Toronto Metropolitan University

# AER626 Applied Finite Elements

# A Detailed Structural Analysis of an Aircraft Component

Final Report

**Authors:** Aman Gilani

# Table of Contents

# List of Figures

## List of Tables

## Introduction and Summary

The objective of this final project is to analyze the given design – a portion of an aircraft structural component. The task specifically pertains to a static structural and a modal analysis of the given structural component in Ansys workbench. The reason for using Ansys workbench instead of Ansys APDL was because of its enhanced graphic user interface. The structure is made of multiple segments riveted together. The angle plates and the base plate are modelled as fully fixed together/glued. Figure 1 illustrates a rough design of the aircraft structural component. The base plate is made of stainless-steel alloy with five triangles pockets at one end. The two stringers are made of different aluminum alloys to form an inverted T-shaped stringer. As seen in the figure the design has an axis of symmetry. One side of the system (side ab) is considered as fully fixed, and the opposite side (side cd) is subjected to a parabolic distributed in-plane tensile load and is held using rollers. The rollers let the system to move horizontally (within the x-y plane and only the z-axis displacements are restricted). A maximum load of 20,000 lb./in is applied on side cd at y = 2.5 in. The parabolic load distribution along face cd follows the following function:

$$P_y = \frac{P_{max}(5y - y^2)}{2.5^2}$$

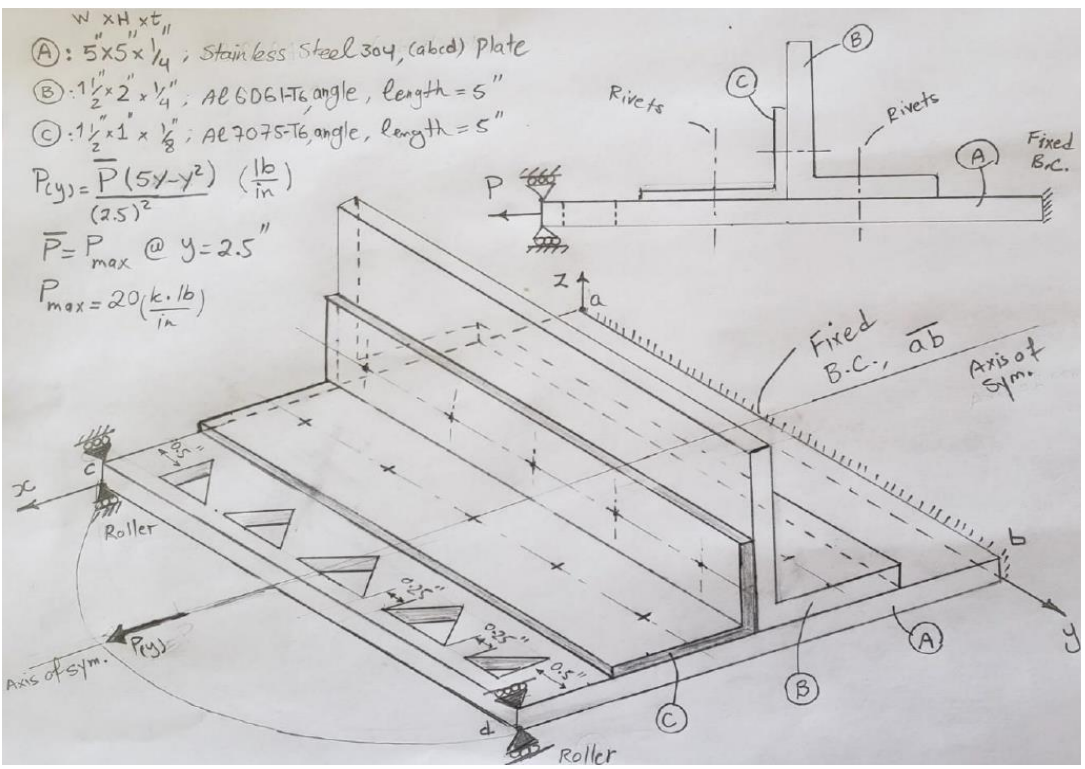


*Figure 1: Rough Sketch of the Aircraft Structural Component.*

A CAD model for the above structure is developed in CATIA V5 prior to conducting a stress and a modal analysis. The model is then imported onto Ansys workbench for its analysis. As per the project guidelines we are required to conduct the analysis for two structures, one with the original boundary conditions and second with a redesigned system by switching the boundary conditions to the perpendicular sides (fully fixed ac and rollers at bd edges), and apply the load along the bd edge, and parallel to y-axis. For each structure, we are required to complete the analysis with two element types. For this project linear solid and quadratic solid elements are used. A convergence test is conducted to show how the mesh refinement affects the deformations and stress graphs for each structure. In theory, it is expected that as the mesh size decreases the stress and deformation results will approach their exact values.

## Procedure

This section will discuss the procedure for the static and modal analysis of the component done in Ansys Workbench. Before doing the analysis, the given component was modelled in CATIA V5 and then imported onto Ansys workbench.

1. A Catia CAD model is designed for each of the three components – a base plate (Part A), a right angle (Part B), and a left angle (Part C). The CAD was created as per the given dimensions in figure 1.
2. After creating a CAD model for each part, they are assembled and constrained such that the overall structure behaves as fully glued.
3. The assembled structure is then imported to Ansys workbench for static structural and modal analysis.
4. Before starting the static analysis in Ansys, the materials provided in figure 1 and their material properties were created in workbench for the three components.
5. The respective materials were then assigned for each component.
6. Next an element type is chosen for the given structure. As mentioned above, the analysis was done for two different element types, a linear solid and a quadratic solid.
7. A body sizing is added for the structure. In order to conduct a convergence test, the analysis was conducted for three different mesh sizes. The first mesh size was 0.21, then 0.15 and lastly 0.1.
8. After assigning the element type and mesh size the system was meshed. The system is meshed for different possible combinations of element type and mesh size. All the combinations are presented in the results section.
9. Displacement and load constrain are added for the structure. Note that there are two structures with different boundary conditions analyzed for this project. For structure 1, the face ab is fully fixed, and vortexes of face cd have roller displacements. The loading is applied along face cd. For structure 2, the face ac is fully fixed, and vortexes of face bd have roller displacements. The loading is applied along face bd.
10. Now the system is solved for total deformation and equivalent von-mises stress. The system is solved for different mesh size and element type for structure 1 and structure 2. This concludes the static structural analysis.
11. For the modal analysis of the system, similar CAD model and similar materials are used.
12. Similar to the static analysis, the modal analysis is conducted for different combinations of element types and mesh sizes.
13. Only the displacements were added for each of the two structures to complete the modal analysis as the forces/pressures have no effect on the free vibration result for the two structures.
14. A maximum number of three free vibrations are compared for the modal analysis.
15. The system is solved for all the three free vibrations and the results are recorded.

# Results and Discussions

The results section is divided into 2 sections and 4 subsections. A static analysis and a modal analysis of each structure are presented in this section. Figure 2 illustrates the first structure with given boundary conditions. Figure 36 illustrates the second structure with the given boundary conditions. As discussed above, a convergence test is completed by comparing 3 mesh sizes for both the structures. Figures 3, 4 and 5 illustrates the mesh sizes 0.21, 0.15, and 0.1 respectively. Note that the mesh size is reduced for each iteration as a requirement for a convergence test. Tables 1 and 3 tabulates the maximum equivalent von-mises stress for each mesh size and element type for structure 1 and 2 respectively. Tables 2 and 4 tabulates the first three natural frequencies for structure 1 and 2 respectively.

