Abstract:

This project aims to investigate the aerodynamic behavior for a blunt boxy vehicle with a drag coefficient vale of around 0.608. The computation fluid dynamics will be performed by evaluating the influence of geometric and flow parameters. Some key parameters under study in this project are lift, drag, pressure distribution and heat transfer.

Software Used:

- Ansys Work Bench – Fluid Flow (Fluent)

Body Selection:

As per the project instructions, the last two digits of the CMS IDs of all members were averaged. The calculations yielded the following value:

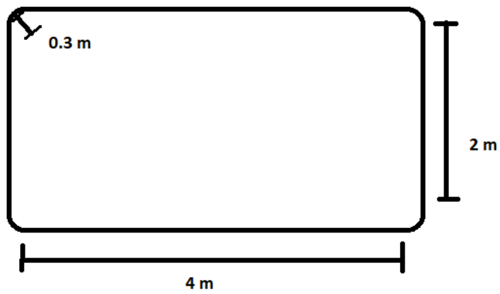
$$\frac{32\ +\ 49\ +\ 47\ +\ 95\ +\ 81}{5} = 60.8$$

So, the drag coefficient will be **0.608**. For this value of C_d, the corresponding bodies are those having rounded edges with blunt overall shapes which cause significant flow separation due to non-streamline shapes. This separation leads to large turbulent wake regions behind the vehicle, hence increasing the pressure drag. Since these vehicles do not allow air flow to stay attached to their surface, hence increasing drag and reducing fuel efficiency.

Body Geometry and Schematics:

The selected body shape is a rectangular prism with rounded edges, having dimensions of **4 m in length, 2 m in width**, with a **corner radius of 0.3 m**. This geometry is chosen to align with a drag coefficient (Cd) of approximately **0.608**, a value characteristic of blunt bodies with some degree of streamlining. The rounded edges serve to reduce flow separation by minimizing sharp discontinuities, thereby decreasing pressure drag compared to a fully sharp-edged body. Despite the rounding, the shape remains sufficiently blunt to generate a considerable wake region, leading to a relatively high drag coefficient.

The following image shows the 2D Schematic of the body under study:



Parameters Under Evaluation:

The parameters under evaluation in terms of geometry and fluid characteristics include:

- **Angle of Attack:** The orientation of the body relative to the flow direction, influencing lift, drag, and flow separation.

- **Inlet Pressure:** The pressure of the fluid entering the domain, affecting pressure distribution and aerodynamic forces.

- **Inlet Velocity:** The speed of the fluid at the inlet, directly impacting drag, lift, and wake formation.

- **Body Length:** The longitudinal length of the body, determining the extent of surface exposure to airflow and wake size.

- **Body Width:** The lateral width of the body, affecting the frontal area and the magnitude of aerodynamic forces.

- **Curvature of Edges:** The degree of rounding at the edges, reducing sharp flow separation and influencing drag.

Simulation Results:

- ### Iteration 1:

For this iteration, the dimensions of the body were **4 x 2 m,** and the corner radius was **0.3 m.** The velocity of fluid at inlet was **200 m/s**. These conditions were analyzed for steady state, transient state as well as heat transfer in this iteration. The mesh size for this iteration was **0.25m** to ensure accuracy in the determined results from the analysis.

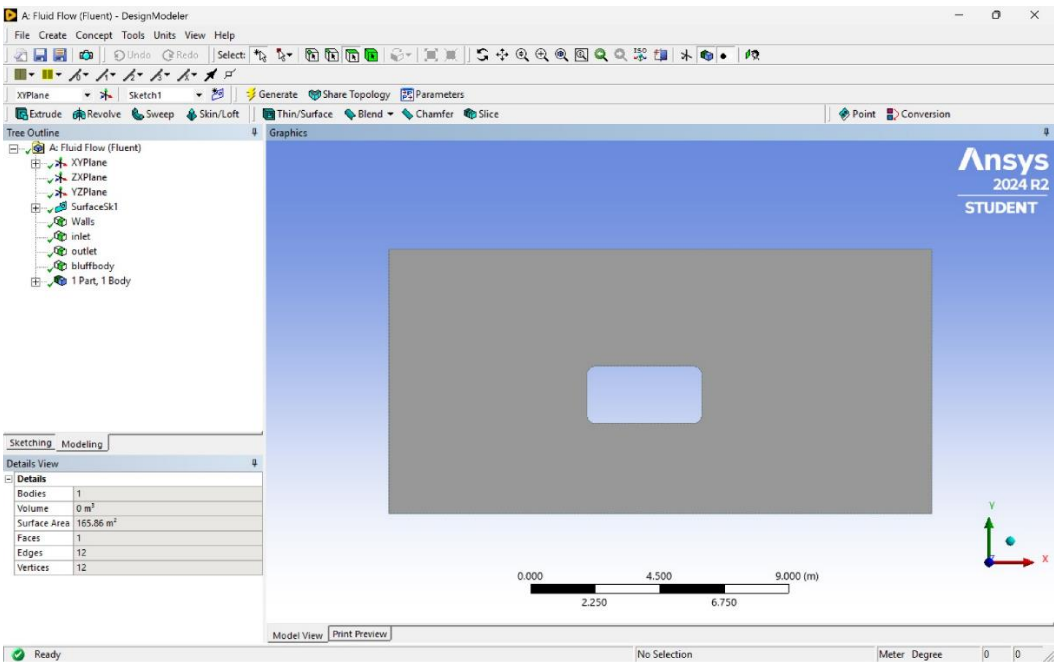


Figure 1 "Geometry Construction of Body in ANSYS"

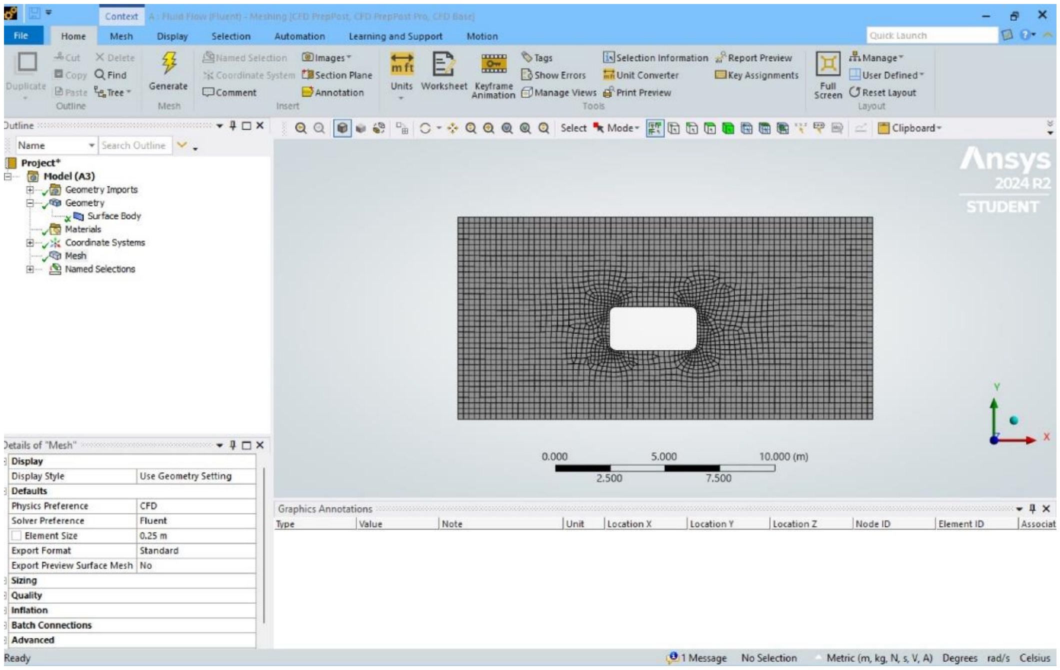


Figure 2 "Meshing on Body"

Iteration 1.1:Steady State Analysis

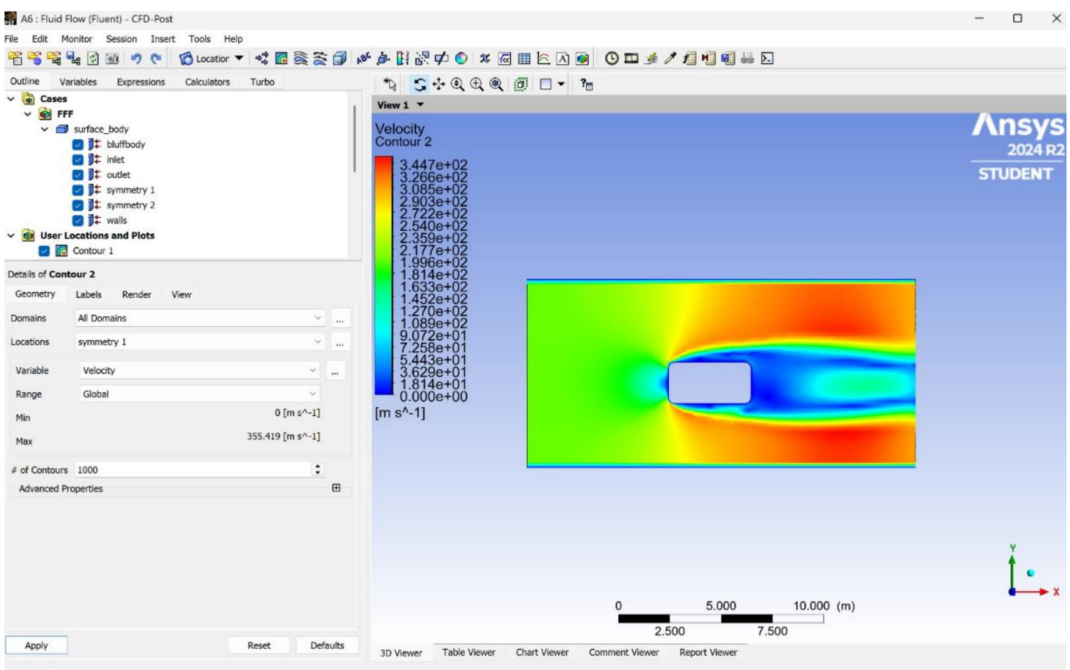


Figure 3 "Velocity Contour on Body"

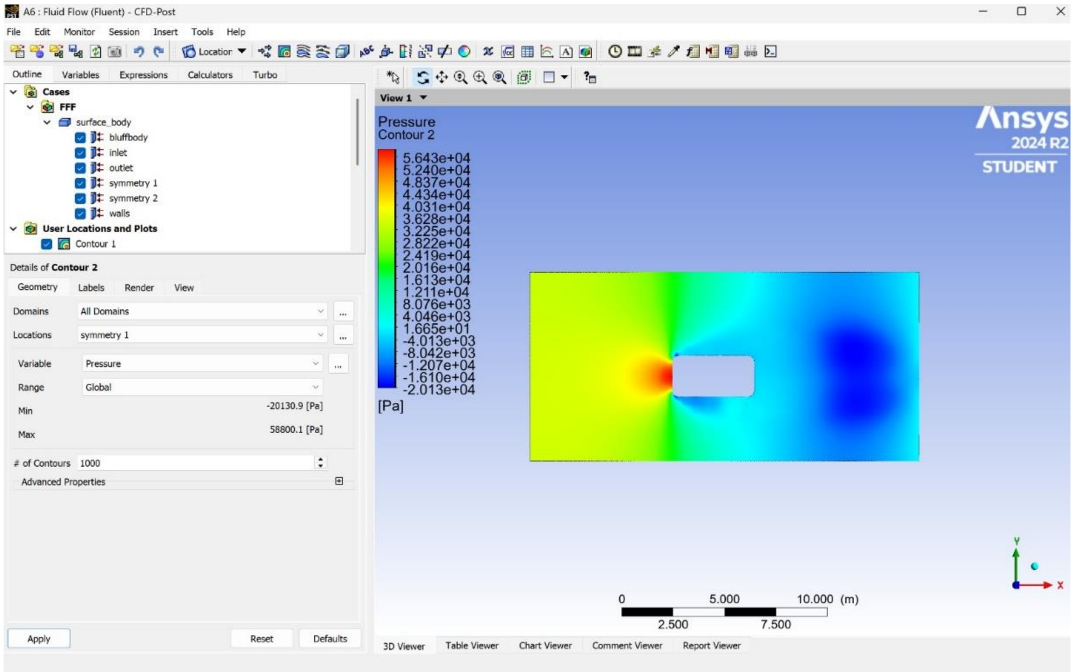


Figure 4 "Pressure Contour on Body"

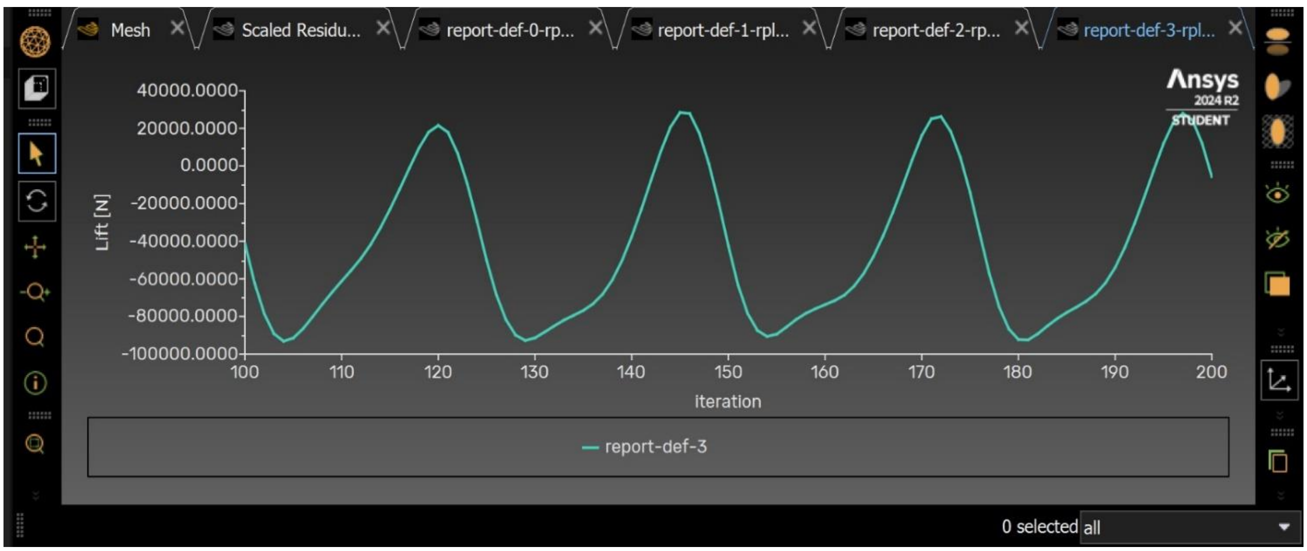


Figure 5 "Lift Force Graph"

