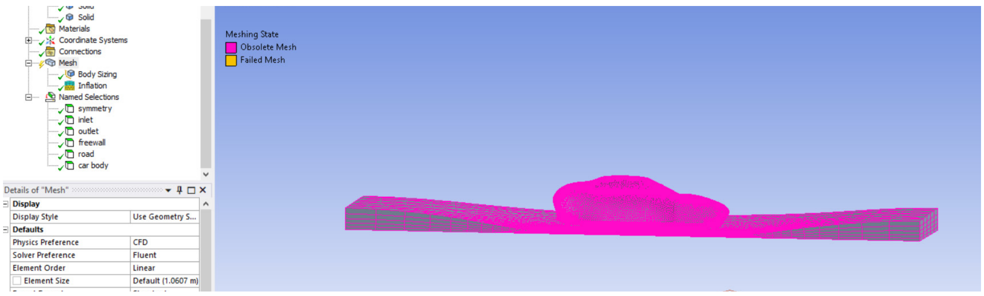
*For simulation setup, I began with a rectangular domain. From this domain, car body needs to be subtracted to create a space through which air could flow around the car, allowing us to simulate air movement over the vehicle's surface.*

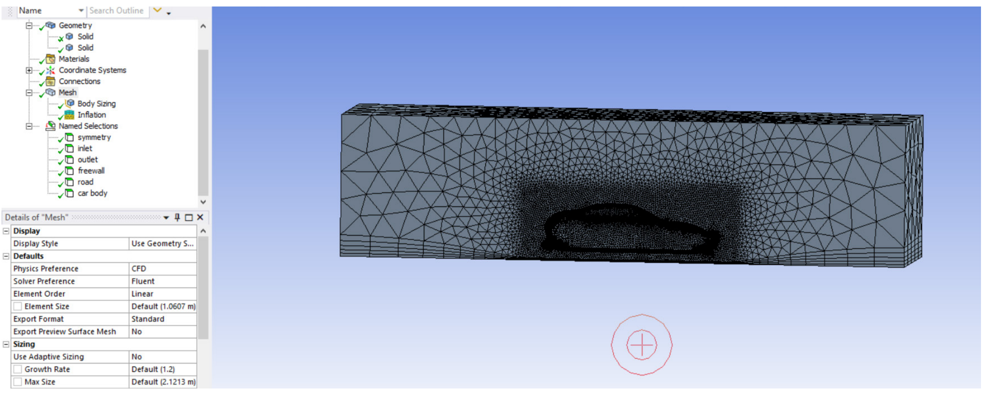


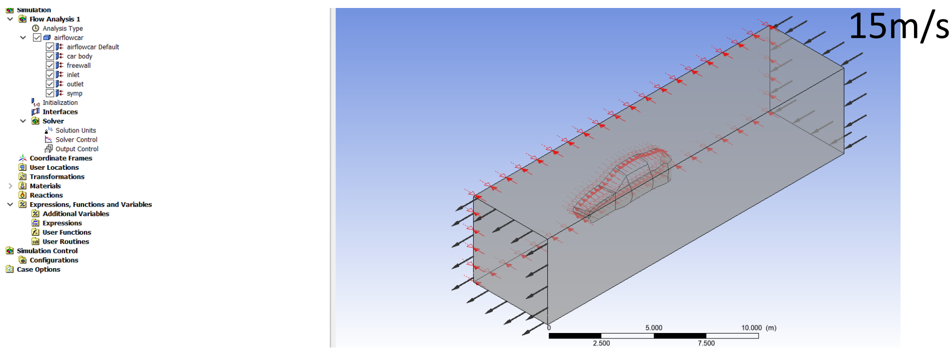


*Following this, I assigned an area as the outlet, which enables the air that has interacted with the car body to exit the domain. This setup provides a controlled environment to analyze the aerodynamic effects on the car, including pressure distribution and airflow patterns.*

## 4.2 Meshing






*In earlier simulations, we employed basic meshing techniques.*

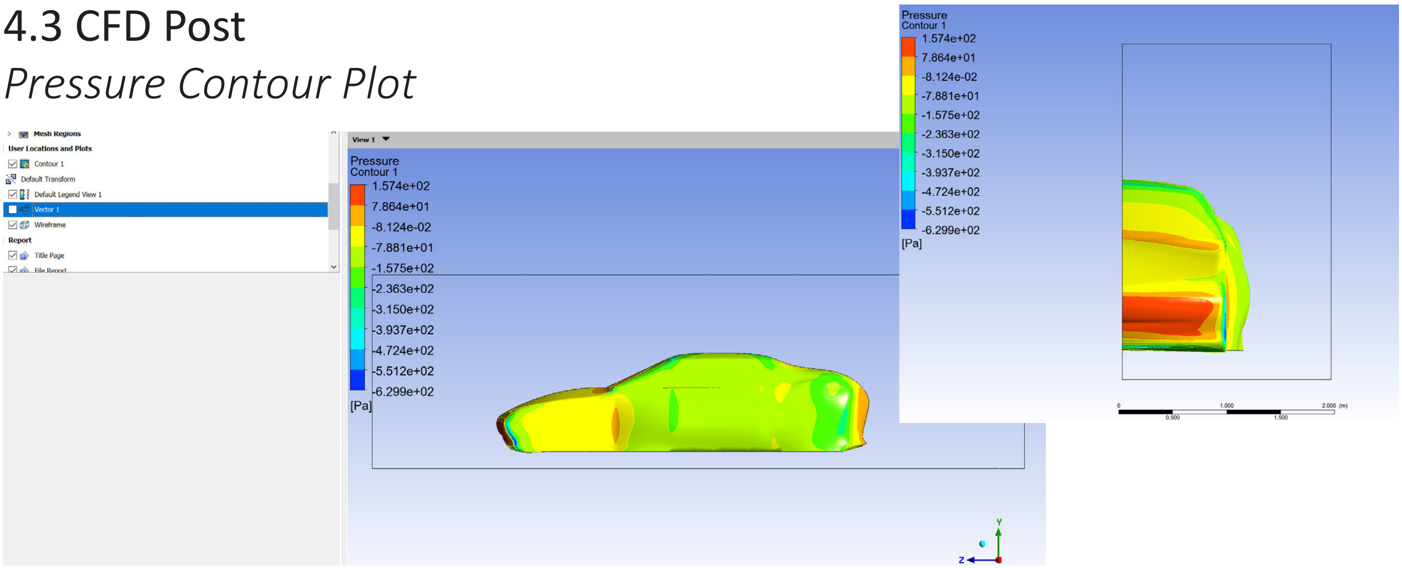
*However, for this real simulation of airflow over a car, I am in need of adopted advanced meshing techniques to ensure a properly defined mesh.*