## Structure 1

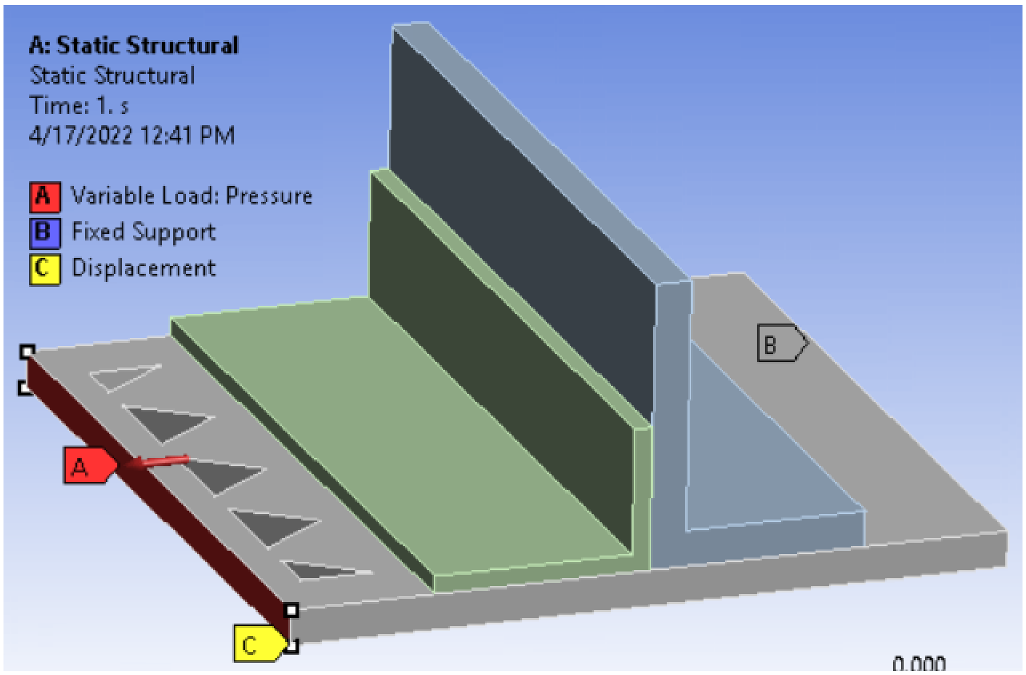


*Figure 2: First Structure.*

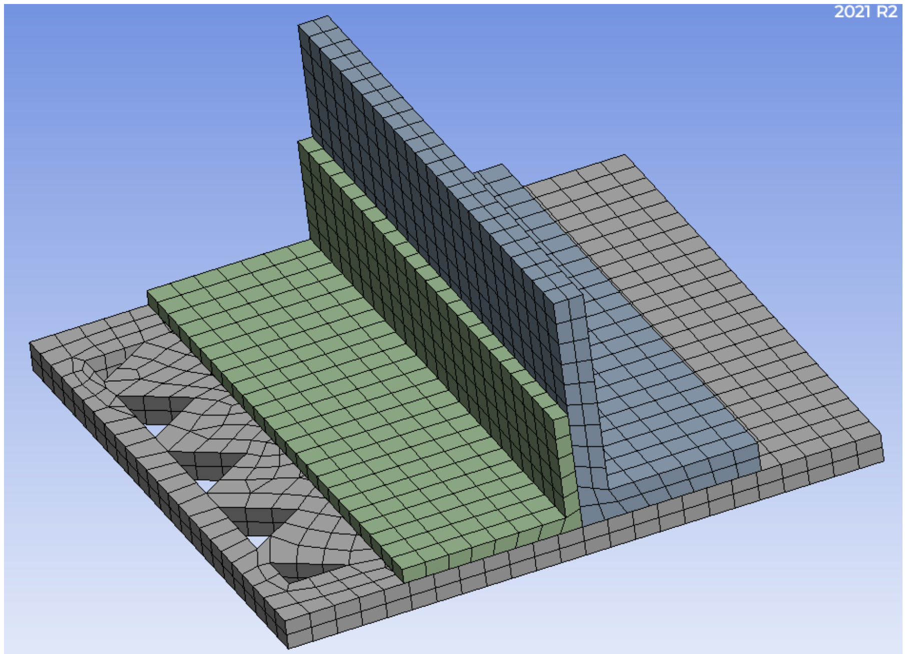


*Figure 3: Structure 1 - Mesh Size 0.21.*

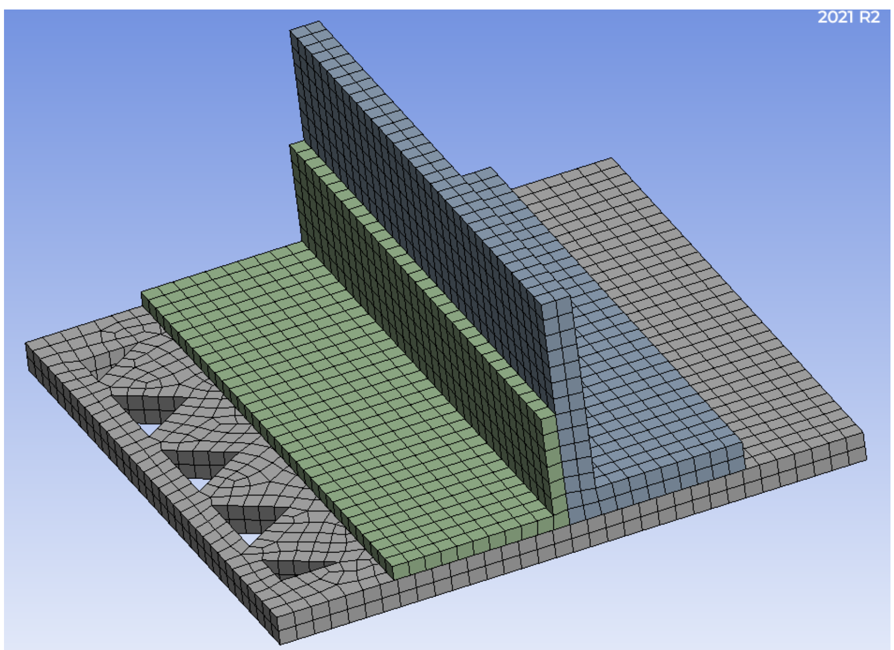


*Figure 4: Structure 1 - Mesh Size 0.15.*

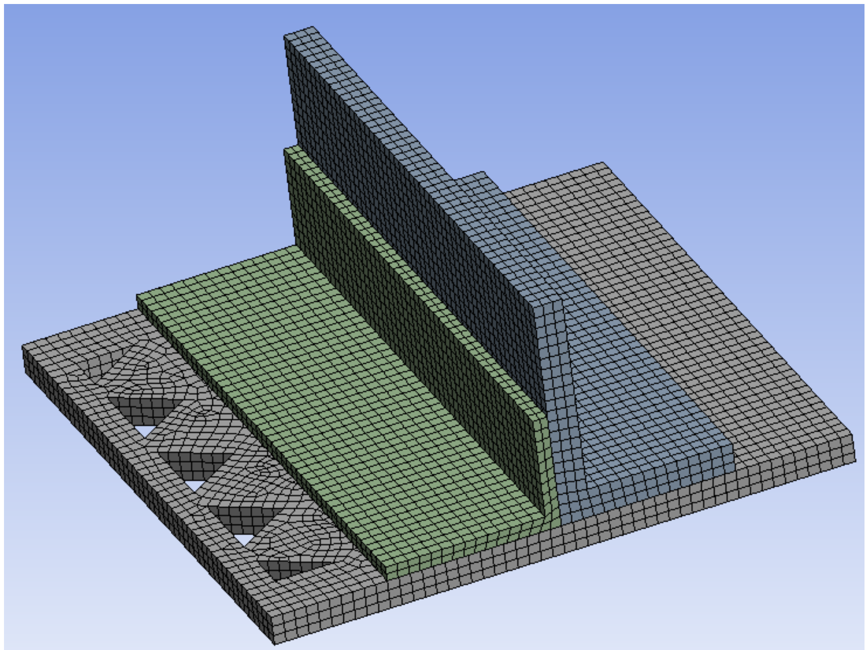

*Figure 5: Structure 1 - Mesh Size 0.1.*

## Static analysis

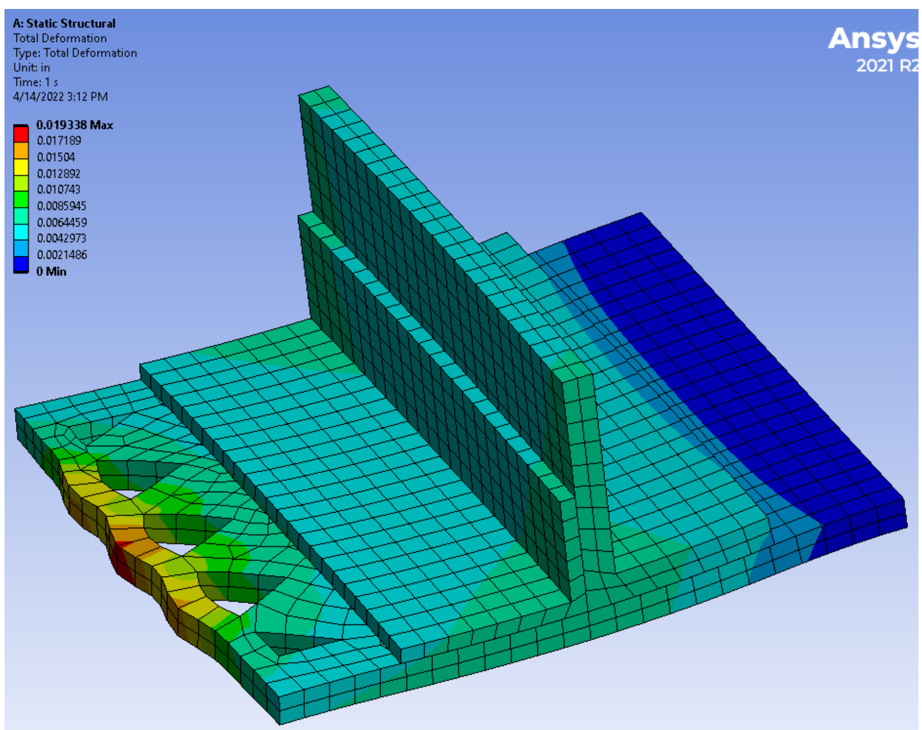