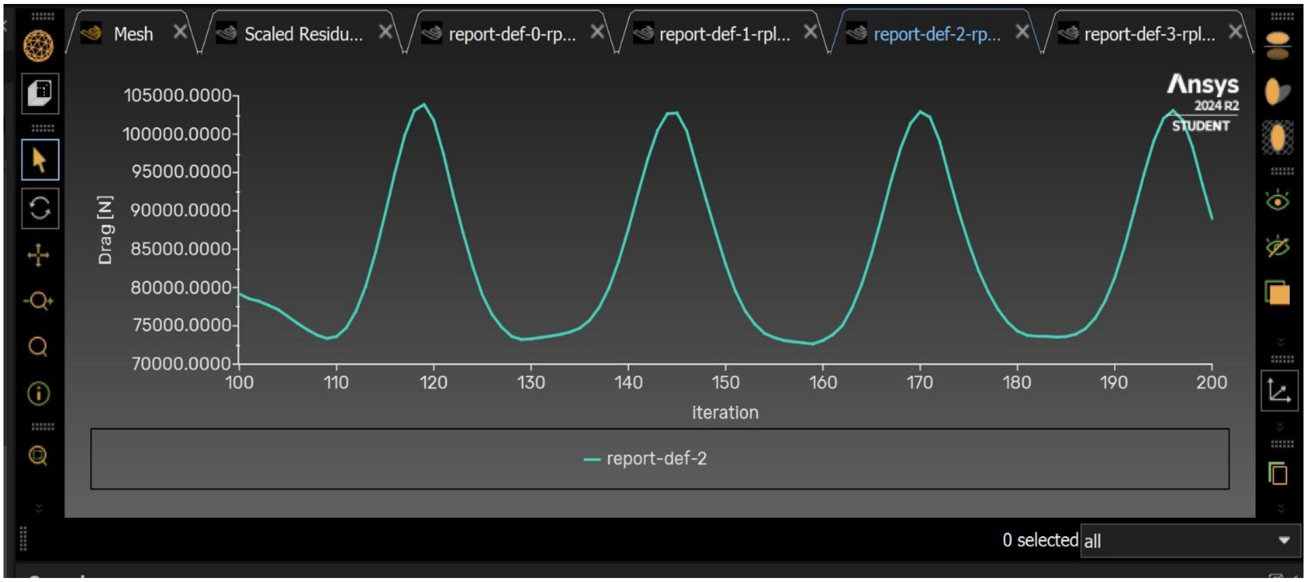


Figure 6 "Drag Force Graph"

Iteration 1.2: Transient State Analysis

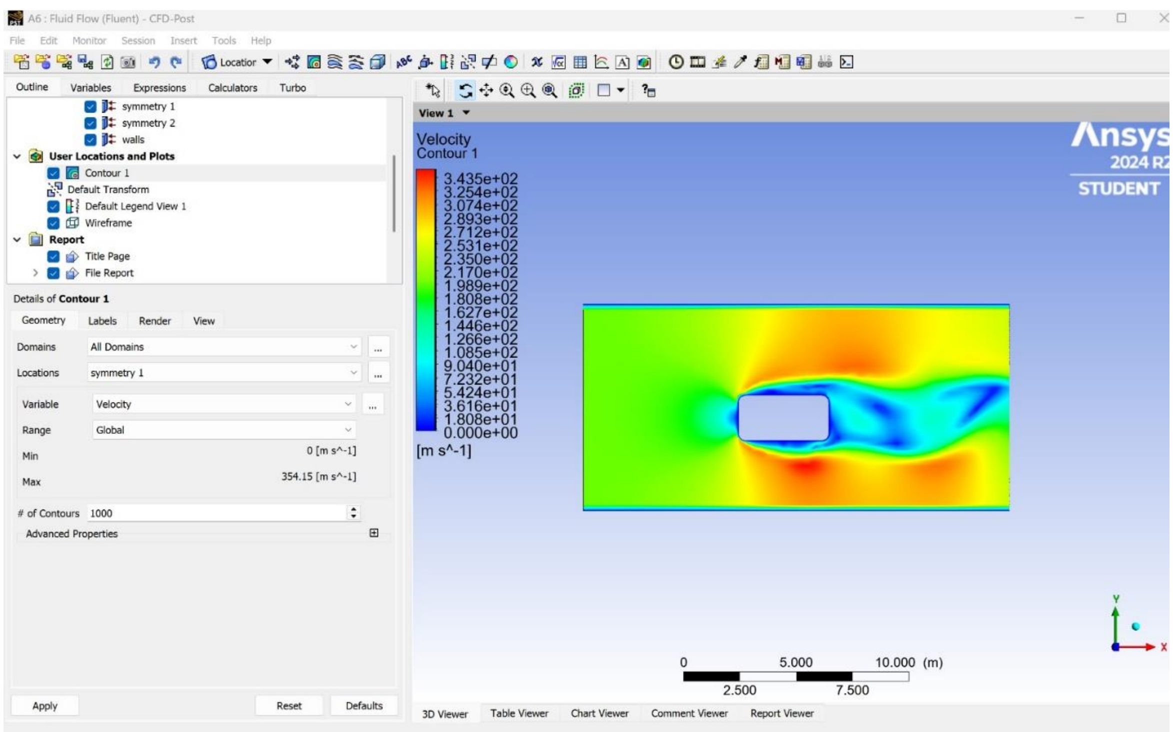


Figure 7 "Velocity Contour"

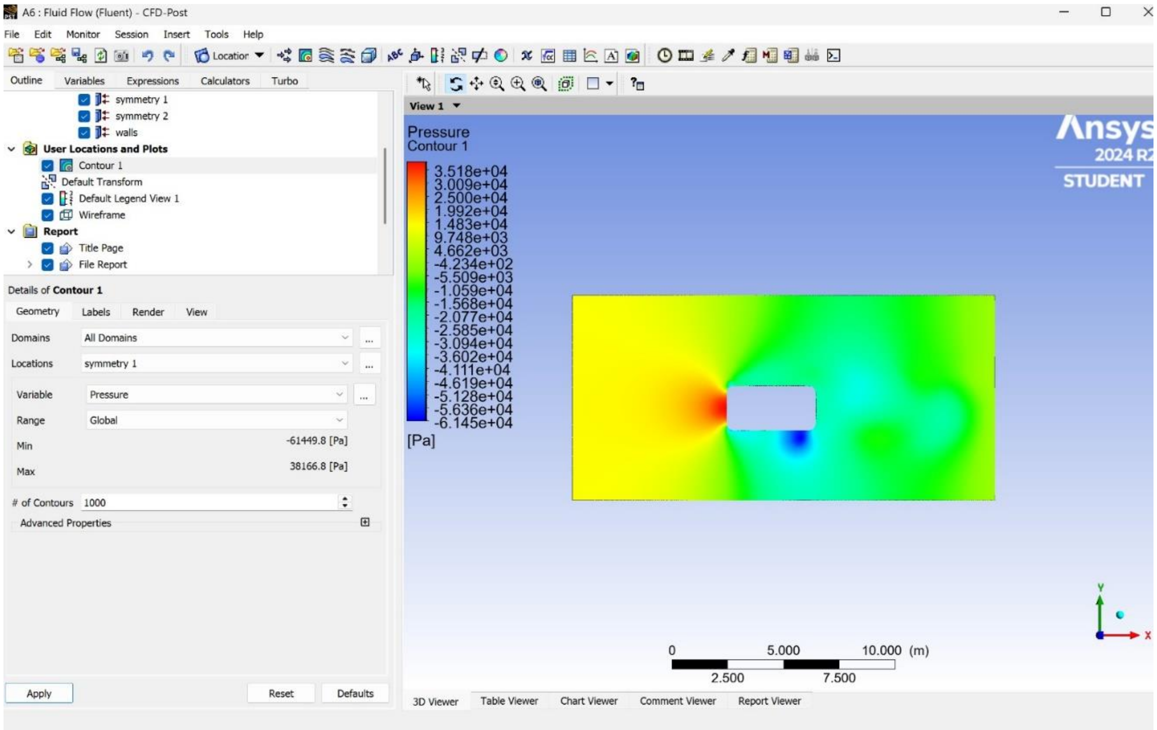


Figure 8 "Pressure Contour"

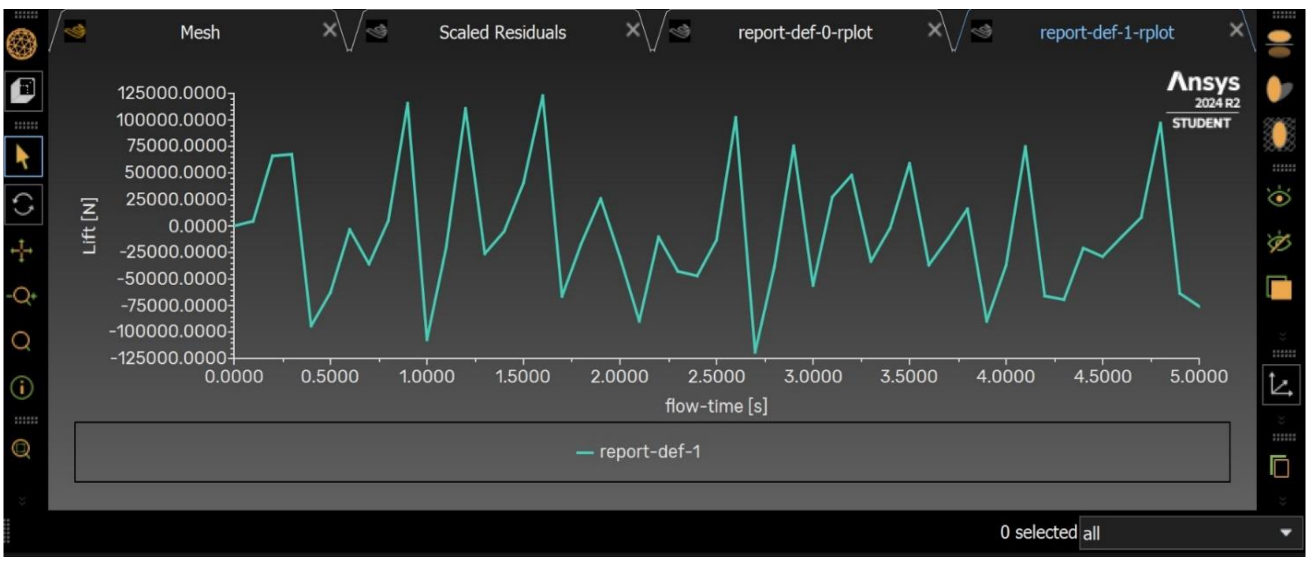


Figure 9 "Lift Force Graph"

Figure 10 "Drag Force Graph"

Iteration 1.3: Heat Transfer Analysis

Figure 11" Drag Force Graph"

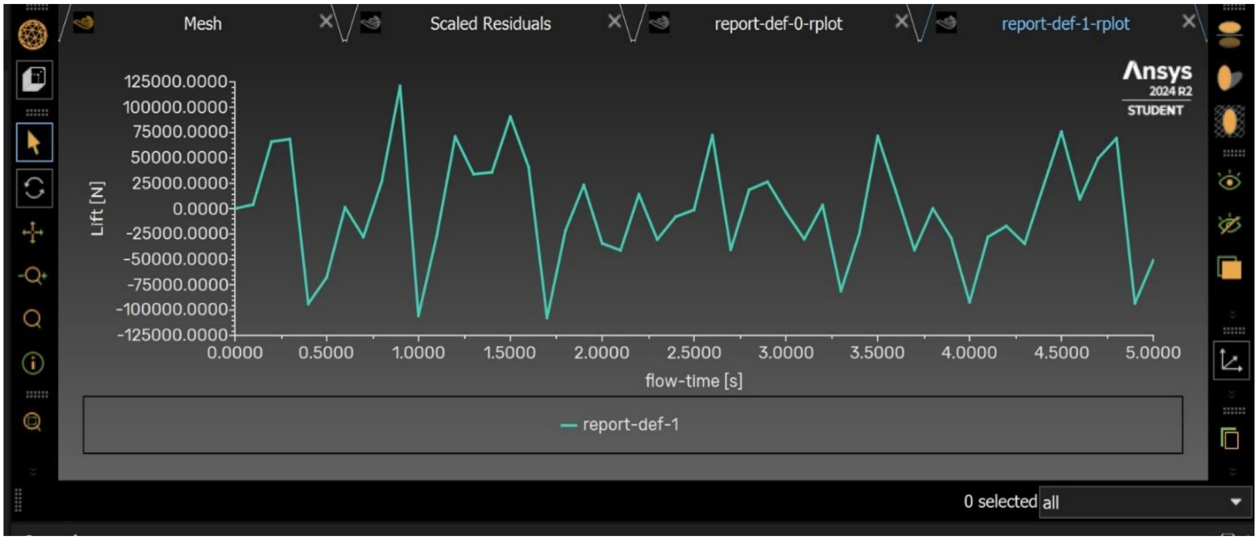


Figure 12" Lift Force Graph"