*This approach was crucial to guarantee that our simulation results would be accurate, reflecting the nuanced aerodynamic interactions between the air and the car's surface.*

*Finally, I assigned an inlet where air is introduced into the domain, directed towards the car body at a velocity of 15 meters per second.*

## 4.3 CFD Post

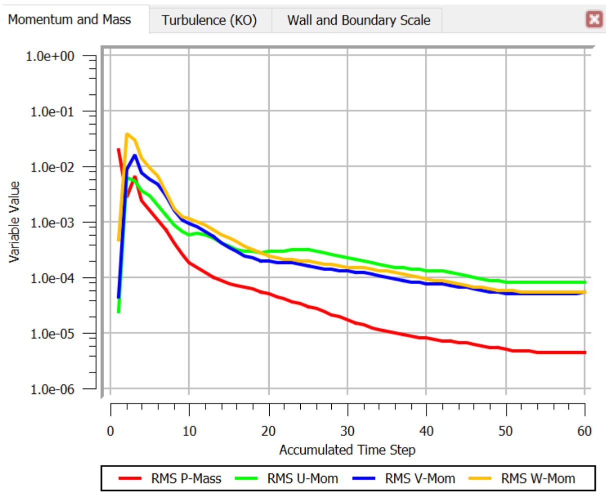
### *Pressure Contour Plot*



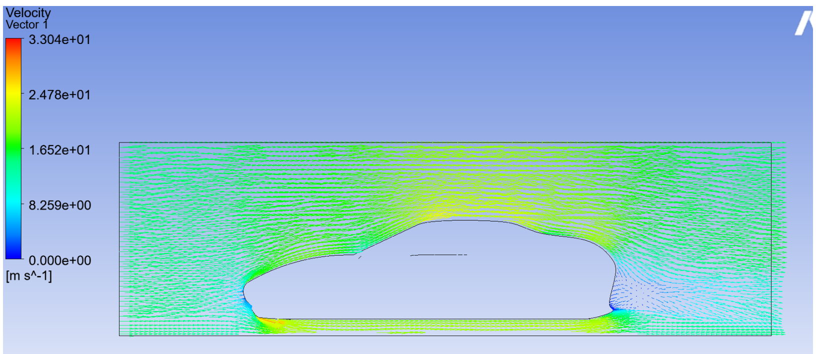

*The car's pressure visualization shows that the bumper has the greatest pressure effecting it. The design of the front end, being more straight and presenting a larger surface area, creates a challenge for air movement.*

*Consequently, when air impacts the bumper, it experiences a significant loss in velocity, resulting in elevated pressure in this region. To mitigate this effect and enhance aerodynamic efficiency, redesigning the front to be sharper could facilitate a gain in maximum velocity, as it would reduce the resistance against the air.*

### RMS Residuals

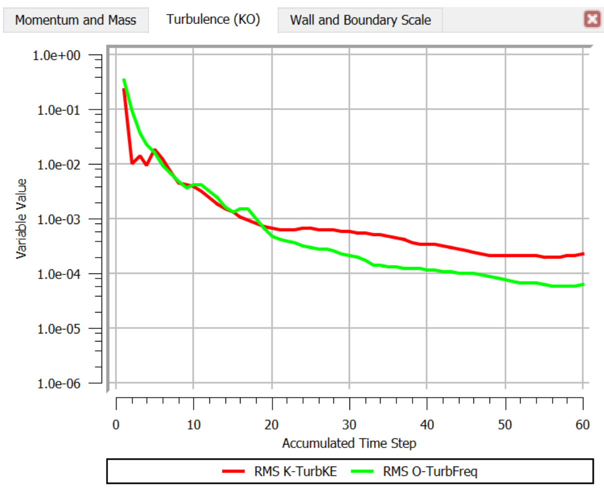



### *Velocity Streamlines*




*Following the impact with the bumper, air particles move along the bonnet and encounter another region of high pressure near the mirror.*

### Turbulence

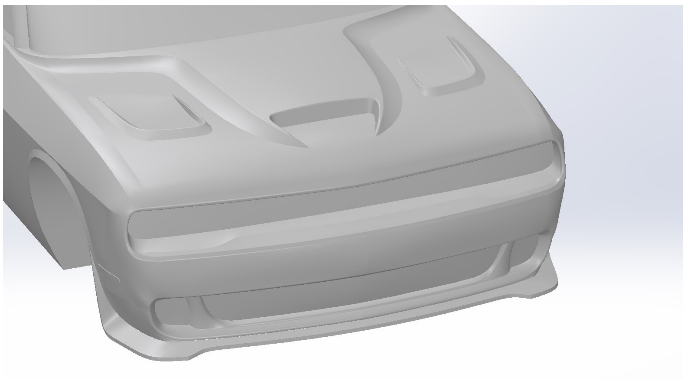


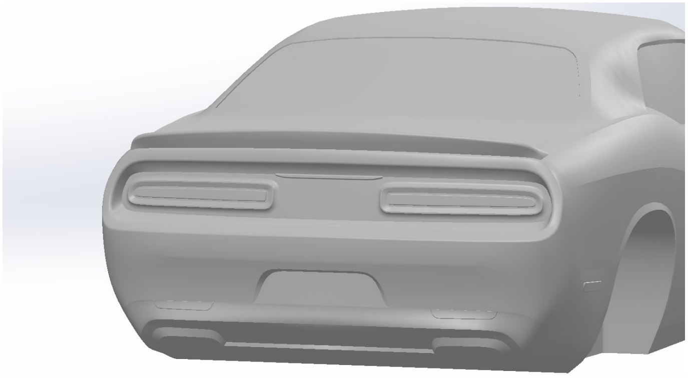

*This increase in pressure is attributed to discontinuities in the car's shape, which disrupt the airflow. To provide a clearer understanding of these dynamics, representing them in vector form can elucidate the relationship between the car's geometry, air flow discontinuities, and the resulting pressure distributions, offering insights into potential areas for aerodynamic optimization.*