*Figure 6: Total Deformation - Mesh Size 0.21 and Linear Element Type.*

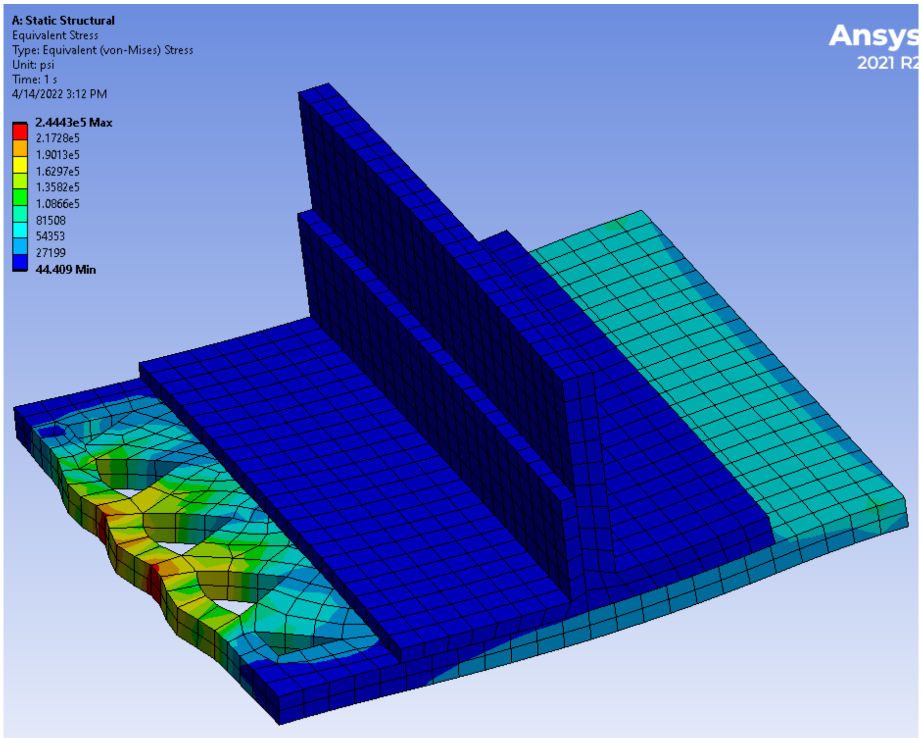


*Figure 7: Equivalent Von-Mises Stress - Mesh Size 0.21 and Linear Element Type.*

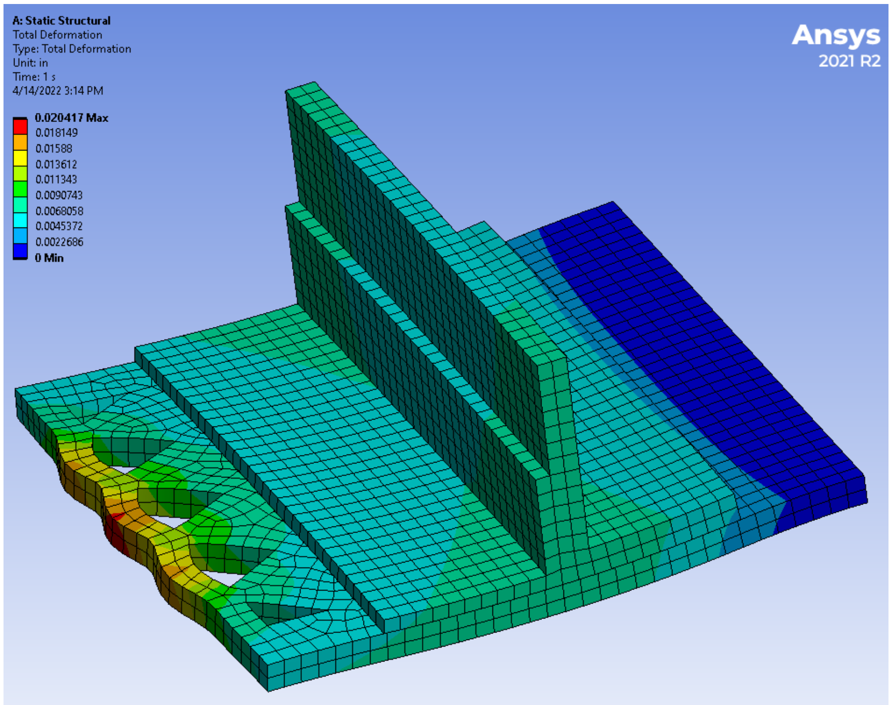


*Figure 8: Total Deformation - Mesh Size 0.15 and Linear Element Type.*

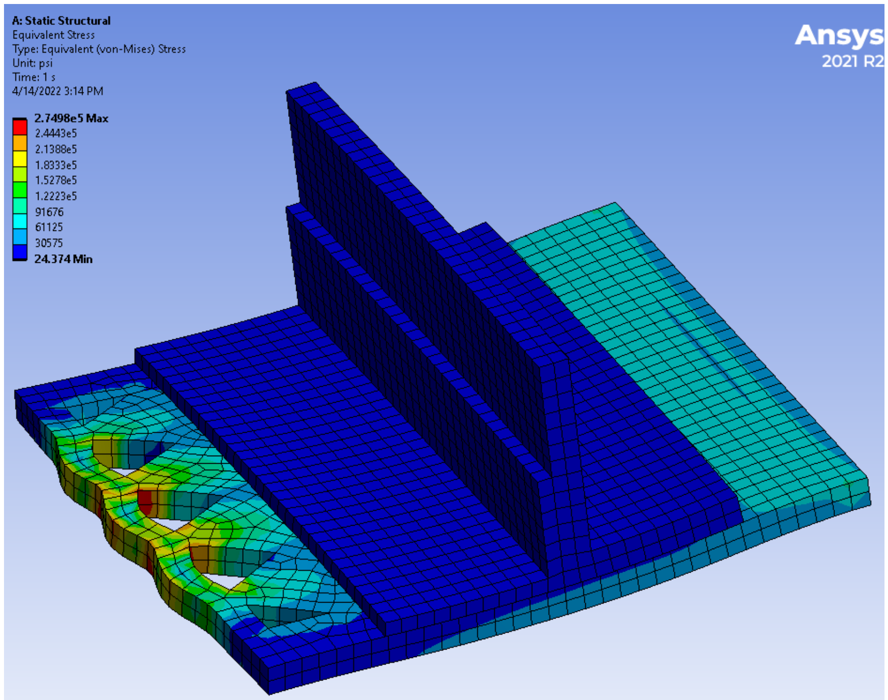


*Figure 9: Equivalent Von-Mises Stress - Mesh Size 0.15 and Linear Element Type.*