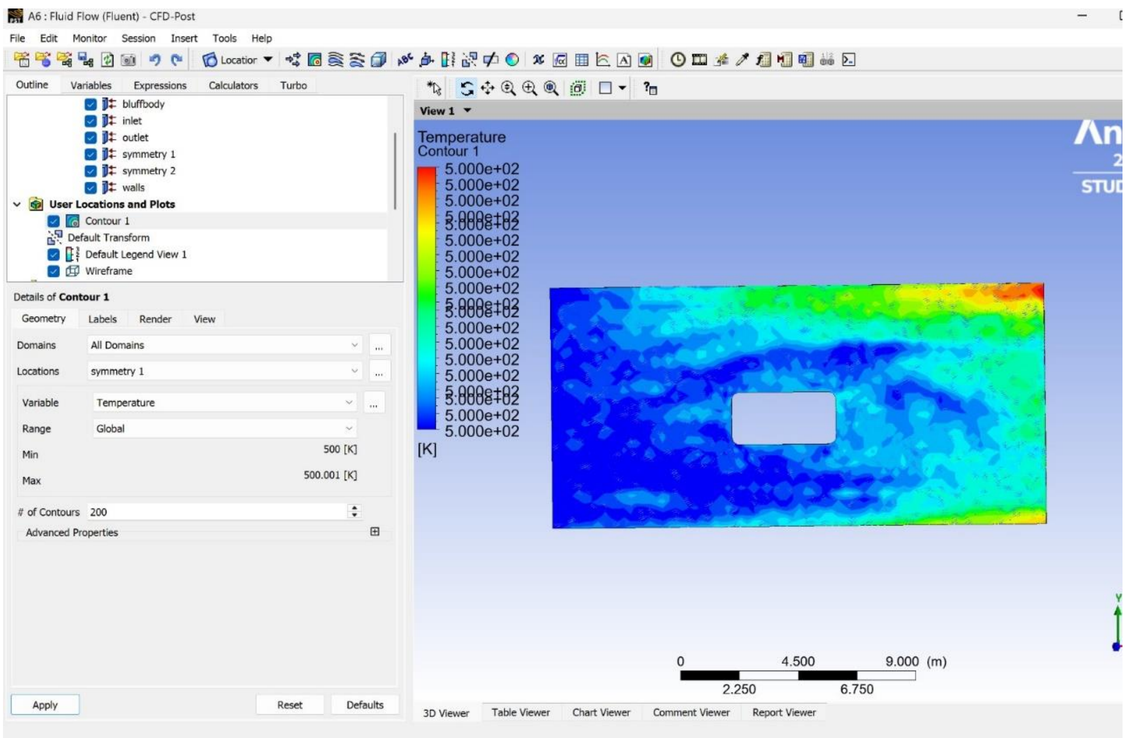


Figure 13"Temperature Contour"

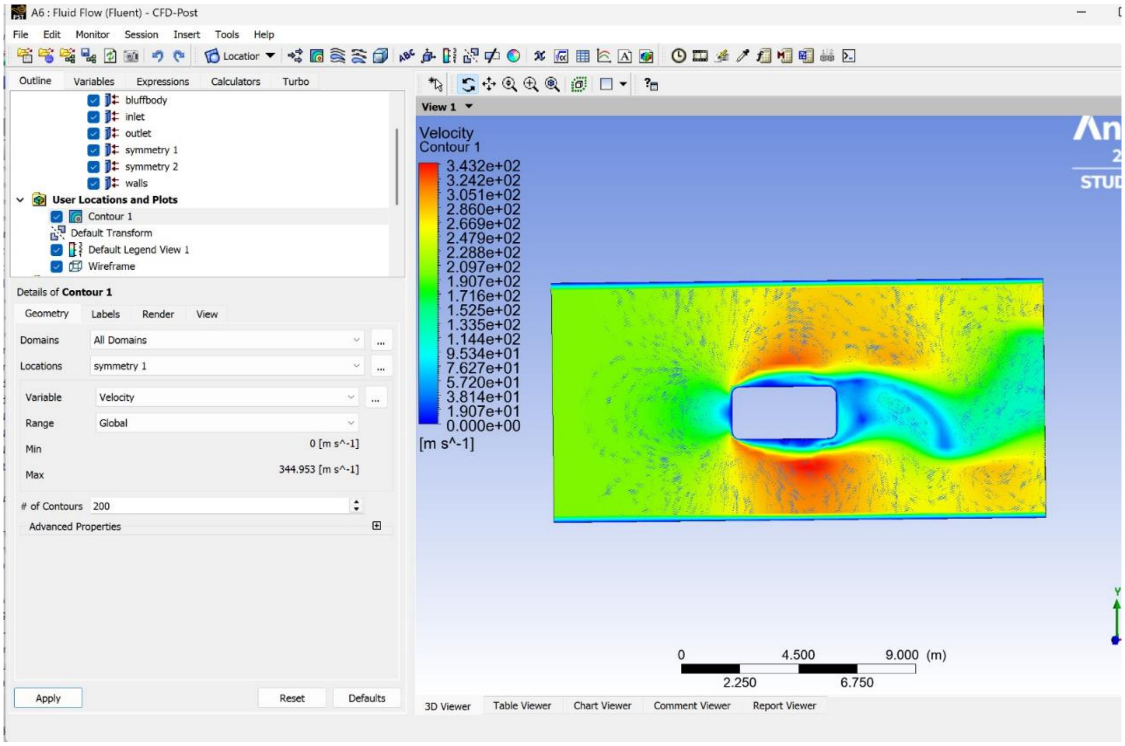


Figure 14 "Velocity Contour"

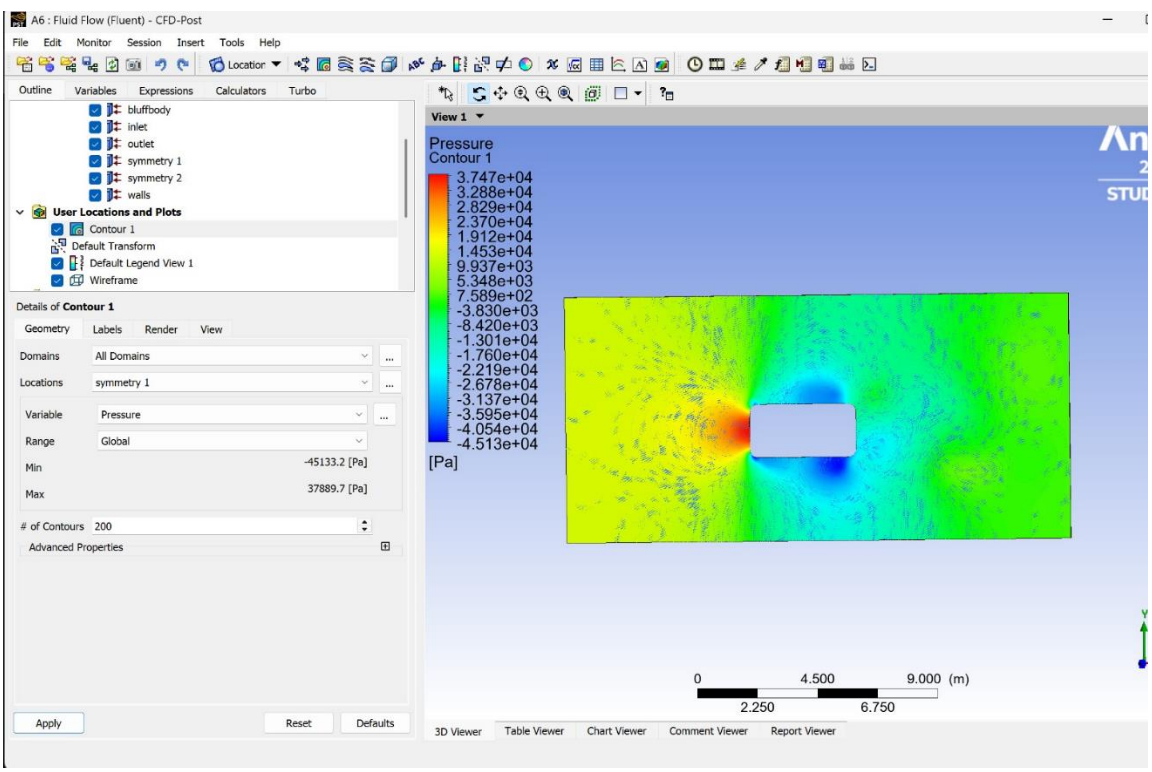


Figure 15 "Pressure Contour"

Discussion of Iteration 1:

In Iteration 1, the analysis focused on the aerodynamic behavior of the blunt boxy vehicle geometry under steady state, transient state, and heat transfer conditions. The images presented in this iteration provide valuable insights into the fluid dynamics and thermal interactions experienced by the body under study.

1. **Steady State Analysis:** In the steady state analysis, the velocity and pressure contours illustrate the aerodynamic characteristics of the body at a constant flow velocity of 200 m/s. The velocity contour reveals significant flow separation at the rear end of the body, forming a large wake region indicating an increased pressure drag. The pressure contour further demonstrates the pressure distribution across the body, with higher pressure observed at the frontal surface and a notable drop in pressure at the wake region. The lift and drag force graphs provide quantitative data, indicating that the geometry induces substantial drag due to its blunt shape and prominent wake formation.
2. **Transient State Analysis:** The transient state analysis examines the dynamic response of the body to fluid flow variations over time. The velocity contour shows the fluctuating nature of the wake region, characterized by alternating vertices shedding from the body's trailing edge. The pressure contour depicts oscillatory pressure distribution, particularly at the rear end, which can lead to periodic lift and drag fluctuations as shown in figures. This unsteady behavior can contribute to flow-induced vibrations, impacting the overall stability of the structure.
3. **Heat Transfer Analysis:** The heat transfer analysis illustrates the thermal effects on the body surface under flow conditions. The temperature contour highlights the temperature distribution, with higher thermal gradients concentrated at the frontal region where airflow stagnates. The velocity contour depicts flow behavior around the body, indicating regions of high-speed airflow and potential thermal dissipation zones. The pressure contour illustrates the pressure distribution in relation to thermal effects, further reinforcing the thermal behavior observed in the temperature contour. These visualizations provide a comprehensive overview of the interaction between aerodynamic forces and heat transfer, crucial for assessing thermal management strategies.

Iteration 2:

For this iteration, the dimensions of the body were **4 x 2 m,** and the corner radius was **0.3 m.** The velocity of fluid at inlet was now changed to a value of **500 m/s**. The angle of attack was changed by rotating the body through a **90° angle**. These conditions were analyzed for steady state, transient state as well as heat transfer in this iteration. The mesh size for this iteration was **0.25m** to ensure accuracy in the determined results from the analysis.

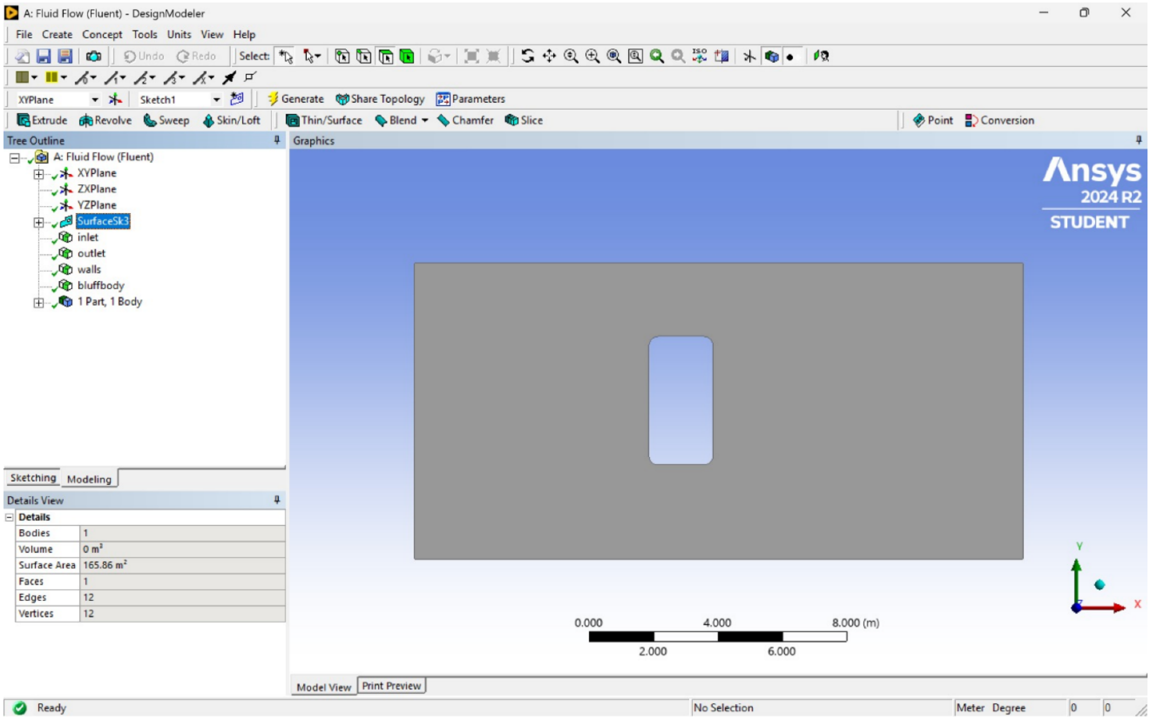


Figure 16 "Geometry Construction of Body in ANSYS"