# Simulation 5
# ***Simplifying Term 1 Car Model***

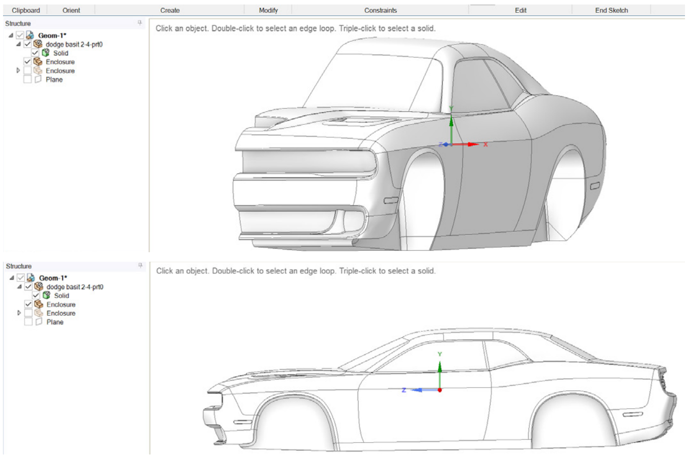
*Through rich data and visuals, CFD simulation and analysis of a simplified car model using the ANSYS application offer a thorough insight of aerodynamic performance. Through the analysis of pressure contours, velocity streamlines, and velocity contours, we can optimize the vehicle design for increased safety, performance, and efficiency.*

## Geometry

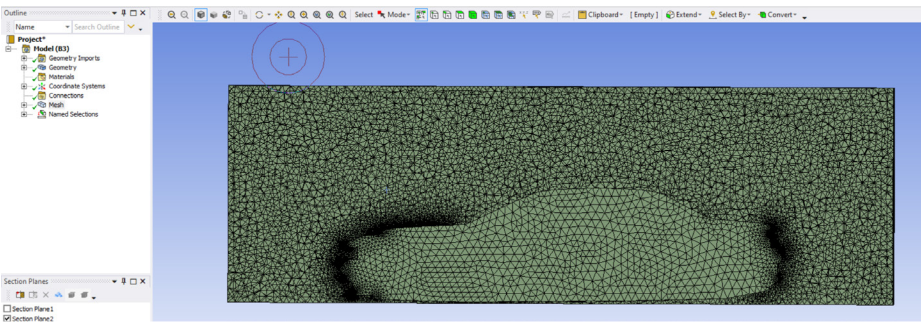


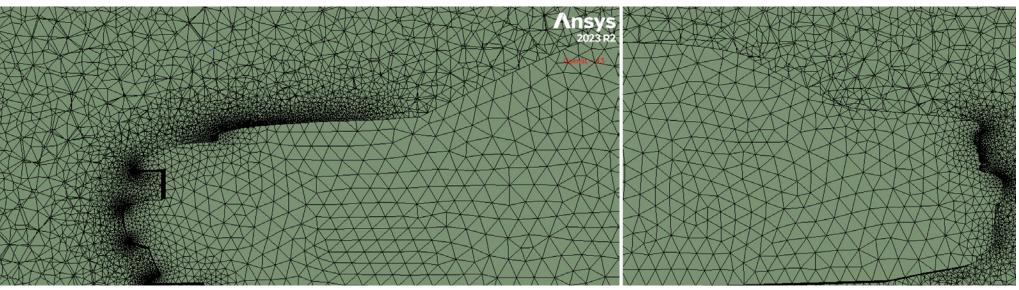


*At first, I simplified the "Dodge Challanger SRT Helcatt 2015" Solidworks CAD model, which I had done in the first term. This is a very important step to take to get the best results out of the simulation as it lets the application to focus on the aerodynamic properties of the car's form without being slowed down by complex details that have little influence on the overall flow patterns. I removed all the small details, such as the front grill, wheels, exhaust, and so on. Then I imported the simplified version into the ANSYS Workbench. Here are the model screenshots taken from SpaceClaim.*



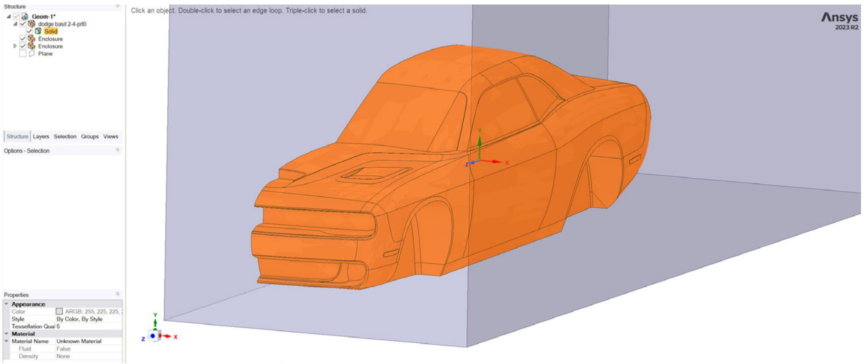
## Meshing





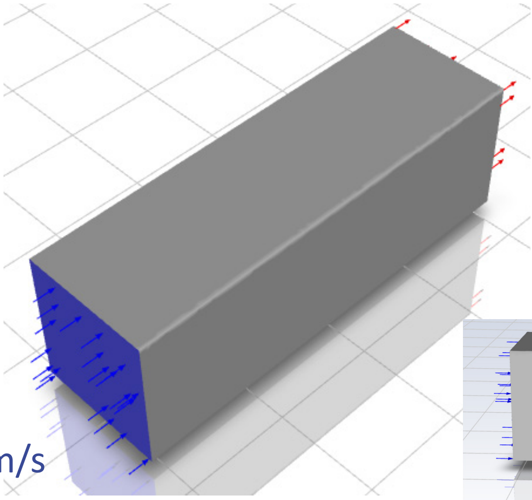

*I used Inflation Layers and Body Sizing to create an effective mesh.*

*Inflation layers is a approach of creating several layers of mesh that progressively increase in size from the surface of the geometry outwardly to the fluid domain. With this method, the boundary layer's velocity gradient and pressure distribution,where the flow characteristics change significantly over a short distance, can be precisely captured. Body sizing lets you determine the ideal mesh element size for particular bodies or sections inside the simulation area. This technique makes it possible to regulate the mesh density in regions of interest, allowing for coarser meshes in areas less crucial to the analysis and finer meshes where high precision is required.*