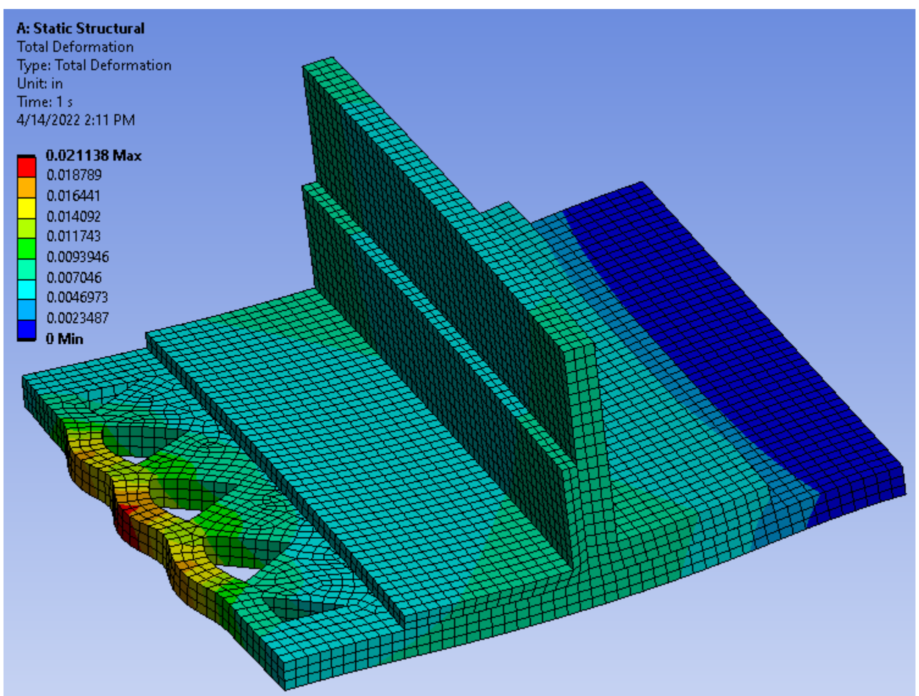


*Figure 10: Total Deformation - Mesh Size 0.1 and Linear Element Type.*

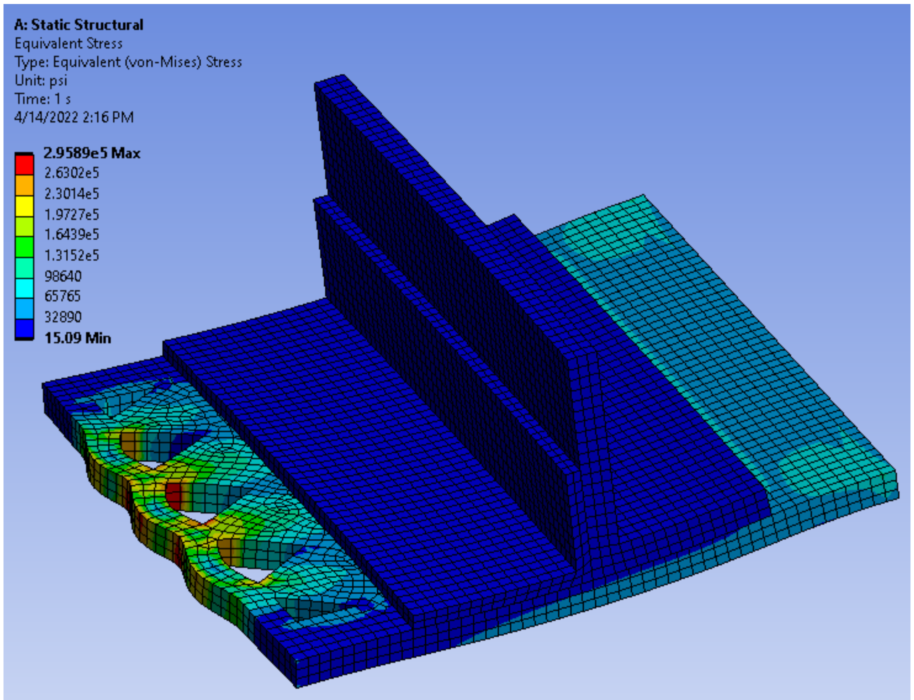


*Figure 11: Equivalent Von-Mises Stress - Mesh Size 0.1 and Linear Element Type.*

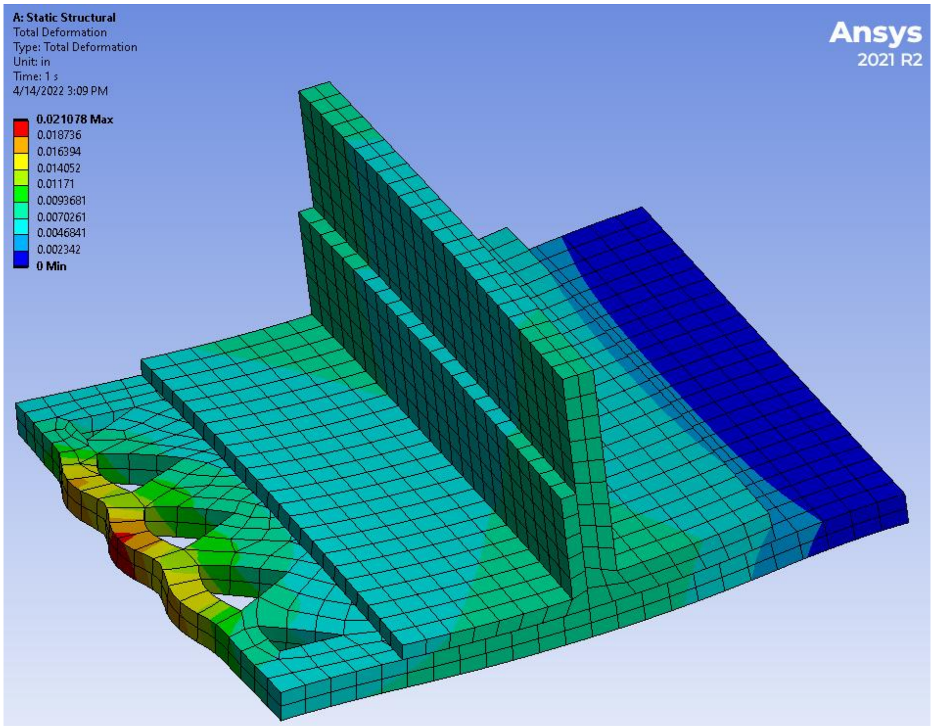


*Figure 12: Total Deformation - Mesh Size 0.21 and Quadratic Element Type.*

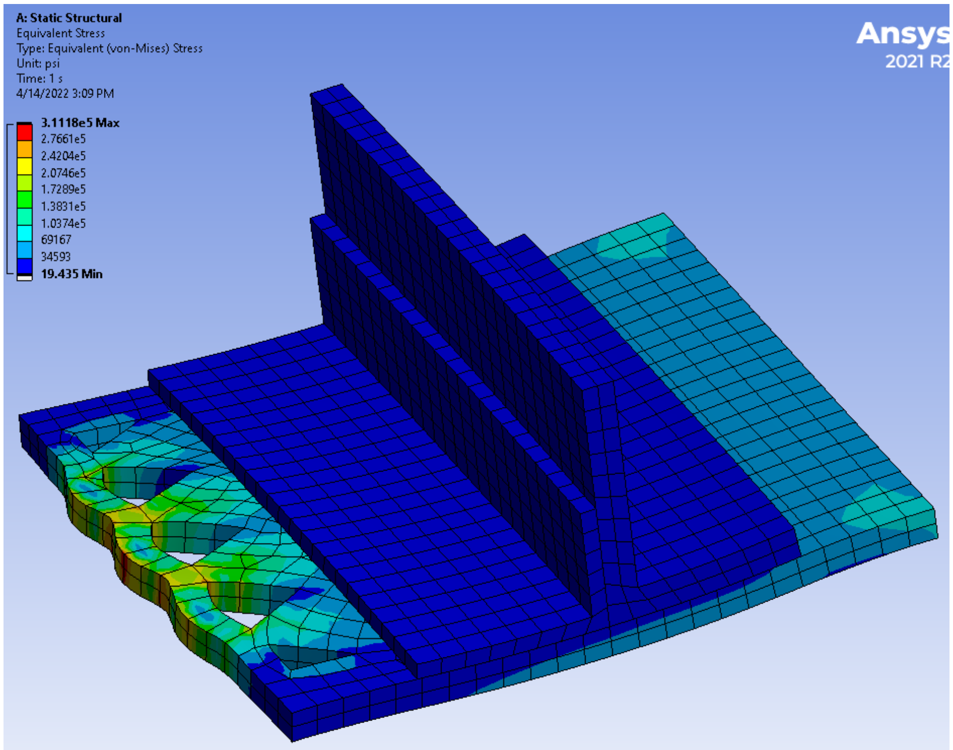


*Figure 13: Equivalent Von-Mises Stress - Mesh Size 0.21 and Quadratic Element Type.*