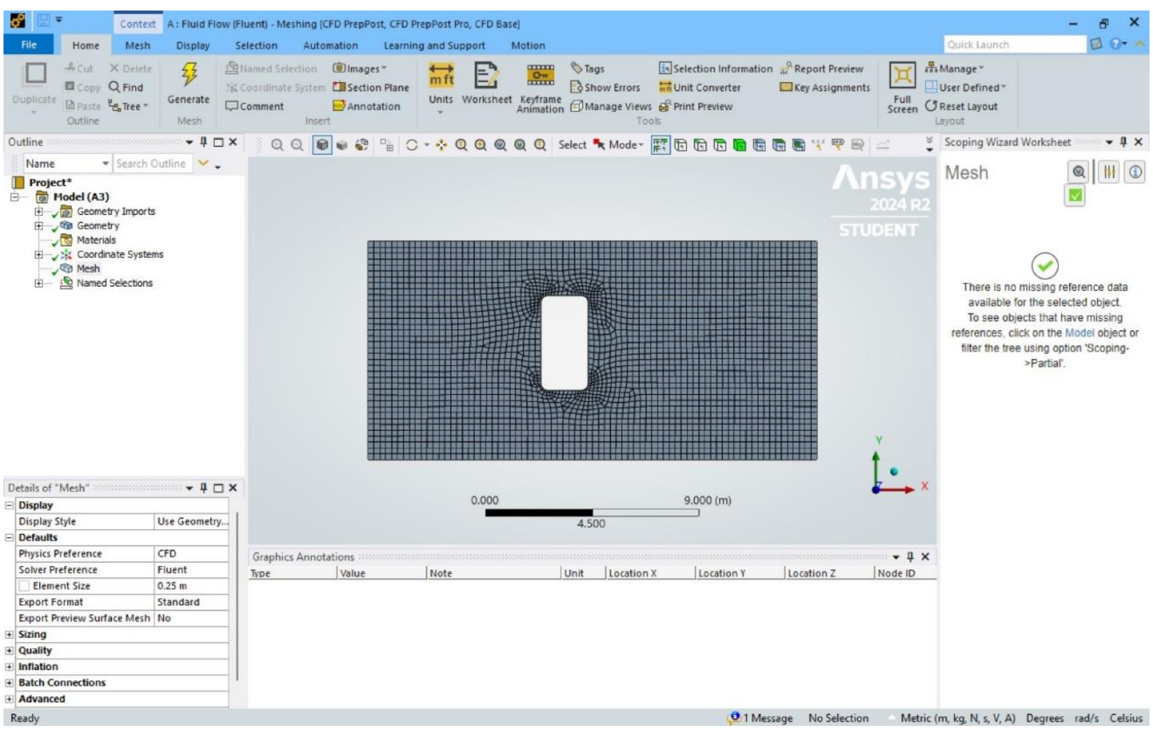


Figure 17 "Meshing on Body"

Iteration 2.1:Steady State Analysis

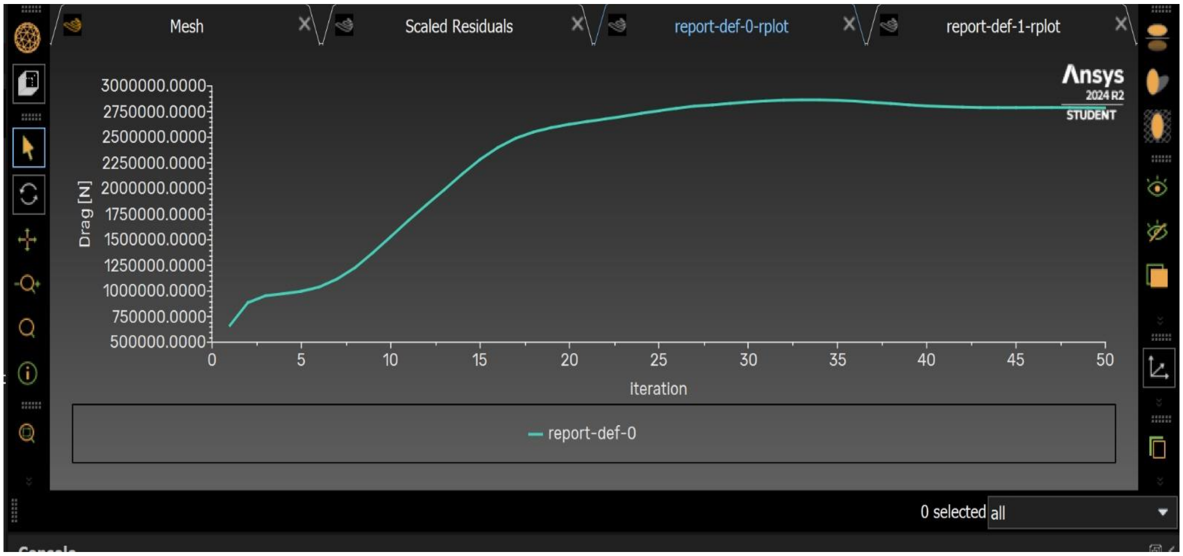


Figure 18"Drag Force Graph"

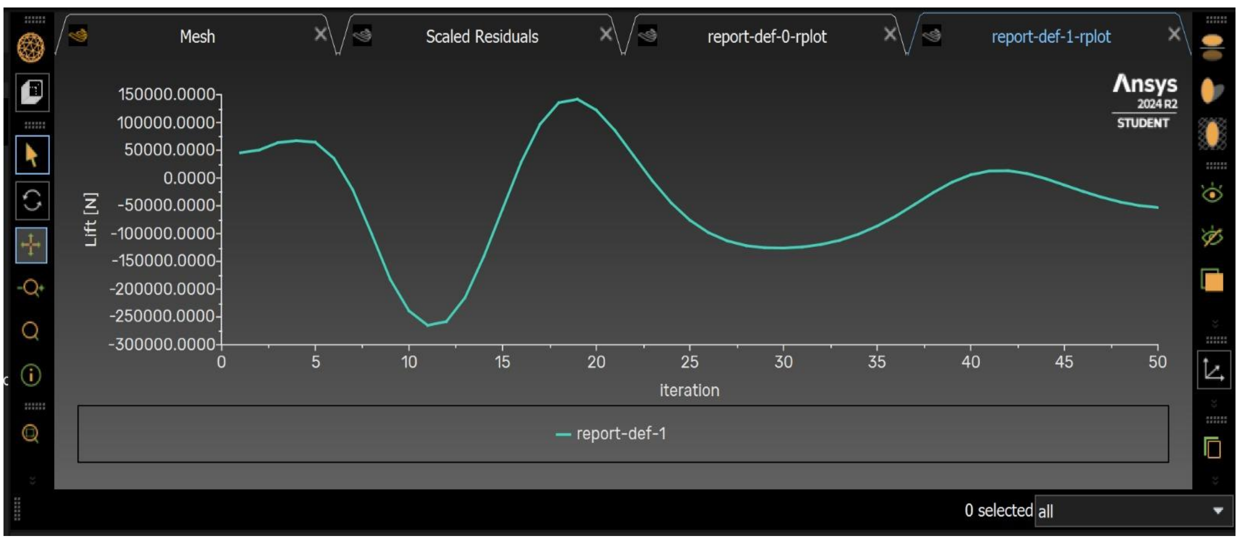


Figure 19 "Lift Force Graph"

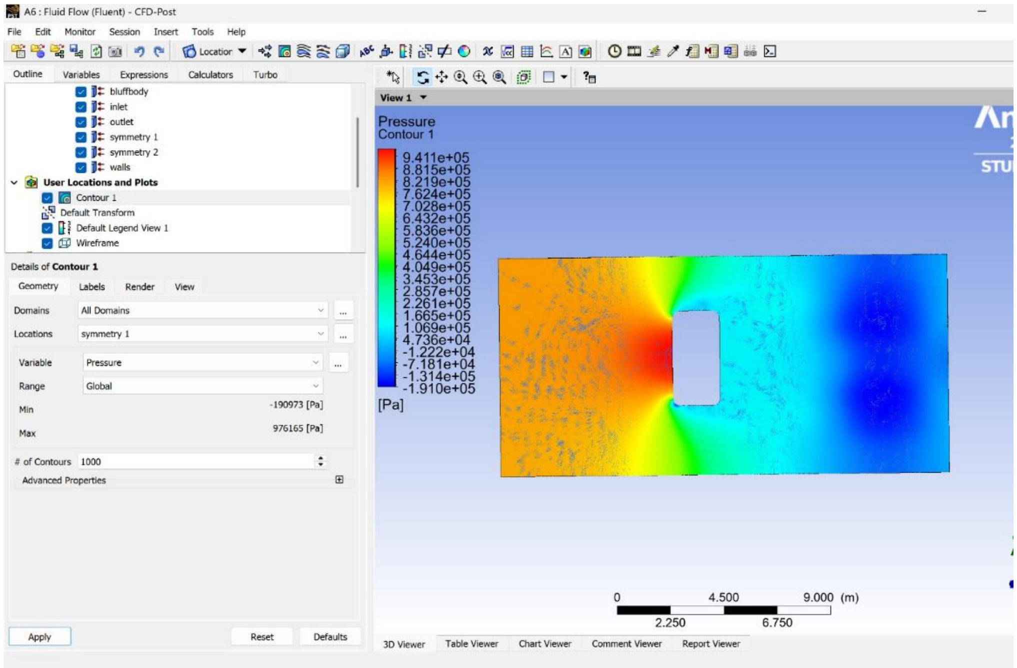


Figure 20 "Pressure Contour"

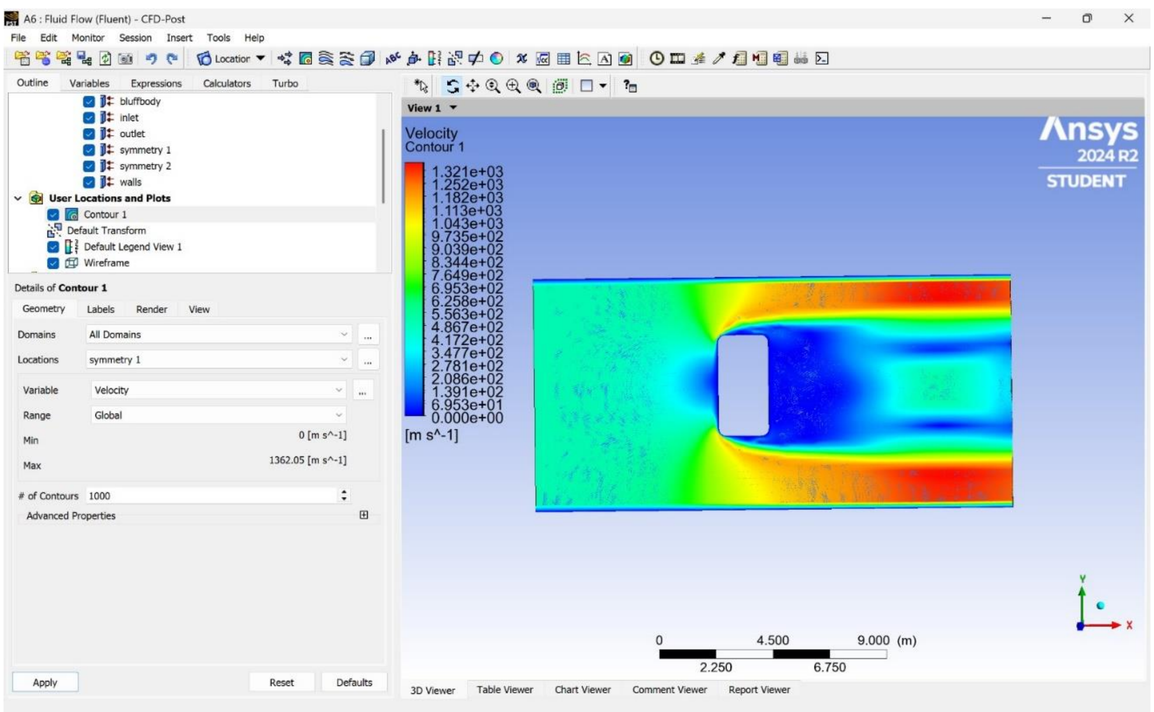


Figure 21 "Velocity Contour"

Iteration 2.2:Transient State Analysis

Figure 22 "Drag Force Graph"

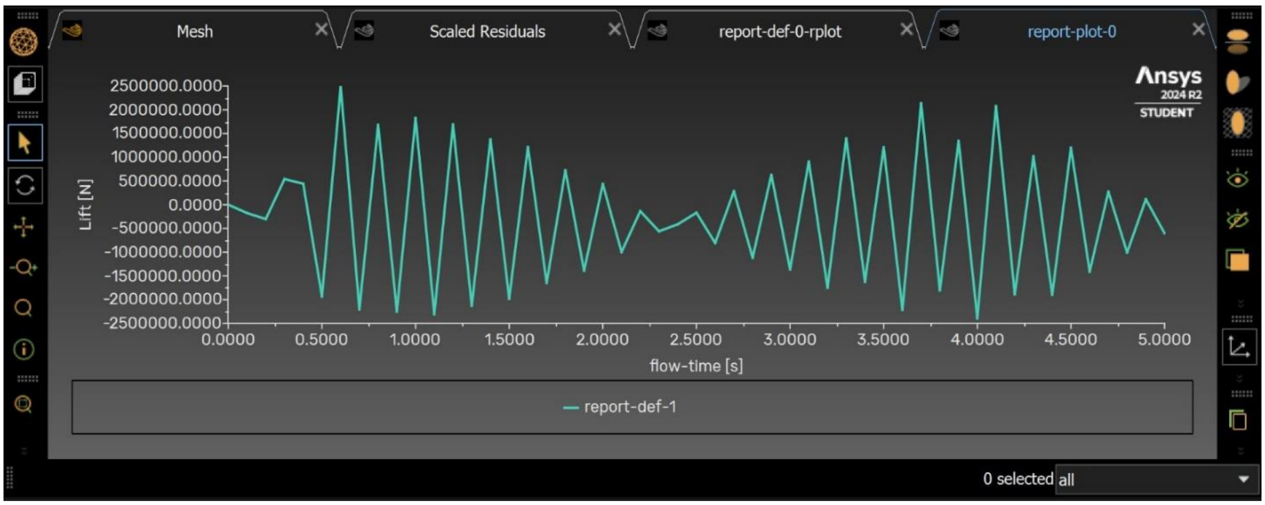


Figure 23 "Lift Force Graph"