*We can see the simplified model of the car inside the rectangular domain. From this domain, we subtracted the car body to create a space through which air could flow around the car, allowing us to simulate air movement over the vehicle's surface just as we have done in Tutorial 4.*

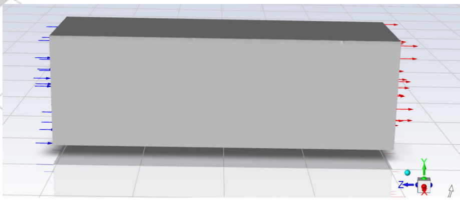
# CFD Simulation & Analysis on Simplified Model



15m/s

*In the simulation setup, one side of the rectangular domain was designated as the inlet, while the opposite side was assigned as the outlet.*

*This configuration creates a controlled environment that facilitates the analysis of aerodynamic impacts on the vehicle, including the examination of pressure distribution and airflow patterns, which are the data I am aiming to get.*

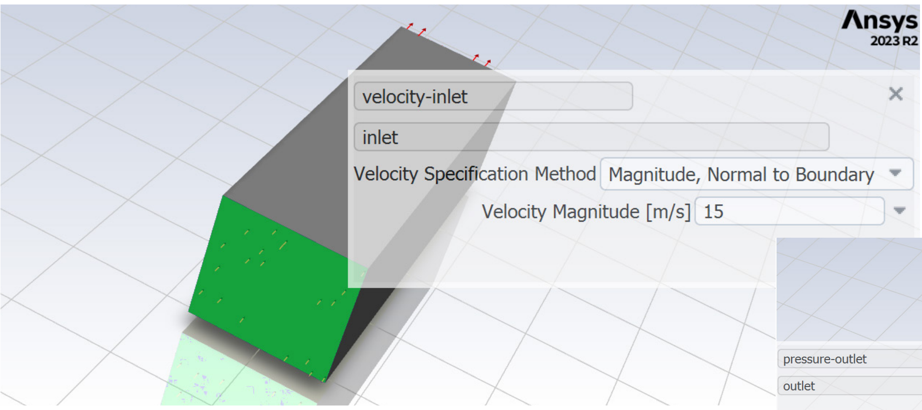


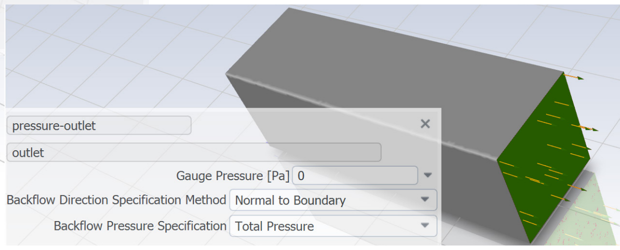
**Reference Values**

Compute from

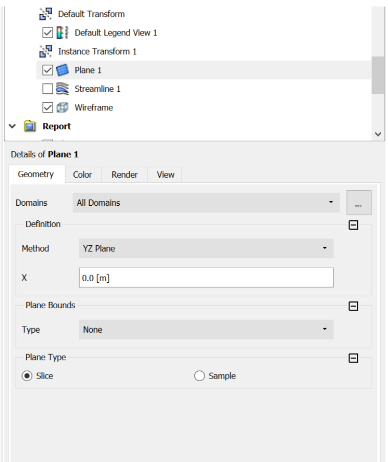
**Reference Values**

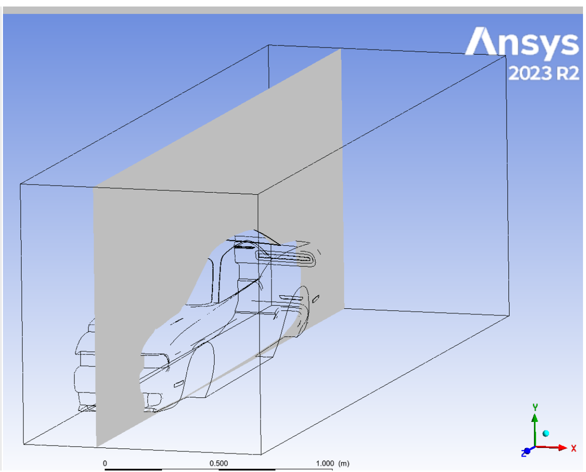
| | |
|---|---|
| Area [m$^2$] | 1 |
| Density [kg/m$^3$] | 1.225 |
| Enthalpy [J/kg] | 0 |
| Length [m] | 1 |
| Pressure [Pa] | 0 |
| Temperature [K] | 288.16 |
| Velocity [m/s] | 1 |
| Viscosity [kg/(m s)] | 1.7894e-05 |
| Ratio of Specific Heats | 1.4 |
| Yplus for Heat Tran. Coef. | 300 |




## Creating the Sampling Plane




*A sampling plane is a plane with evenly spaced sampling points on it. It allows us to see a detailed visualization of flow features across the plane.*

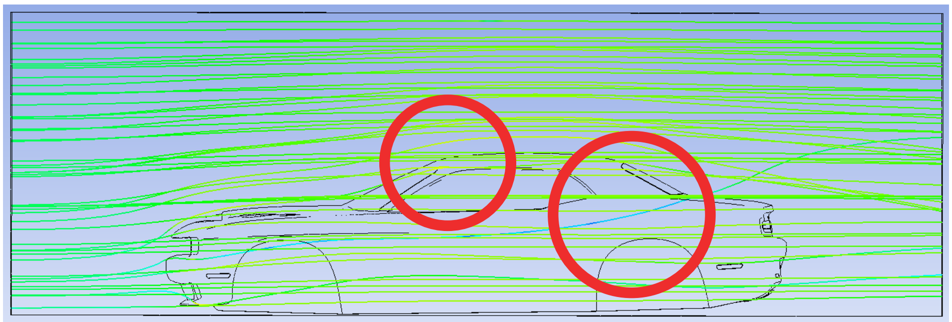
*I created this sample plane and placed it across the car model.*