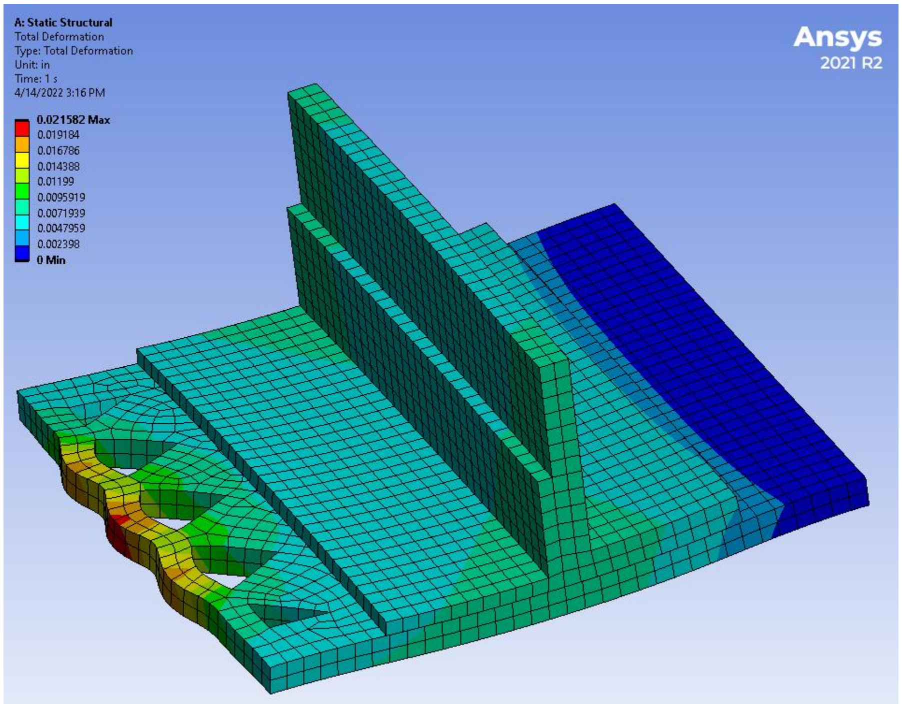


*Figure 14: Total Deformation - Mesh Size 0.15 and Quadratic Element Type.*

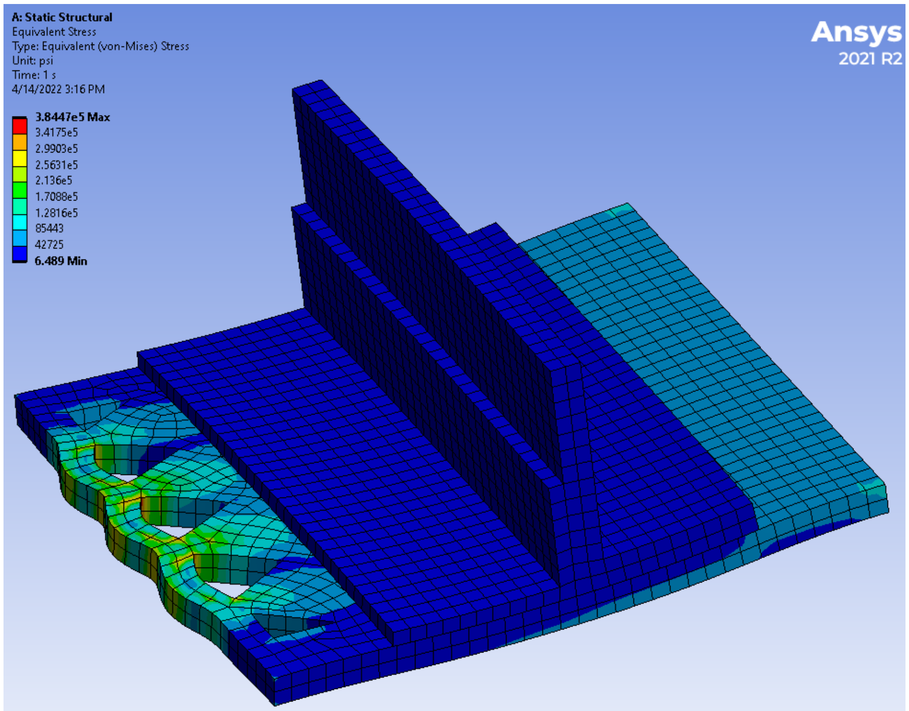


*Figure 15: Equivalent Von-Mises Stress - Mesh Size 0.15 and Quadratic Element Type.*

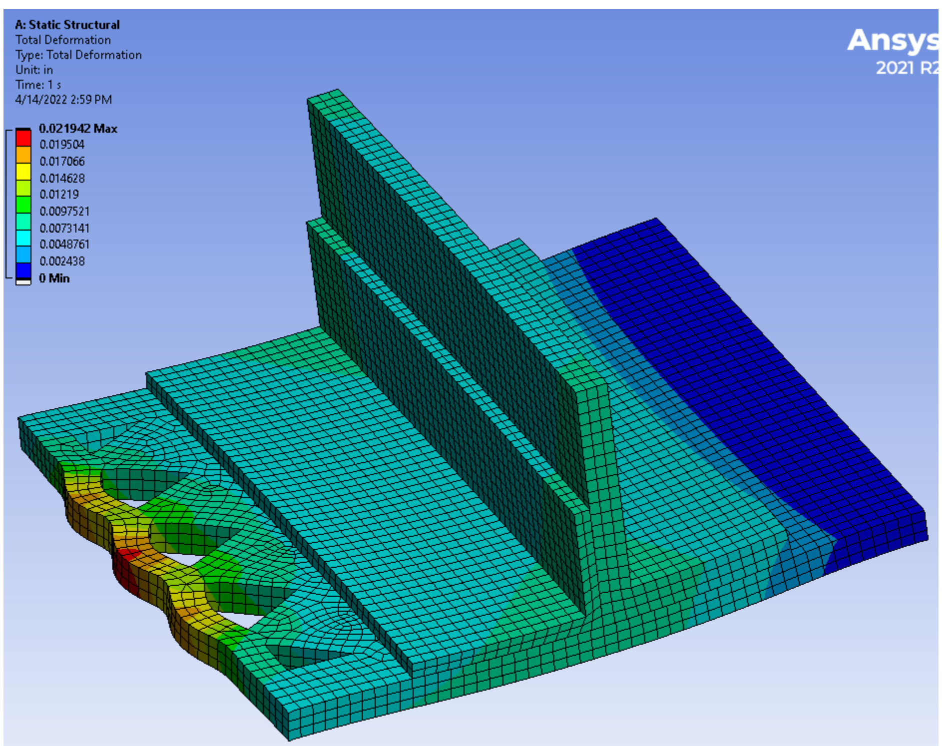


*Figure 16: Total Deformation - Mesh Size 0.1 and Quadratic Element Type.*

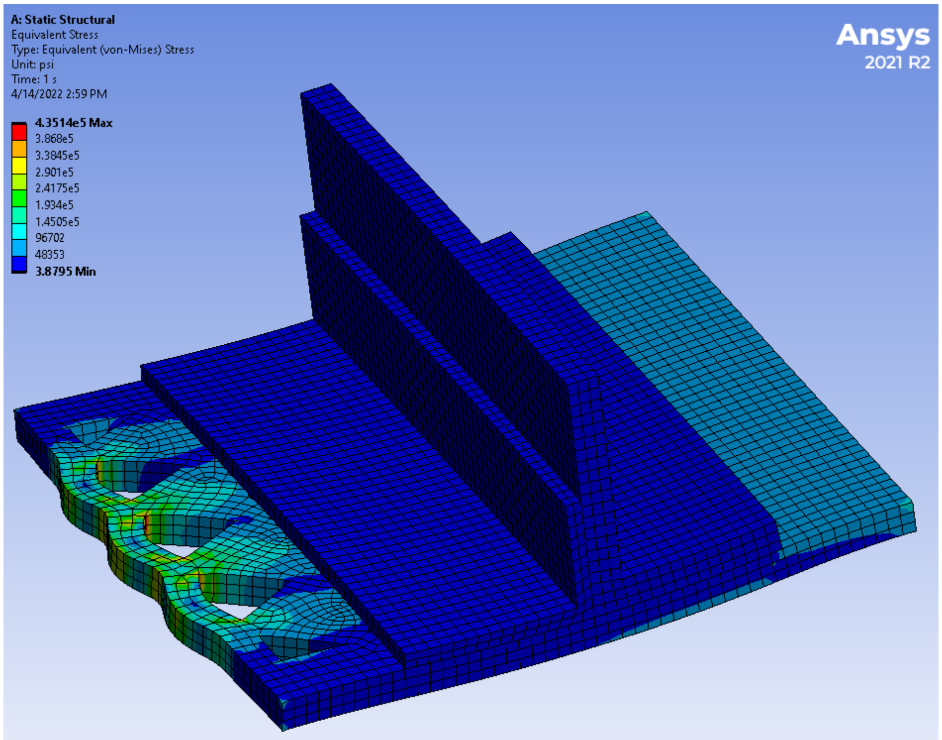


*Figure 17: Equivalent Von-Mises Stress - Mesh Size 0.1 and Quadratic Element Type.*

*Table 1: Convergence Test for Structure 1.*

| Mesh size and Element Type | Maximum Equivalent Von-Mises Stress |
|---|---|
| 0.21 and Linear Element | $2.4*10^5$ Psi |
| 0.15 and Linear Element | $2.8*10^5$ Psi |
| 0.1 and Linear Element | $2.95*10^5$ Psi |
| 0.21 and Quadratic Element | $3.1*10^5$ Psi |
| 0.15 and Quadratic Element | $3.8*10^5$ Psi |
| 0.1 and Quadratic Element | $4.3*10^5$ Psi |

As hypostasized before doing the analysis and as seen in the above table (Table 1), the maximum stress values approach its theoretical values as the mesh sizes are refined. For a linear element the stress converges with decreasing the mesh size. The variation in stress values for a given mesh size for the two-element types is because a quadratic element considers nodes in between the two nodes of a linear element. Therefore, giving a more accurate result.