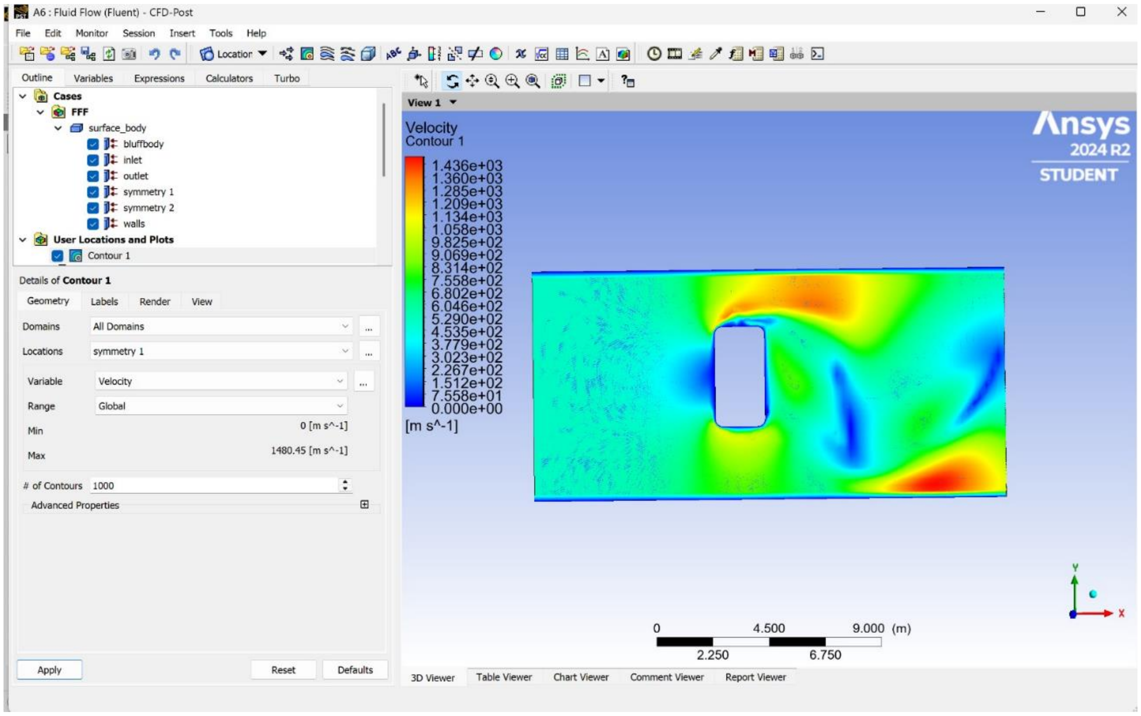


Figure 24 "Velocity Contour"

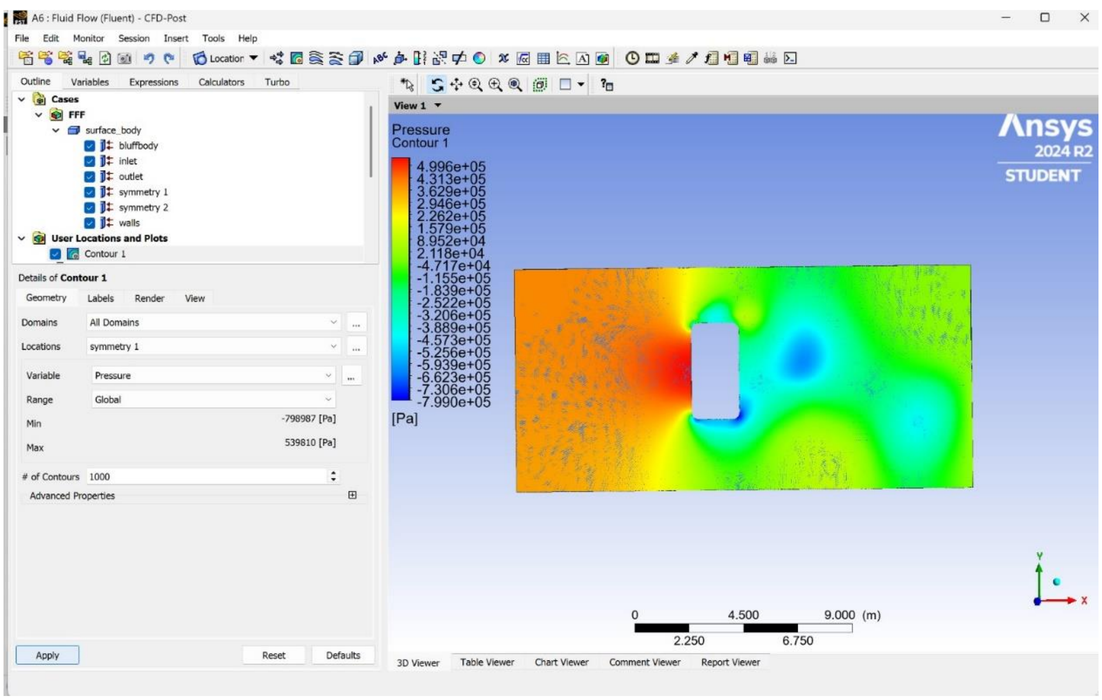


Figure 25" Pressure Contour"

Iteration 2.3:Heat Transfer Analysis

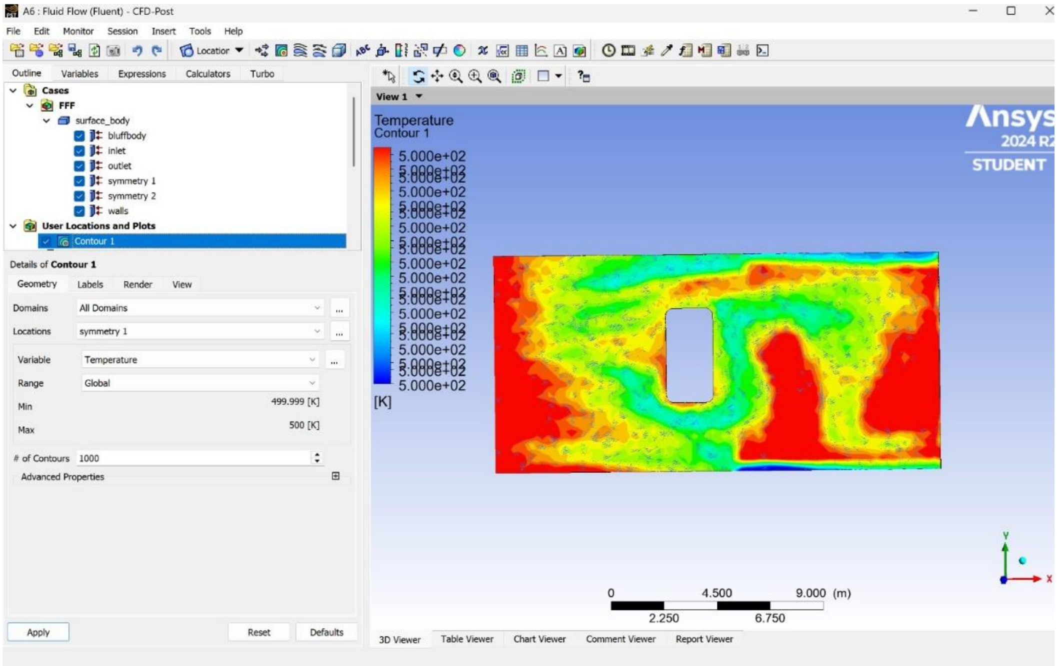


Figure 26 "Temperature Contour"

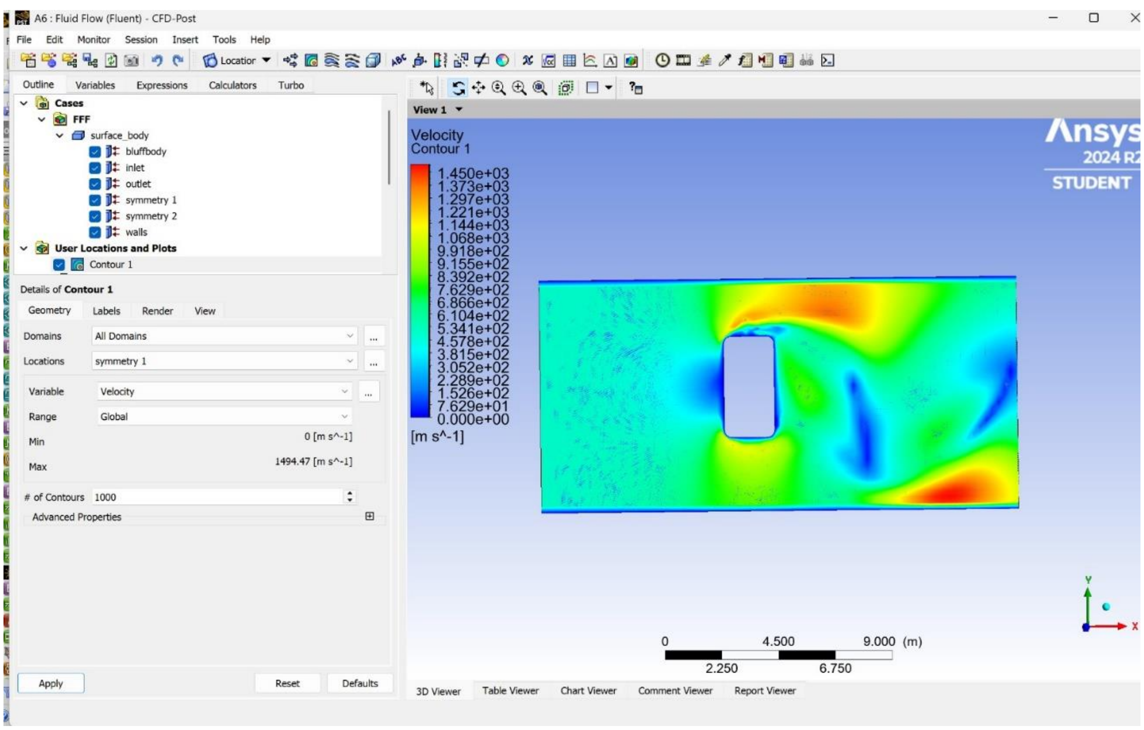


Figure 27 "Velocity Contour"

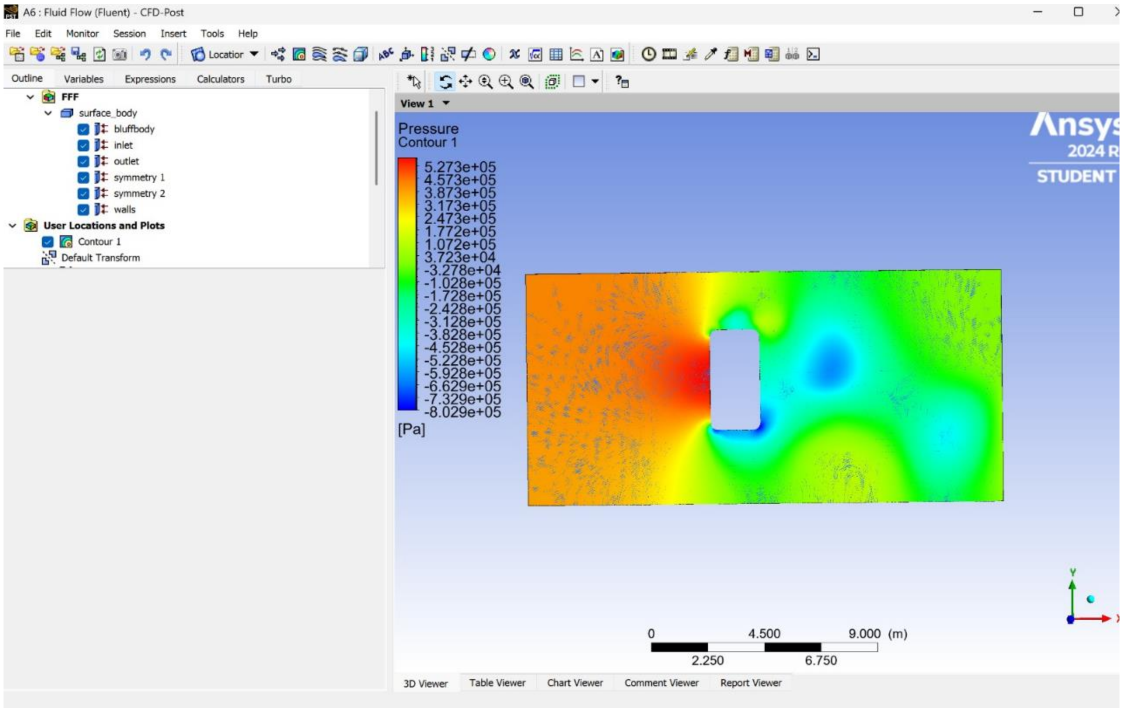


Figure 28 "Pressure Contour"

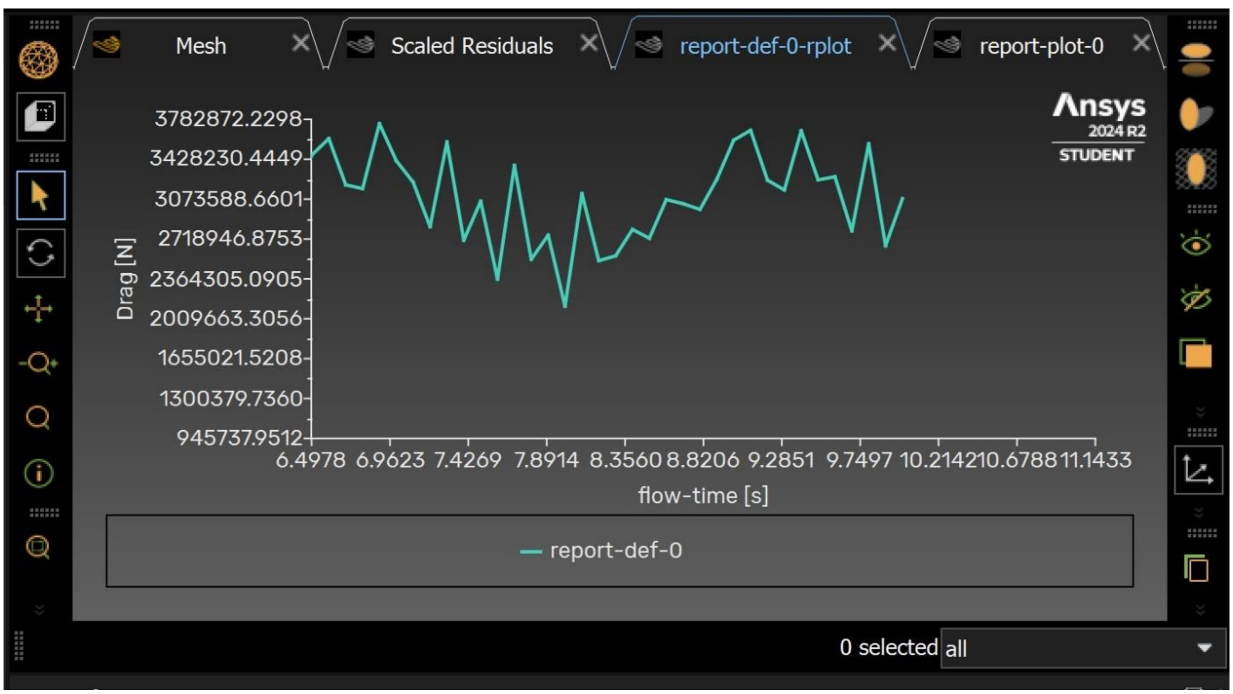


Figure 29 "Drag Force Graph"