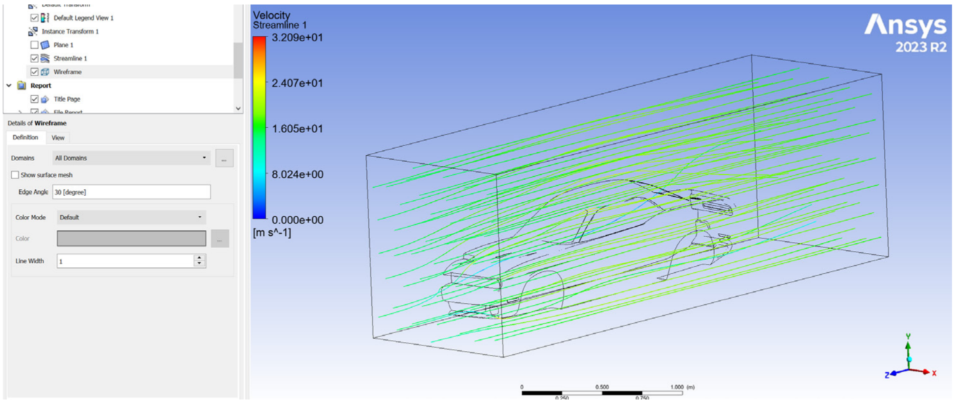
*If necessary, focused investigations of particular areas of interest can be conducted using sampling planes as an alternative to the sometimes resource-intensive analysis of the entire volume.*

# Results

## Velocity Streamlines

*Velocity Streamlines are the trajectory that air particles take as they pass along the vehicle is seen in this graphic. Streamlines aid in the visualization of flow direction and velocity by emphasizing regions of continuous flow as well as those experiencing separation and recirculation. Through the examination of these patterns, we are able to pinpoint undesired aerodynamic elements, like regions where drag might build up.*



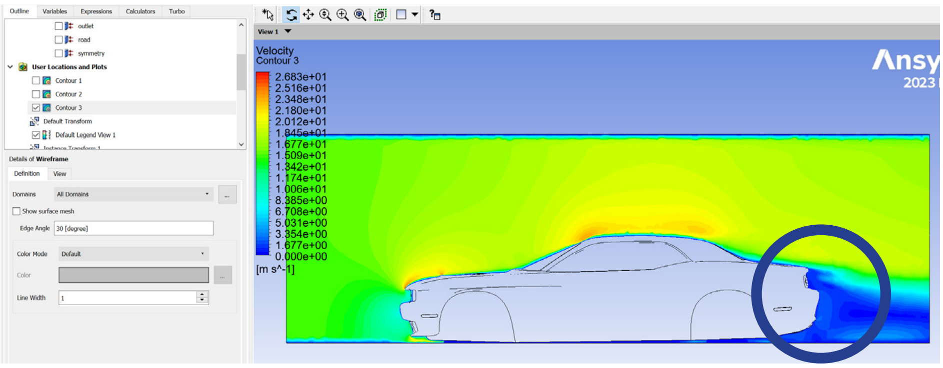


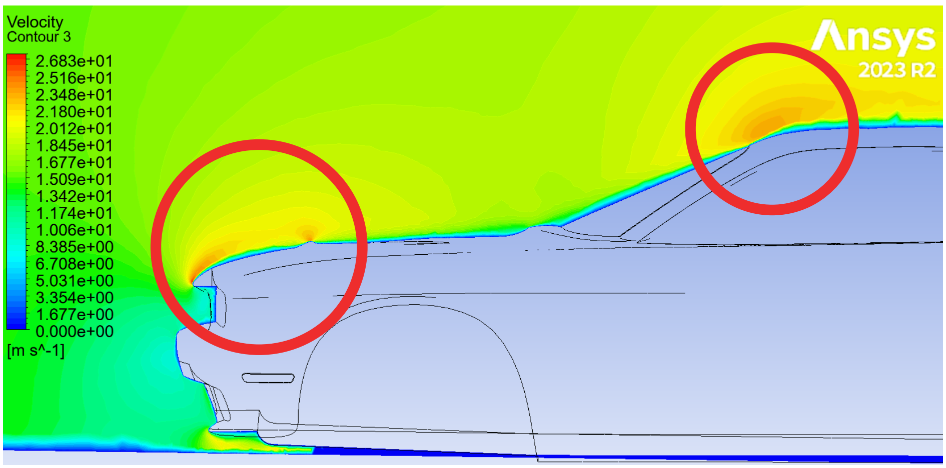

*The analysis of streamlines around the rear wheels demonstrated that slower airflow patterns due to the vehicle's design.*

*This design feature plays a crucial role in adding downforce at the rear end of the vehicle by creating areas of low air pressure above it.*

*Such an increase in downforce is advantageous for enhancing traction and stability, particularly in the rear-wheel-drive configuration of the car, contributing positively to its overall performance and handling characteristics.*

## Velocity Contour





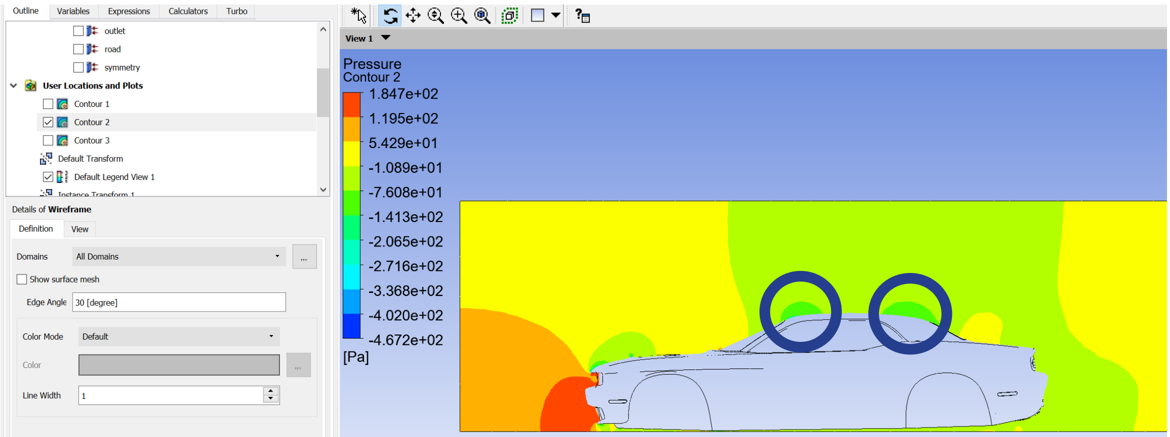

*A color-coded plane of the flow velocity magnitude in various vehicle portions is provided by Velocity Contour Plots.*