## Modal Analysis

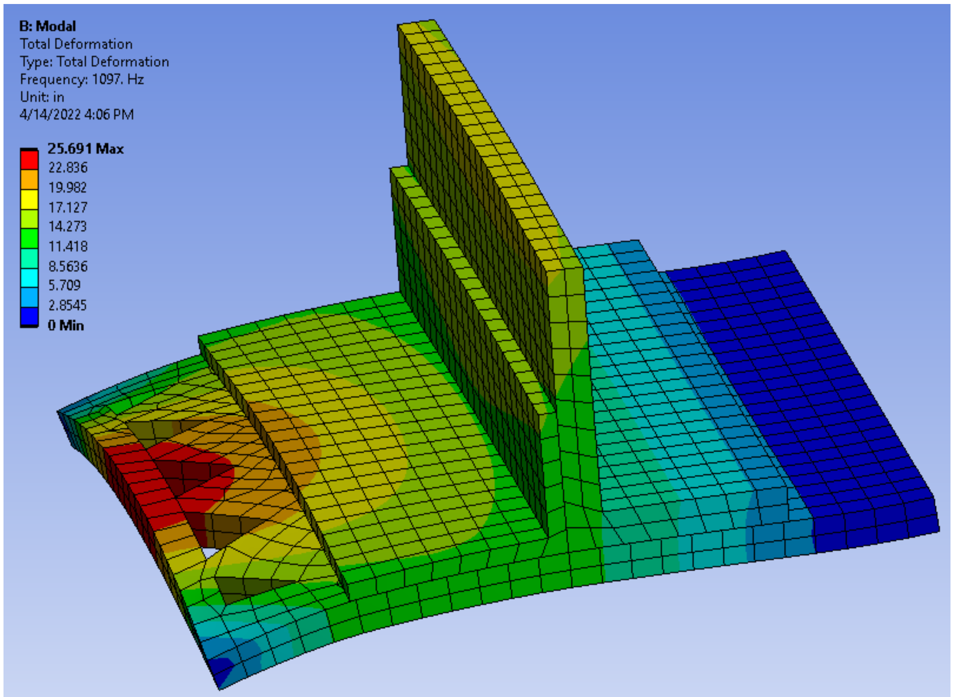


*Figure 18: Modal Analysis - First Frequency, Mesh Size 0.21, and Linear Element Type.*

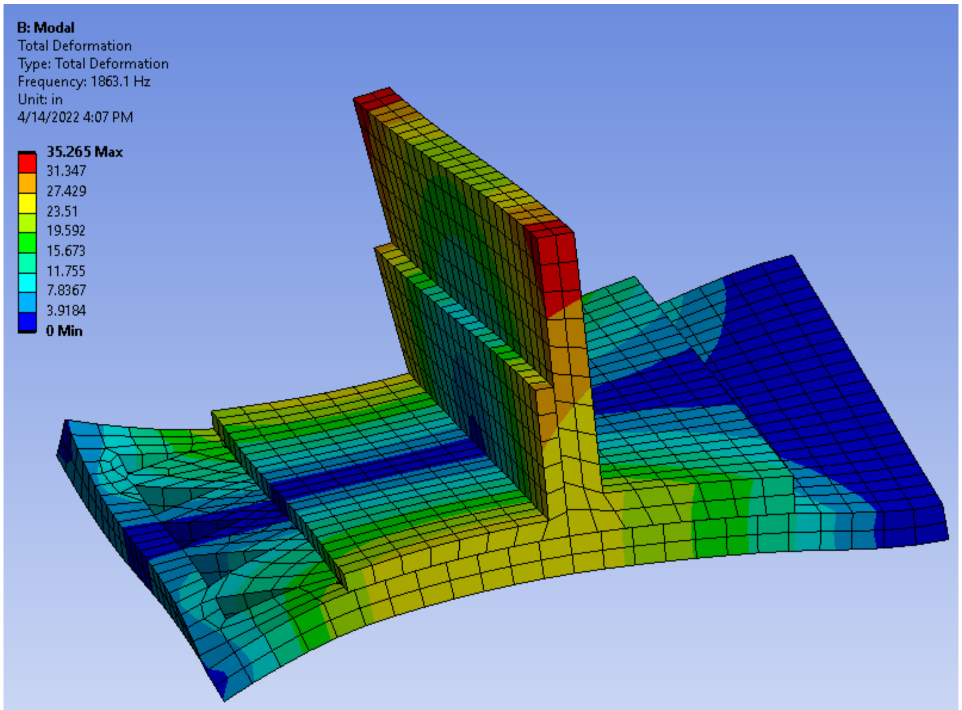


*Figure 19: Modal Analysis - Second Frequency, Mesh Size 0.21, and Linear Element Type.*

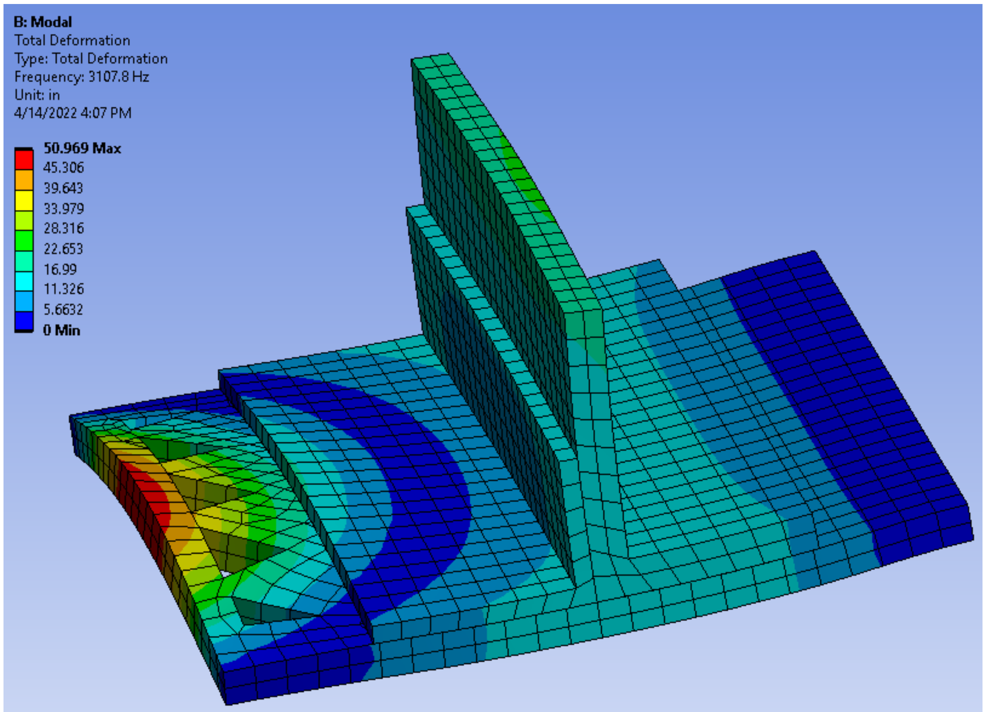

*Figure 20: Modal Analysis - Third Frequency, Mesh Size 0.21, and Linear Element Type.*

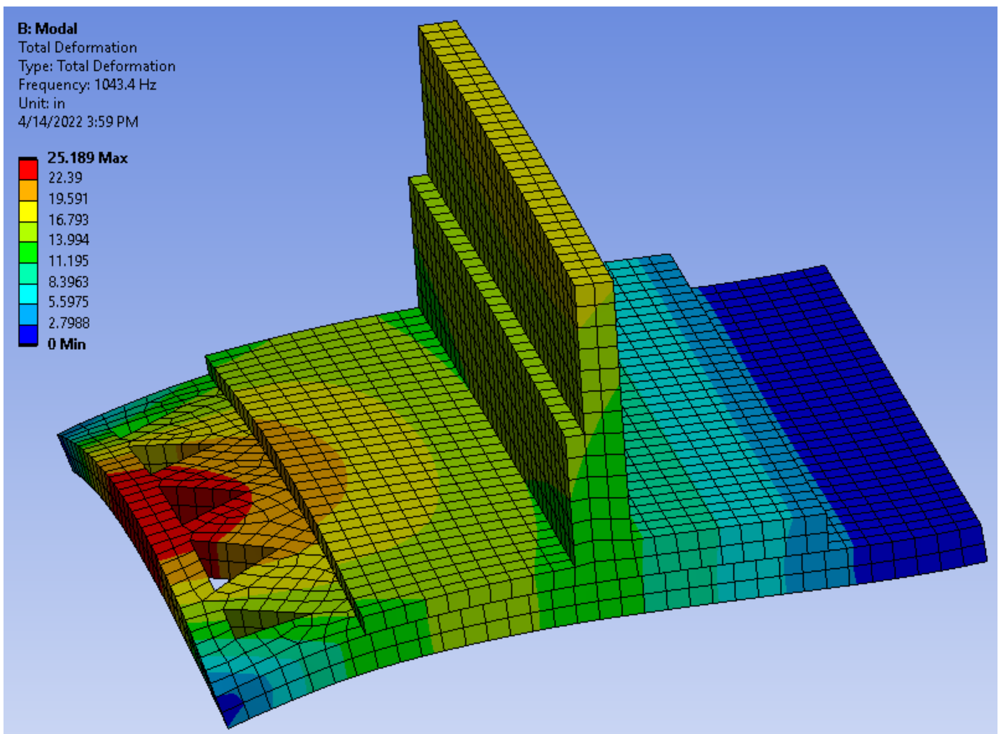

*Figure 21: Modal Analysis - First Frequency, Mesh Size 0.15, and Linear Element Type.*