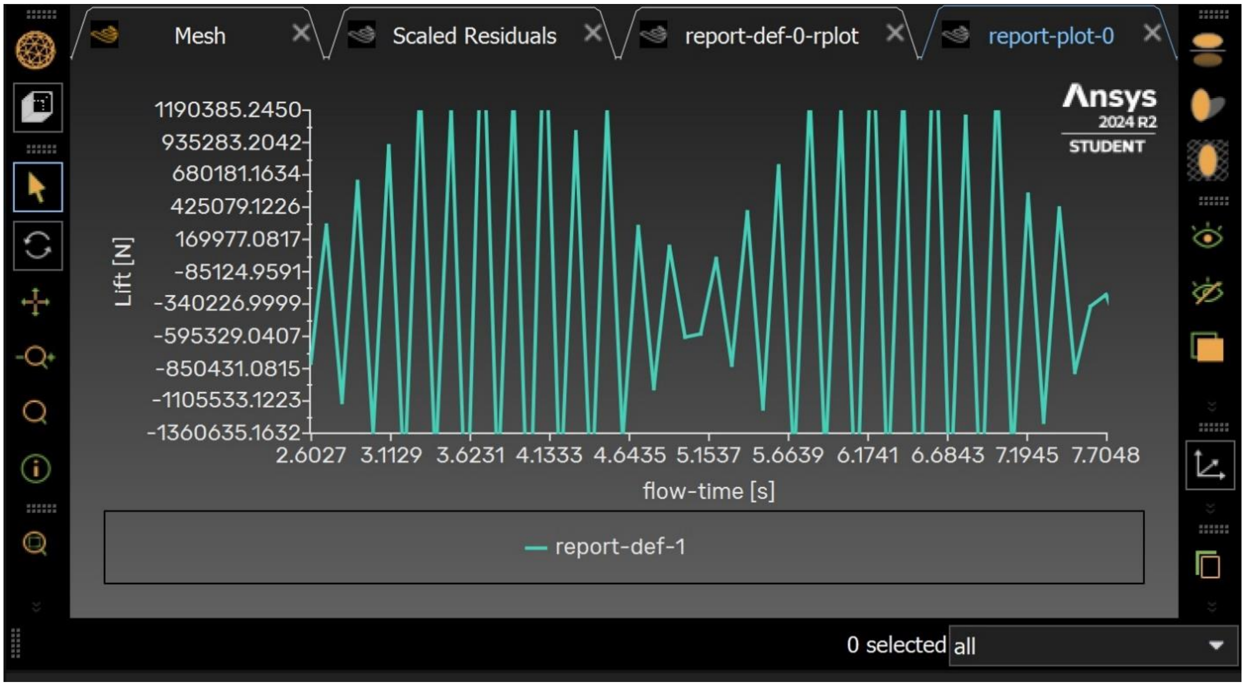


Figure 30 "Lift Force Graph"

Discussion of Iteration 2:

1. **Steady State Analysis:** In Iteration 2, the body was oriented at a 90° angle of attack with an increased inlet velocity of 500 m/s. The steady state analysis revealed higher drag and lift forces as shown in figures. The pressure contour highlighted increased pressure accumulation at the leading edge and a more pronounced wake region due to the higher flow velocity. The velocity contour indicated accelerated flow around the body, leading to greater pressure drag.

2. **Transient State Analysis:** In the transient analysis, the higher inlet velocity led to intensified vortex shedding, as observed in the velocity contour. The pressure contour showed fluctuating pressure distributions at the rear end, indicating increased flow instability. Lift and drag forces exhibited greater oscillations, reflecting the unsteady aerodynamic behavior under these conditions.

3. **Heat Transfer Analysis:** The heat transfer analysis for Iteration 2 illustrated a substantial rise in temperature gradients, particularly at the leading and trailing edges, due to the higher inlet velocity. The temperature contour showed significant thermal accumulation at the frontal region, while the velocity contour indicated increased heat dissipation along the body surface. The pressure contour further emphasized the impact of temperature variations on aerodynamic forces.

Iteration 3:

For this iteration, the dimensions of the body were **5 x 3 m,** and the corner radius was now changed to **0.7 m.** The velocity of fluid at inlet was now changed to a value of **200 m/s**. The temperature at both inlet and outlet was **500K**. The overall body temperature is **800K** .These conditions were analyzed for steady state, transient state as well as heat transfer in this iteration. The mesh size for this iteration was **0.25m** to ensure accuracy in the determined results from the analysis.

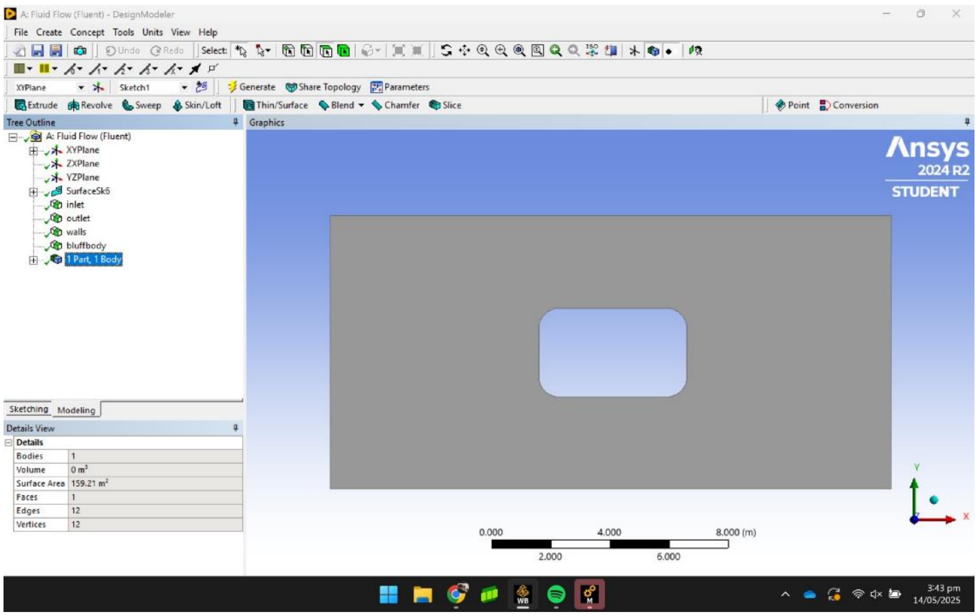


Figure 31 "Geometry Construction of Body in ANSYS"

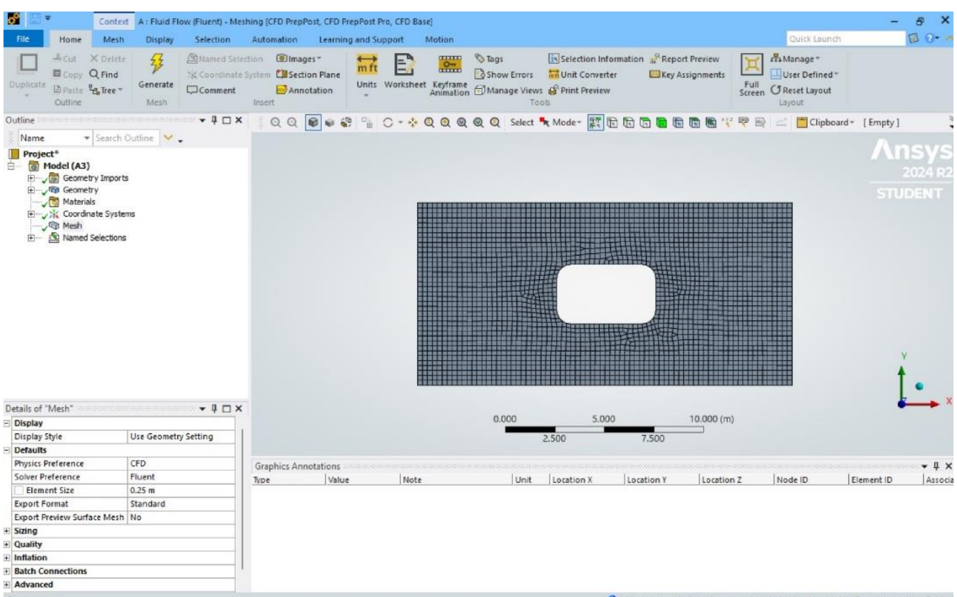


Figure 32 "Meshing on Body"

Iteration 3.1:Steady State Analysis

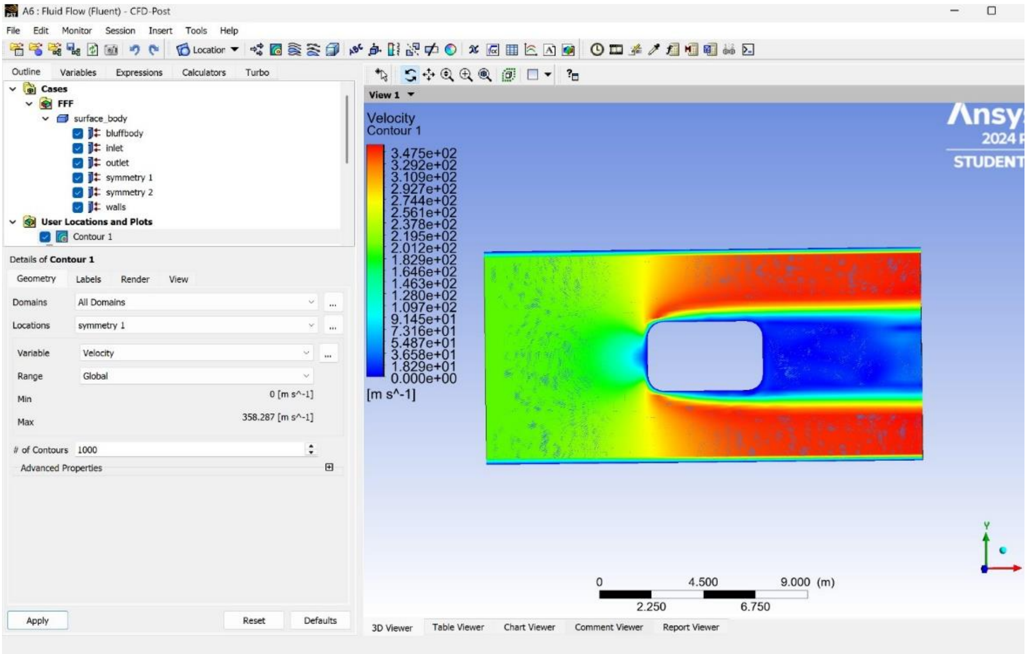


Figure 33 "Velocity Contour"

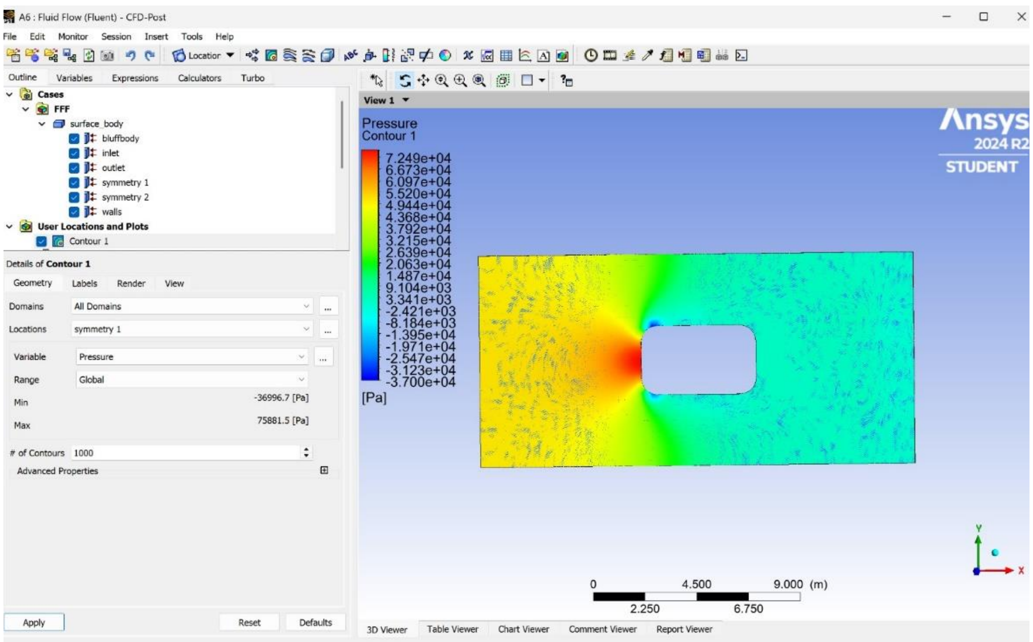


Figure 34 "Pressure Contour"