*The speed gradients surrounding the vehicle can be identified with this figure, highlighting areas where the form of the vehicle could be improved to minimize drag or maximize airflow for cooling.*

*We can determine the parts of the body where the velocity of the air particles increases by checking the red cirles and the blue circles, which indicate the areas where the particles slow down.*

*We can see the Slipstream forms at the rear end of the model, where the*

# Pressure Contour Plot



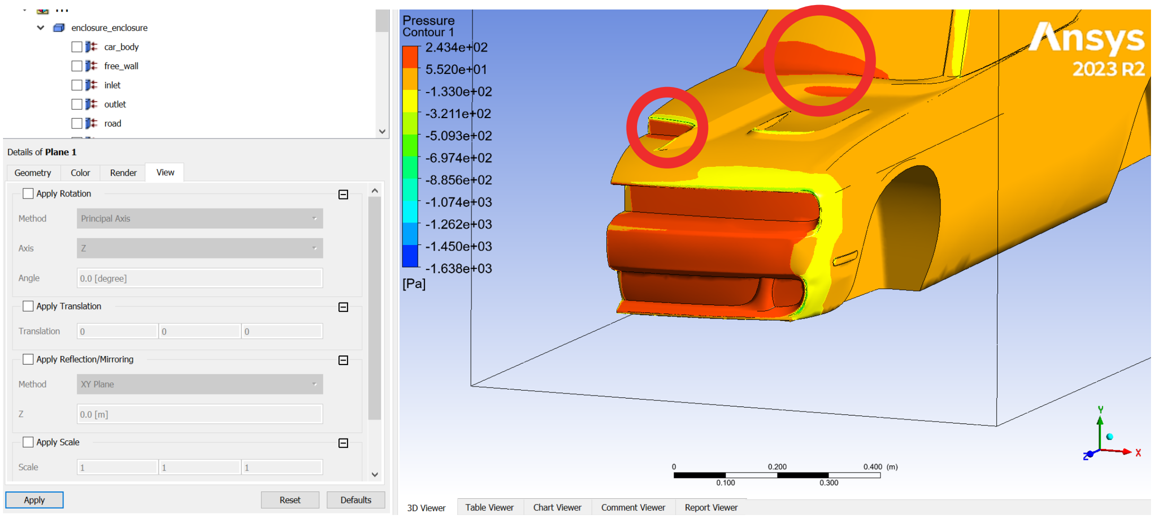

*Laminar Air Flow:*

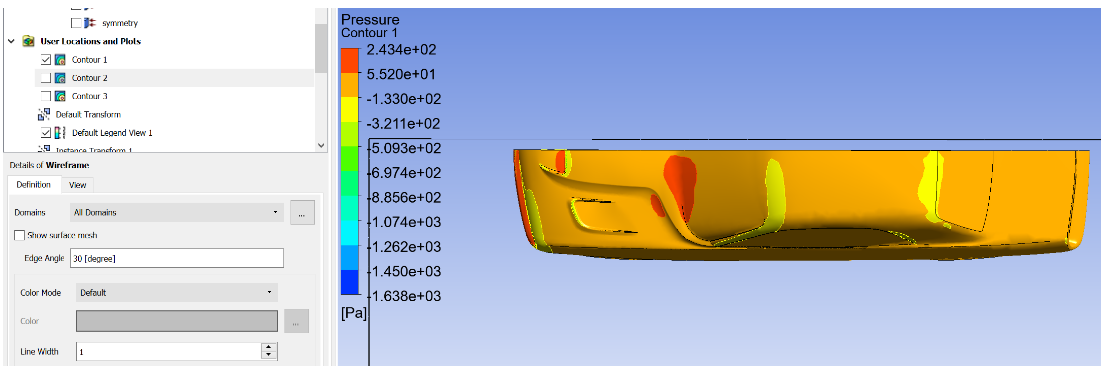
***Flat surfaces aligned parallel to the airflow experience an effect known as laminar flow, where air particles flow smoothly over the surface with minimal resistance.***

*Pressure Contour Plots provide a visual depiction of the pressure distribution throughout the surface of the vehicle, much like velocity contours do.*

*Low-pressure places may indicate flow separation, while high-pressure areas may indicate possible lift or aerodynamic resistance spots.*

*Optimizing aerodynamic performance, lowering drag, and increasing downforce for better grip and handling all depend on an understanding of pressure distribution.*




*As we can see, the front of the car, the closed air vent on the hood, and the part where the hood and the front glass meet are colored red, indicating high pressure.*

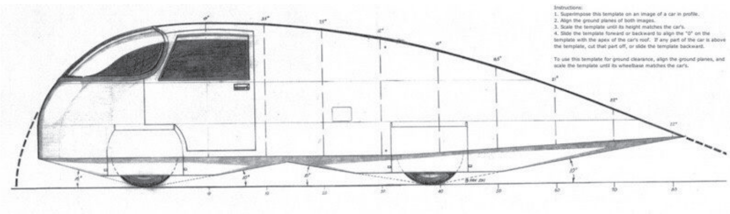
*Form drag happens as a result of compressed air particles creating a high-pressure area in front of the car. Because it must use more effort to push through this compressed air, thus the car uses more gasoline and is less efficient.*

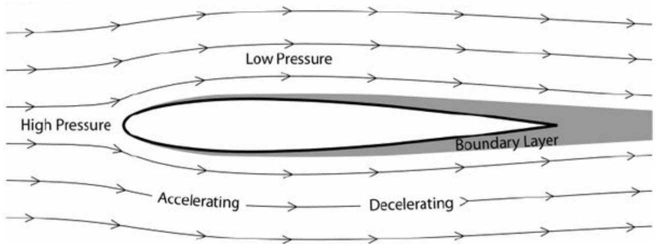
*On the other hand, the rest of the car surface has an orange color, indicating tolerable levels of air pressure.*

# Discussion and Implications

*Simulation results indicate that the front end of the car, being the initial point of interaction with air particles, features a notably flat surface. This design increases air pressure and consequently, aerodynamic drag, which is not beneficial for performance-oriented vehicles aiming for a low drag coefficient.*

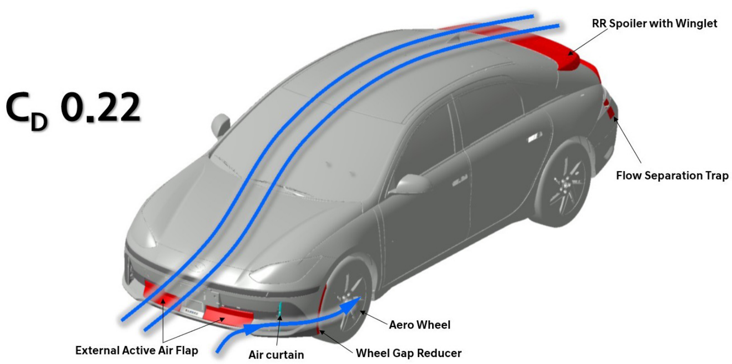
*Theoretically, the optimal shape for a car to achieve the lowest possible drag coefficient and superior aerodynamics is similar to a teardrop, mirroring the principles of airfoil designs. This shape is designed to streamline airflow, minimizing resistance and enhancing efficiency.*


Teardrop car 1.