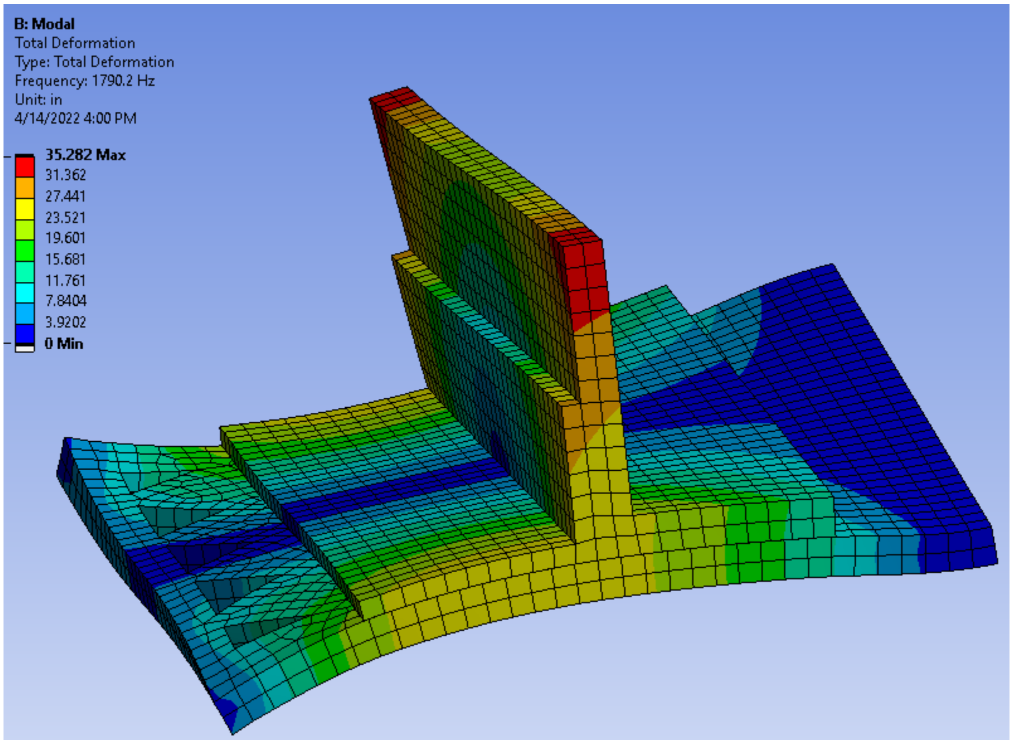


*Figure 22: Modal Analysis - Second Frequency, Mesh Size 0.15, and Linear Element Type.*

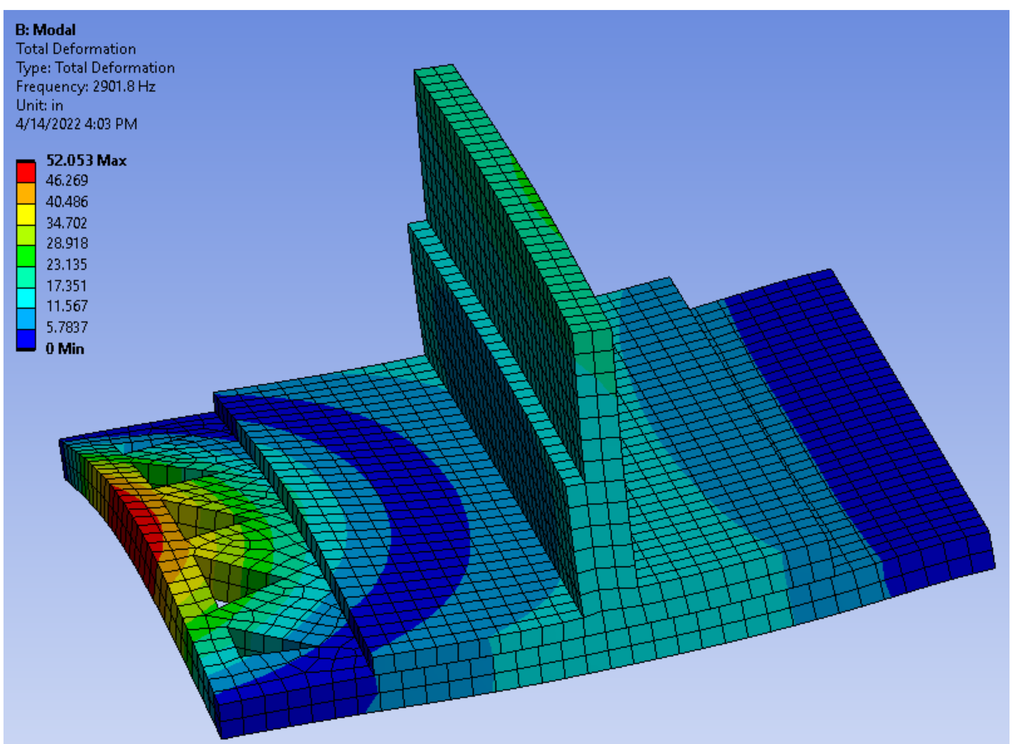


*Figure 23: Modal Analysis - Third Frequency, Mesh Size 0.15, and Linear Element Type.*

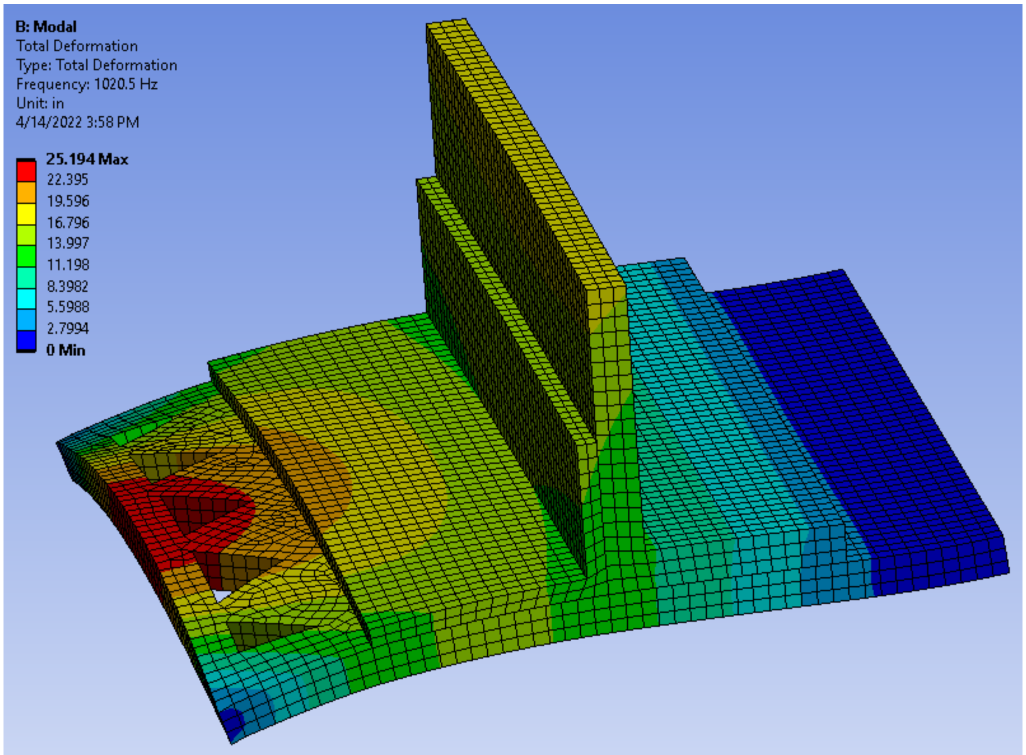


*Figure 24: Modal Analysis - First Frequency, Mesh Size 0.1, and Linear Element Type.*

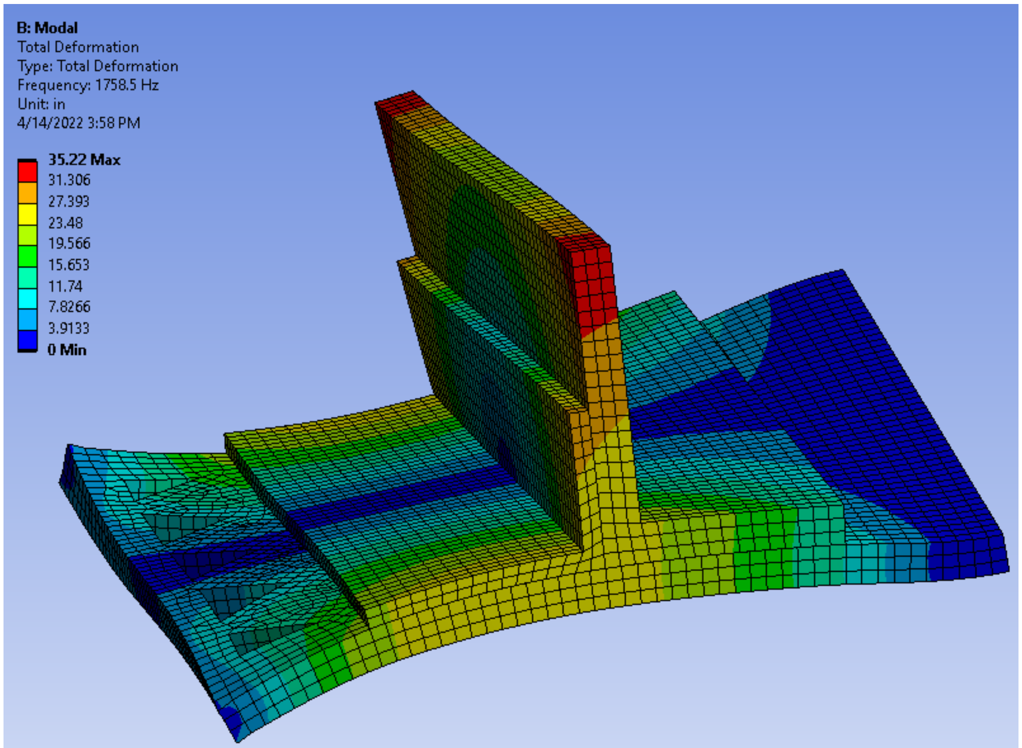


*Figure 25: Modal Analysis - Second Frequency, Mesh Size 0.1, and Linear Element Type.*