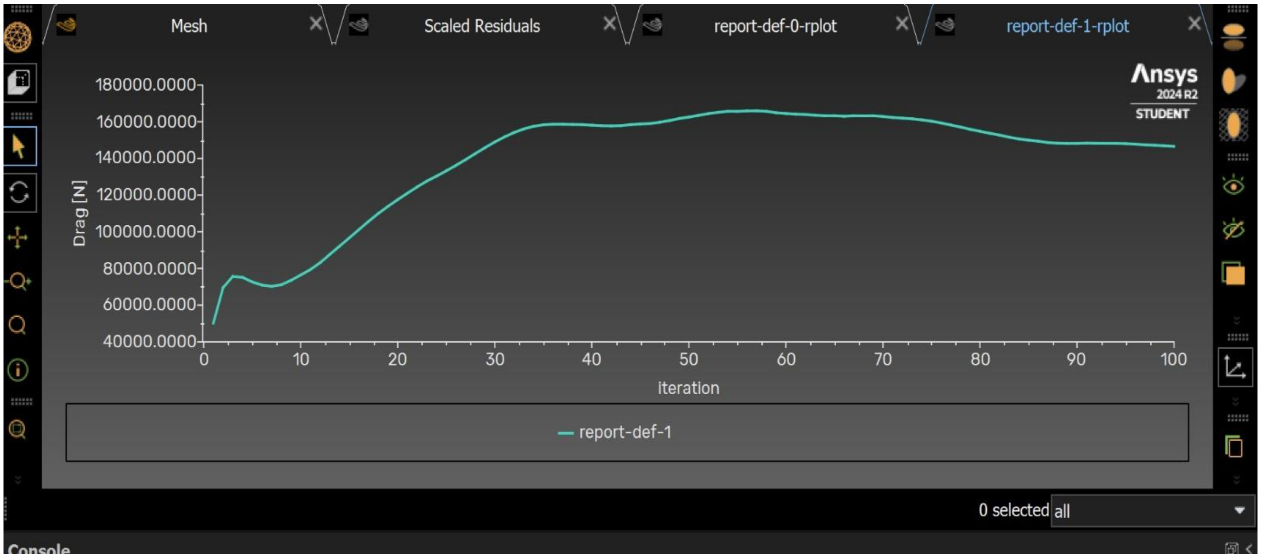


Figure 35 "Drag Force Graph"

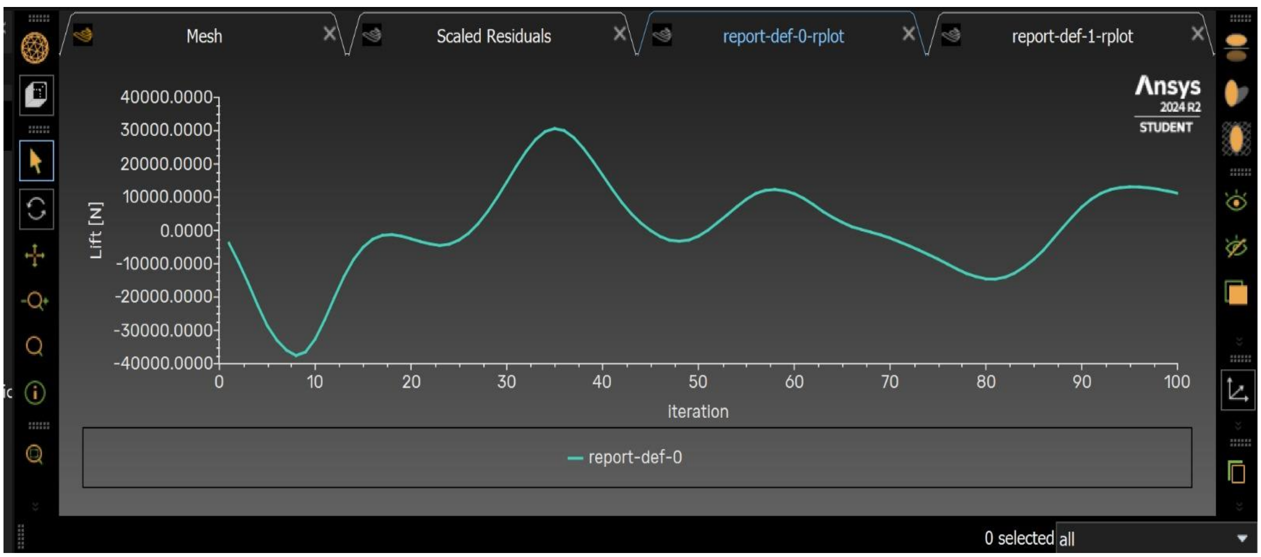


Figure 36 "Lift Force Graph"

Iteration 3.2:Transient State Analysis

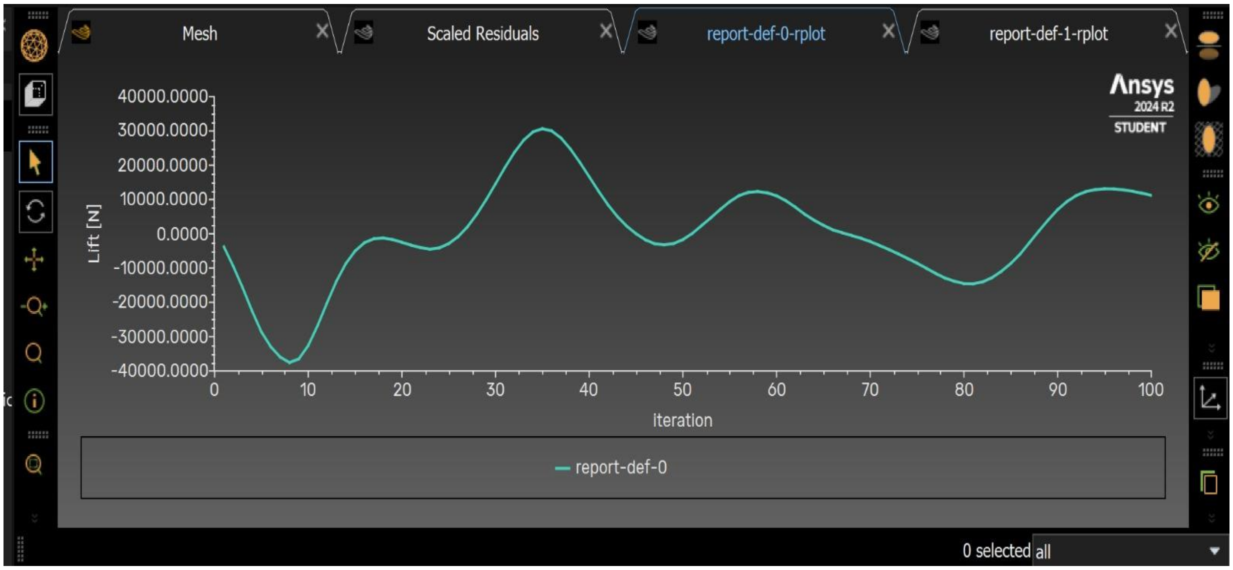


Figure 37 "Lift Force Graph"

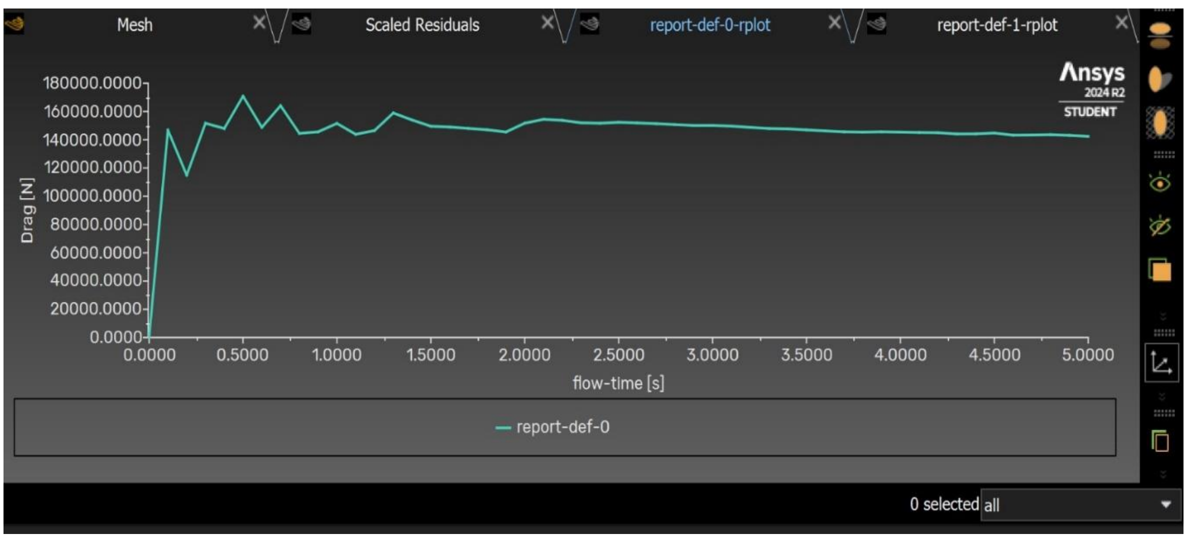


Figure 38 "Drag Force Graph"

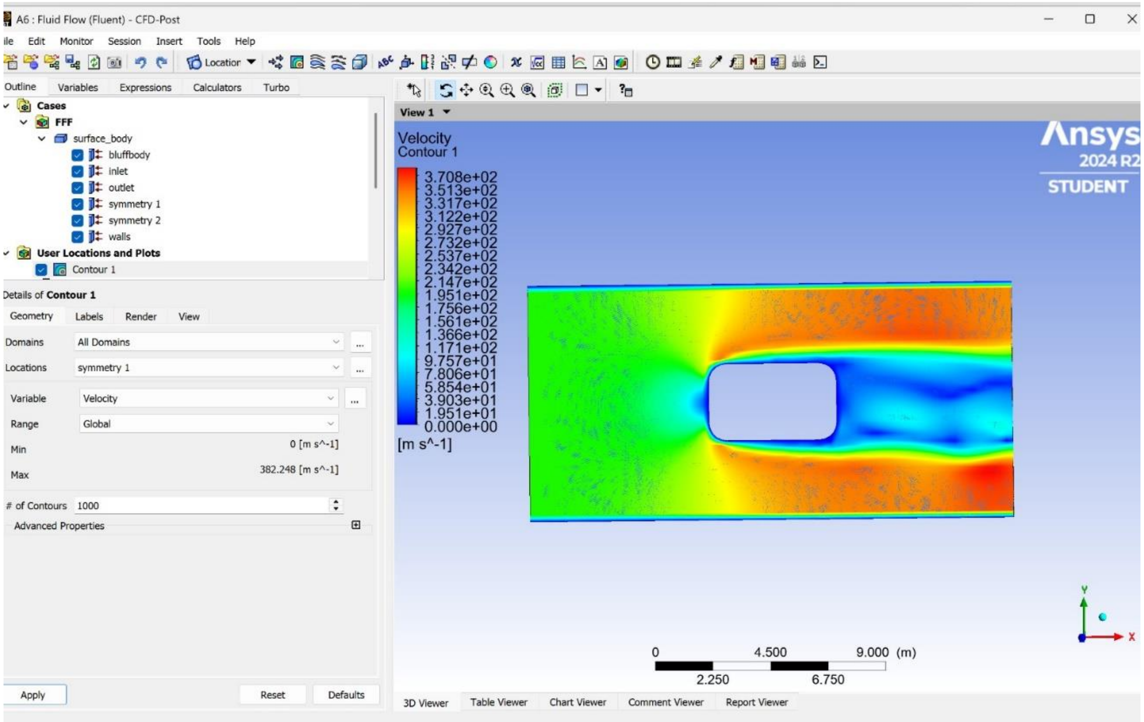


Figure 39 "Velocity Contour"

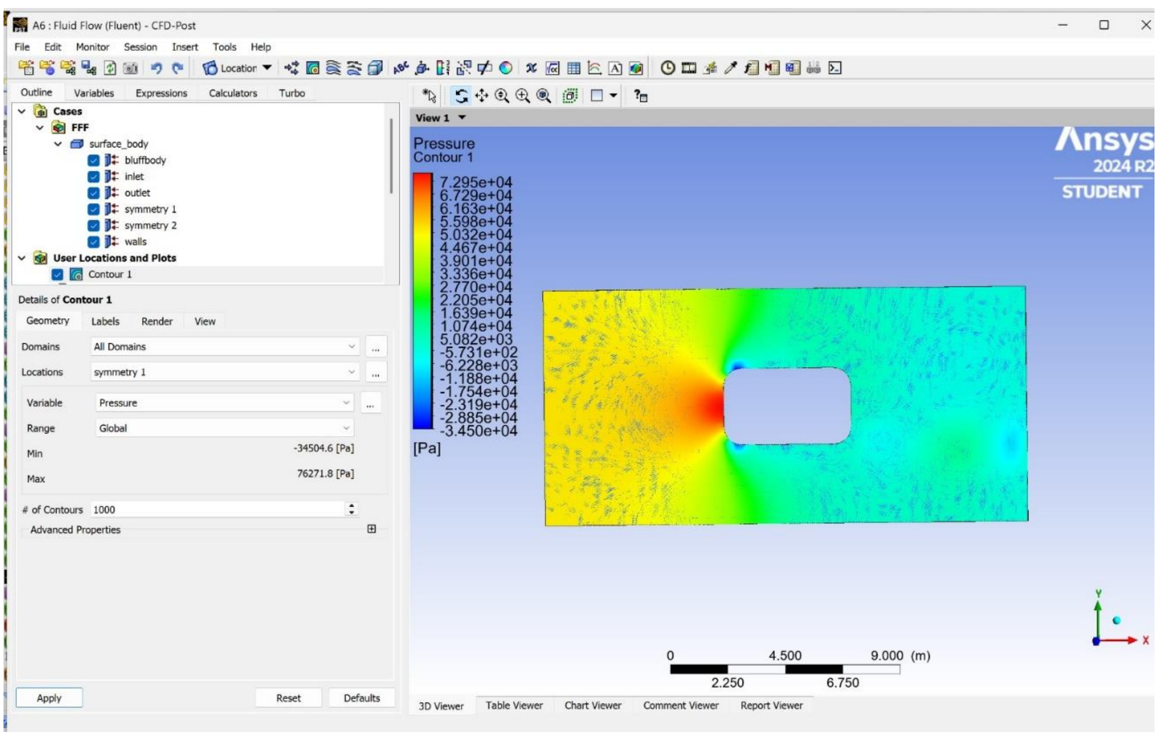


Figure 40 "Pressure Contour"