Teardrop Shape 2.

*Let's take the Hyundai Ioniq 6 as an example. It has a very aerodynamic design since it really looks alike with the teardrop shape.*



Ioniq Simulation 3.


Ioniq 6 Backend 4.

*This principle of aerodynamics is similarly applied in the design of airplane wings, which are engineered to be aerodynamic for reducing drag while also providing lift. These wings include various angles and features to fulfill both roles effectively.*

*In the context of high-performance vehicles, the significance of aerodynamic design cannot be underestimated. For instance, without its powerful 6.2-liter supercharged V8 engine delivering 707 horsepower, the impact of aerodynamic drag would be considerably much greater. However, such a powerful engine can compensate for increased drag, ensuring that performance remains unaffected.*

*It's also important to remember that a vehicle's length affects its aerodynamic characteristics; in general, longer cars have smoother aerodynamics since they generate less drag from their more gradual airflow separation.*

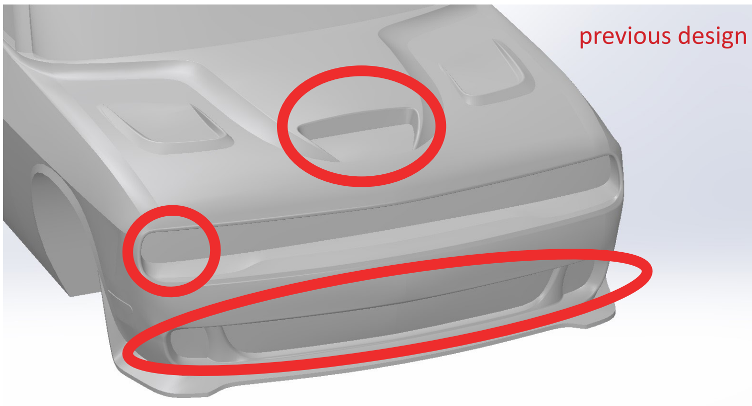
*Therefore, modifying the car's front end to be more rounded rather than flat could be beneficial to improve aerodynamics and minimize drag. A change of this kind would bring the vehicle's shape closer to the teardrop*

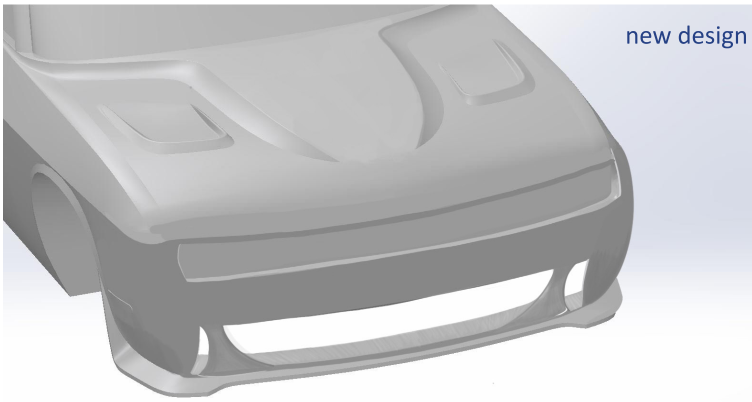
*Based on findings from previous simulations and research on optimal drag coefficient principles, I have made a new design for the car's front end to enhance its aerodynamics significantly.*

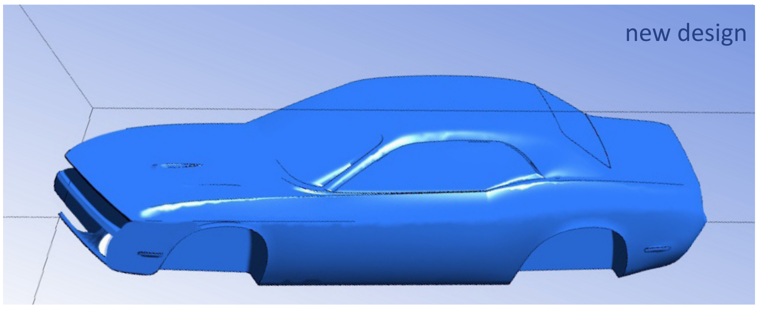
*This redesigned front end includes three strategically placed openings, letting airflow to pass through with greater ease, which reduces the need for the air to exert force directly on the car's surface. A more rounded overall form has been achieved, corresponding to the aerodynamically efficient teardrop profile. Since the front end of the vehicle makes initial contact with flowing air, this adjustment minimizes the frontal area exposed to high air pressure in addition to reducing aerodynamic drag.*

*To accommodate this new form, alterations have been made to minimize the space allocated for front lights and engine cooling filters. At this stage, the primary objective is to evaluate the impact of surface design changes on aerodynamics, setting aside engine cooling considerations for later development steps.*

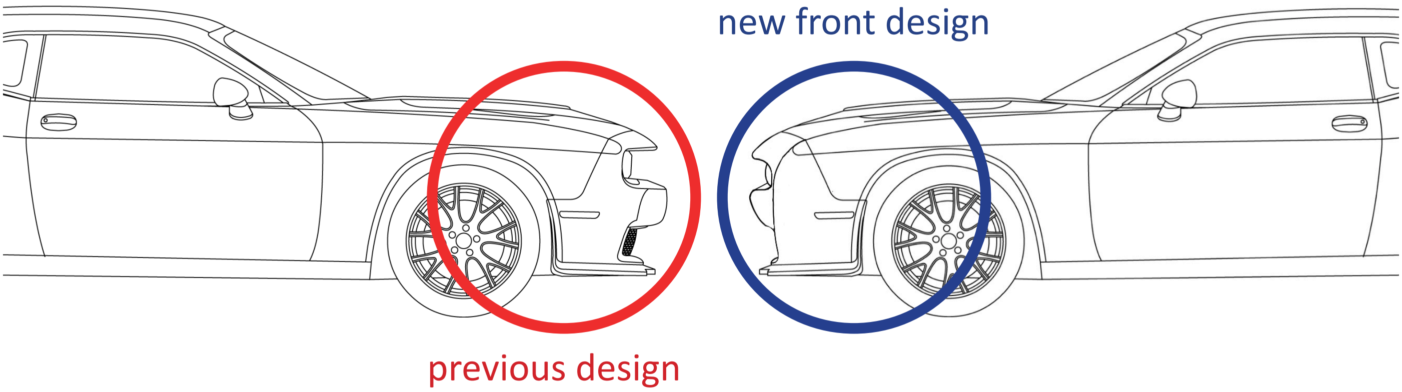
*The transformation of the car's frontal area will lead to significantly lower drag, as demonstrated by its previously identified as the highest air-pressured surface. The introduction of a rounded front, resembling a teardrop shape, and the addition of three blank spaces are expected to contribute to easier airflow, generating lower air pressure above the front lip's surface. This design is expected to enhance downforce on the front end—a desirable outcome, especially for muscle cars known for their power and performance.*



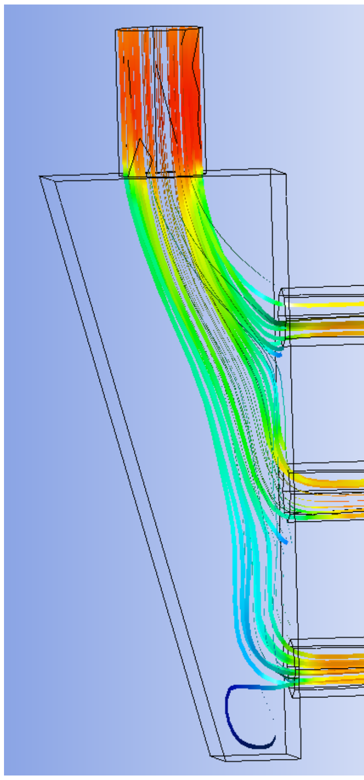




*Given that this car has been equipped with a rear-wheel-drive system and powered by a powerful engine, it has a tendency to lift the front end during quick acceleration, such as in racing scenarios. The precision of steering and the efficacy of launches may be compromised by this lifting action. By adding greater downforce to the front wheels, the suggested design modifications seek to address these problems by providing more stable road contact and better steering responsiveness, especially at high speeds.*



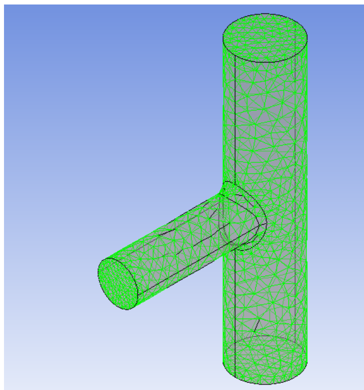
# Conclusion

*In conclusion, the series of Computational Fluid Dynamics (CFD) simulations conducted using ANSYS have been done for all 4 tutorials and on simplified Term 1 car model.*

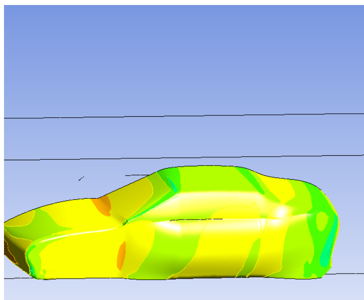


*Using an iterative process of study, modeling, and design improvement, we have effectively determined and addressed the main causes of aerodynamic drag.*

*It has been much easier to reduce drag and increase downforce with the adoption of a more aerodynamically advantageous front-end design that features rounded forms and strategically placed air channels. This is especially important for the performance of muscle cars.*



*These results highlight how useful virtual simulations are for expediting the design process and enabling quick prototyping and optimization without the high costs and lengthy lead times associated with physical models.*



*My work not only shows how ANSYS can be used to facilitate complex aerodynamic analyses, but also how careful design changes may result in considerable performance gains.*

# References


Teardrop car 1.

“Is the Shape of a Raindrop the Most Aerodynamic Shape? Wouldn’t the Fluid Change Shape as the Air Acts on It to Be Perfectly Aerodynamic?” Quora, 2019, www.quora.com/Is-the-shape-of-a-raindrop-the-most-aerodynamic-shape-Wouldnt-the-fluid-change-shape-as-the-air-acts-on-it-to-be-perfectly-aerodynamic. Accessed 20 Mar. 2024.

Teardrop Shape 2.

“What Is the Most Aerodynamic Shape (That Doesn’t Change Shape or Need to Contain Anything)?” Quora, 2019, www.quora.com/What-is-the-most-aerodynamic-shape-that-doesnt-change-shape-or-need-to-contain-anything. Accessed 20 Mar. 2024.

Ioniq Simulation 3.

“Hyundai Reveals Secrets behind Ioniq 6’S Top Aerodynamic Efficiency.” InsideEVs, insideevs.com/news/621207/hyundai-reveals-secrets-behind-ioniq-6-top-aerodynamic-efficiency/.

Ioniq 6 Backend 4.

“Hyundai 2023 IONIQ 6 SE Long Range RWD Achieves EPA-Estimated Driving Range of 361 Miles.” Green Car Congress, 2023, www.greencarcongress.com/2023/02/20230201-ioniq.html. Accessed 20 Mar. 2024.

“Ride like the Wind: Hyundai’s IONIQ 6 Takes “Drag” to a New Low and Range to a New High.” Www.hyundai.news, www.hyundai.news/uk/articles/press-releases/hyundais-ioniq-6-takes-drag-to-a-new-low-and-range-to-a-new-high.html. Accessed 20 Mar. 2024.

Talis Reading List- Online Book:

Yasuki Nakayama. Introduction to Fluid Mechanics. San Diego, Elsevier Science, 4 Jan. 2018, ebookcentral.proquest.com/lib/brunelu/reader.action?docID=5212785.

Bora Sen- 2017194

Brunel University London