Iteration 3.3:Heat Transfer Analysis

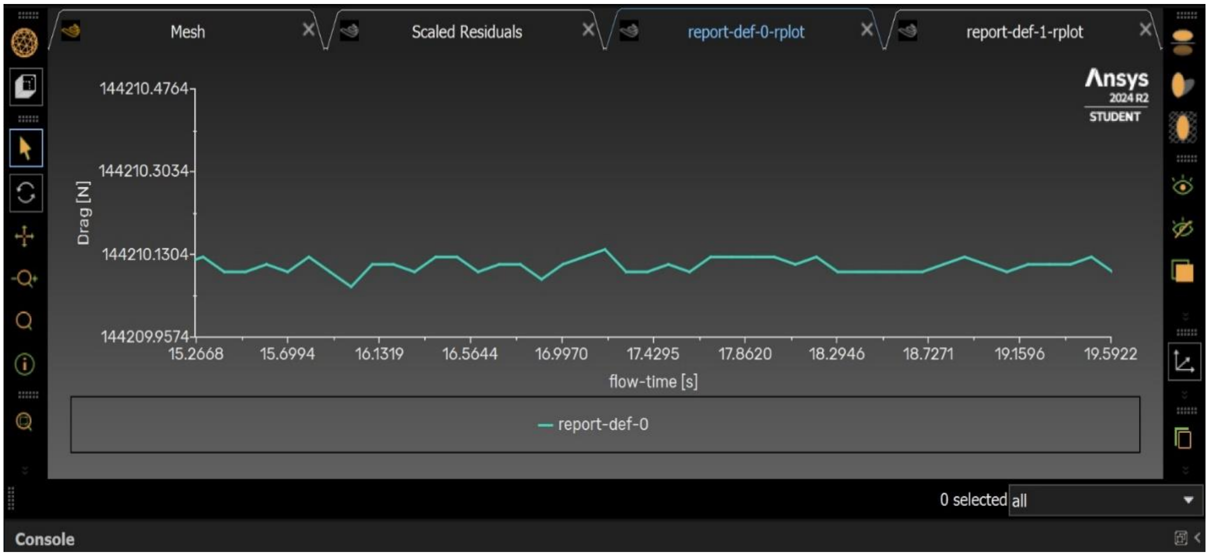


Figure 41 "Drag Force Graph"

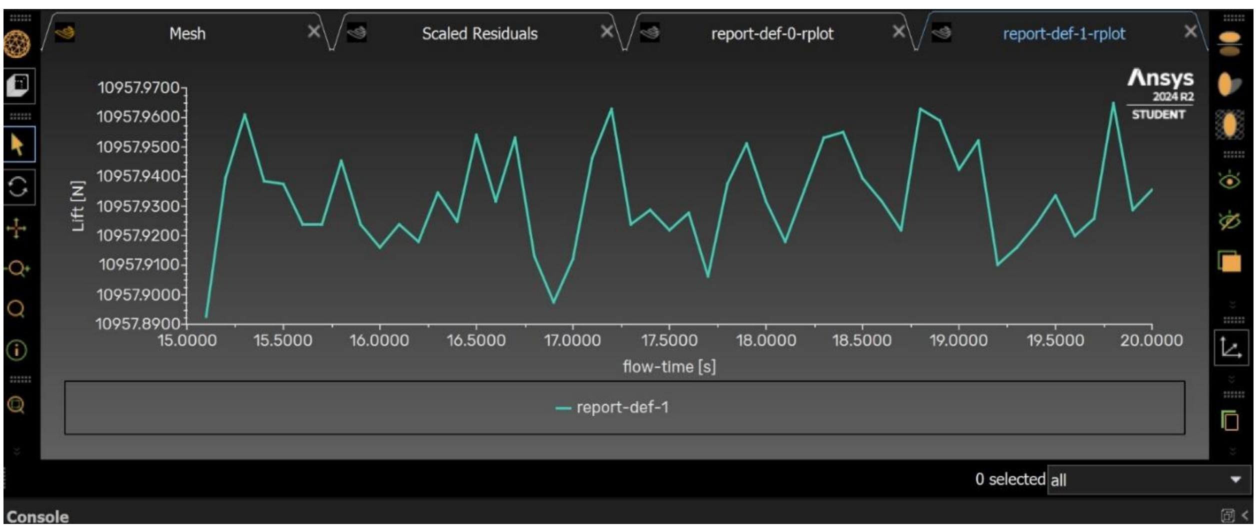


Figure 42 "Lift Force Graph"

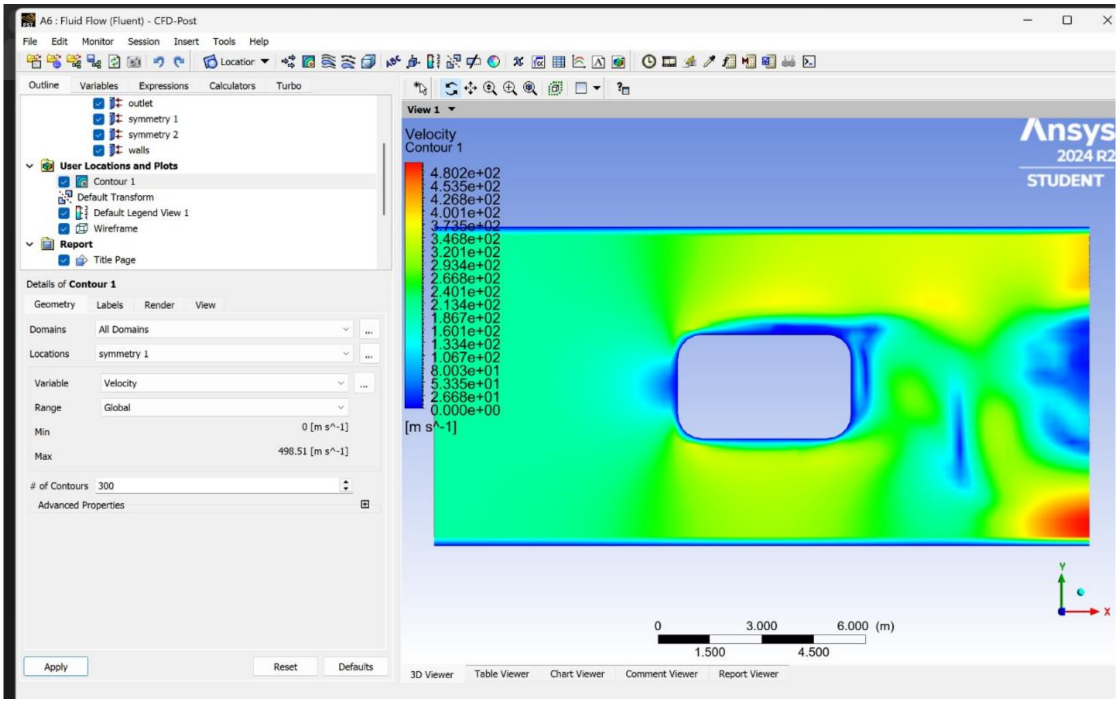


Figure 43 "Velocity Contour"

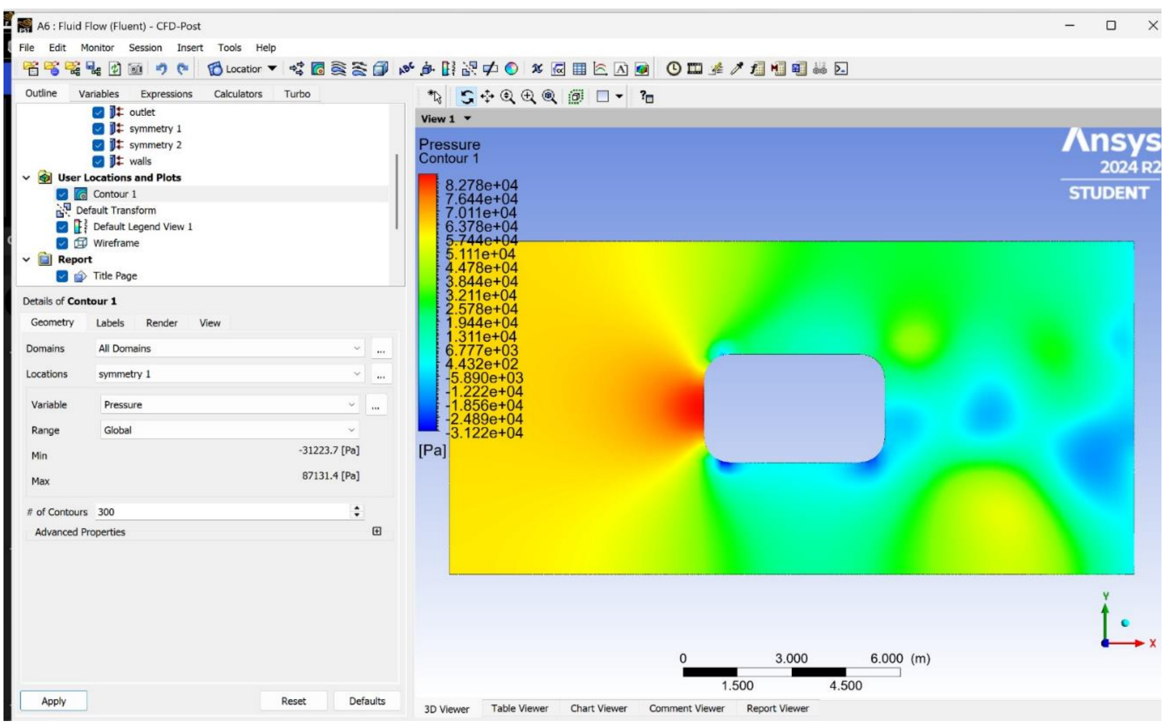


Figure 44 "Pressure Contour"

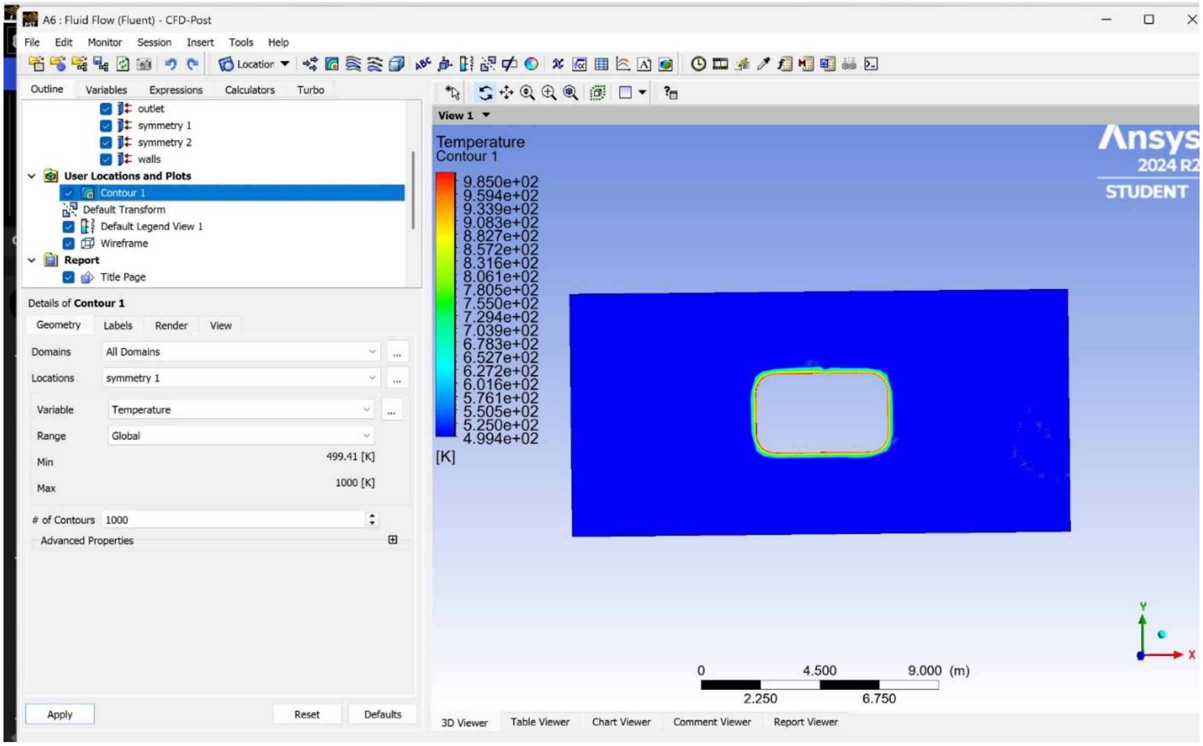


Figure 45 "Temperature Contour"

Discussion of Iteration 3:

1. **Steady State Analysis:** Iteration 3 involved a larger body size (5 x 3 m) with a corner radius of 0.7 m and a consistent inlet velocity of 200 m/s. The steady state analysis showed increased drag due to the larger frontal area. The velocity contour revealed intensified flow separation at the rear end, contributing to greater wake formation. Lift and drag forces indicated higher aerodynamic resistance under these conditions.

2. **Transient State Analysis:** The transient analysis in Iteration 3 displayed notable vortex shedding patterns at the rear end, resulting from the increased body size. The pressure contour revealed fluctuations in pressure distribution, with larger eddies forming due to the enhanced surface area. Lift and drag graphs showed more pronounced oscillations compared to previous iterations.

3. **Heat Transfer Analysis**: The heat transfer analysis for the larger body size indicated significant temperature gradients due to the increased surface area and higher body temperature of 800K. The temperature contour showed extensive thermal accumulation at the frontal region, while the velocity contour highlighted areas of rapid heat dissipation along the side surfaces. The pressure contour emphasized the thermal impact on aerodynamic forces under steady and transient conditions.

Conclusion:

The analysis across three iterations revealed how variations in body geometry, angle of attack, and flow velocity significantly impact aerodynamic and thermal characteristics. Steady state analysis consistently demonstrated increased drag with higher velocity and larger body sizes, while transient analysis highlighted intensified vortex shedding patterns under fluctuating flow conditions. The heat transfer analysis emphasized thermal accumulation at the frontal region, particularly under elevated temperatures and higher velocities. Comparing the three cases, Iteration 1 served as a baseline with moderate velocity and standard geometry, resulting in substantial wake formation. Iteration 2, with increased velocity and altered orientation, exhibited intensified aerodynamic forces and higher thermal gradients. Iteration 3, with increased body size and surface temperature, displayed the highest drag and thermal accumulation, emphasizing the compound effects of size and heat on aerodynamic performance. These findings underscore the importance of optimizing geometric configurations and flow parameters to achieve aerodynamic efficiency while mitigating thermal impacts.