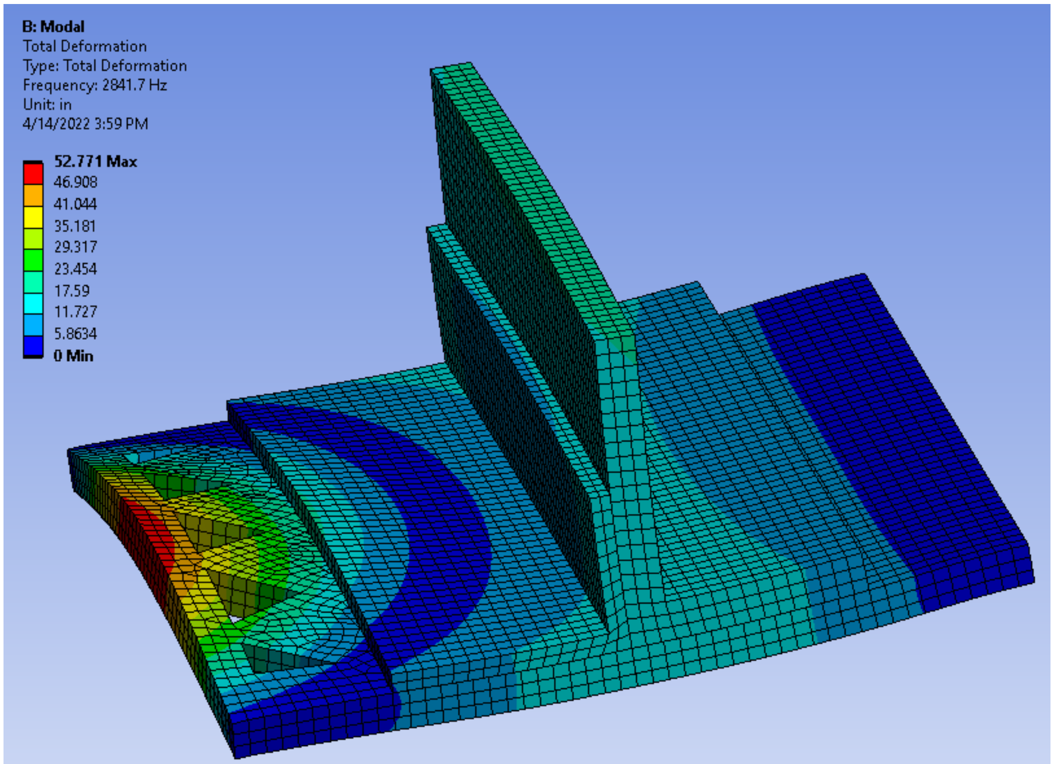


*Figure 26: Modal Analysis - Third Frequency, Mesh Size 0.1, and Linear Element Type.*

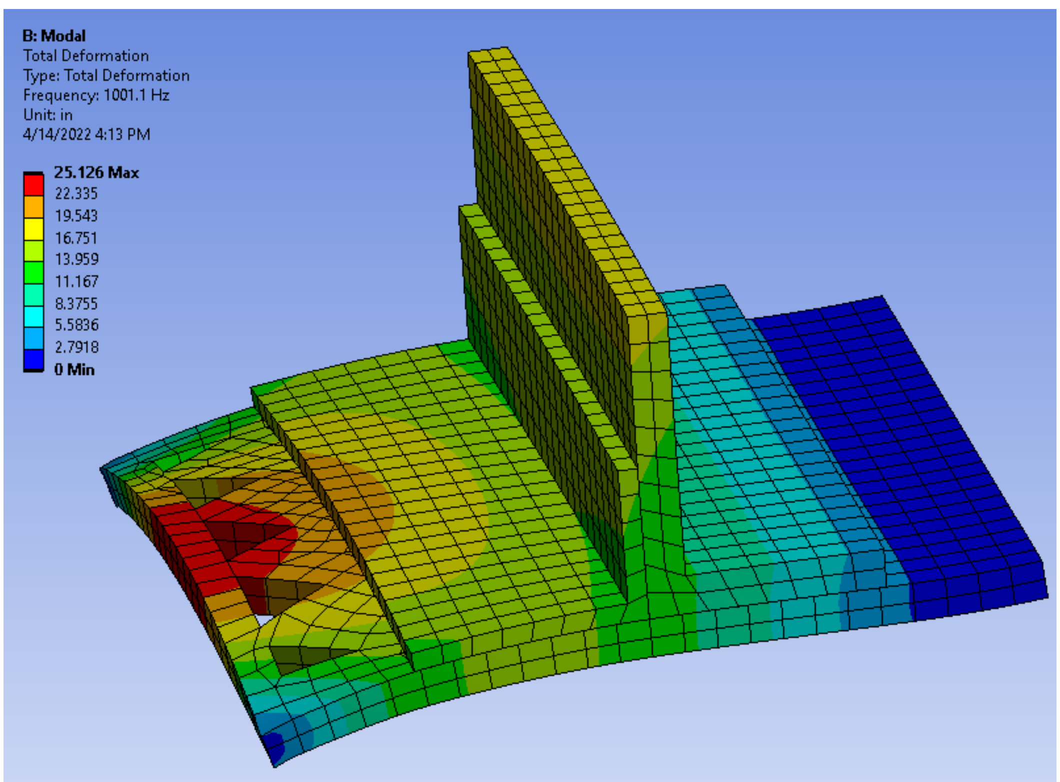


*Figure 27: Modal Analysis - First Frequency, Mesh Size 0.21, and Quadratic Element Type.*

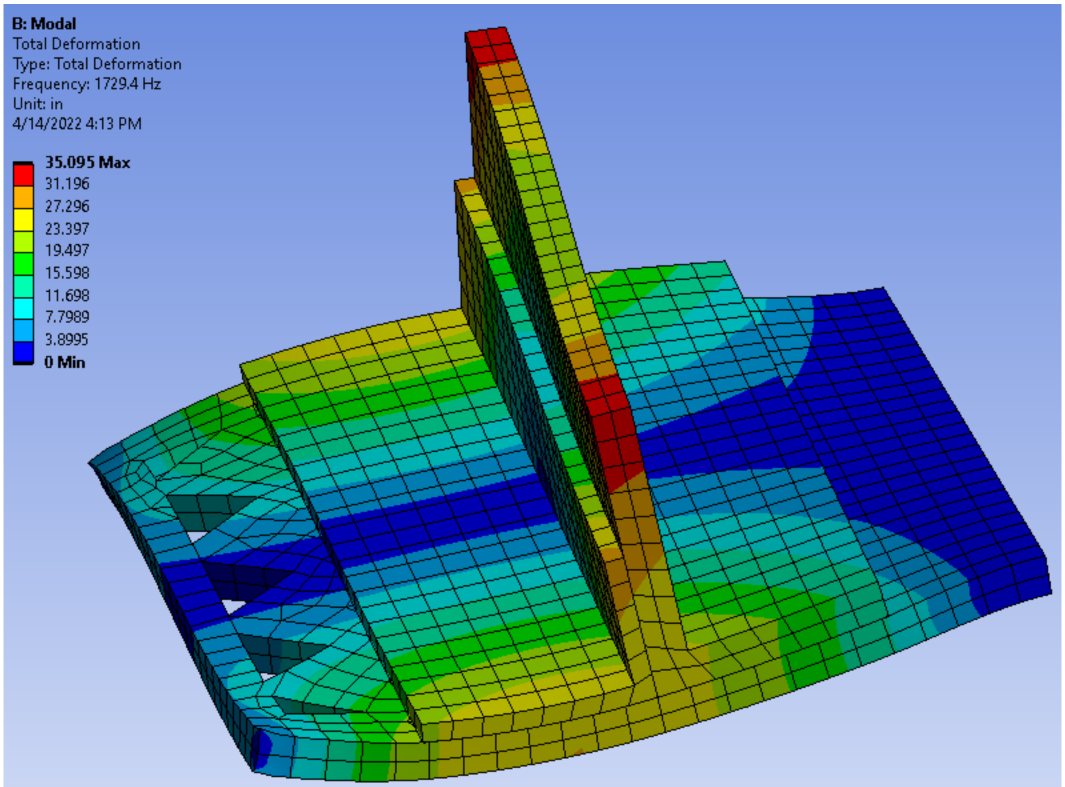

*Figure 28: Modal Analysis - Second Frequency, Mesh Size 0.21, and Quadratic Element Type.*

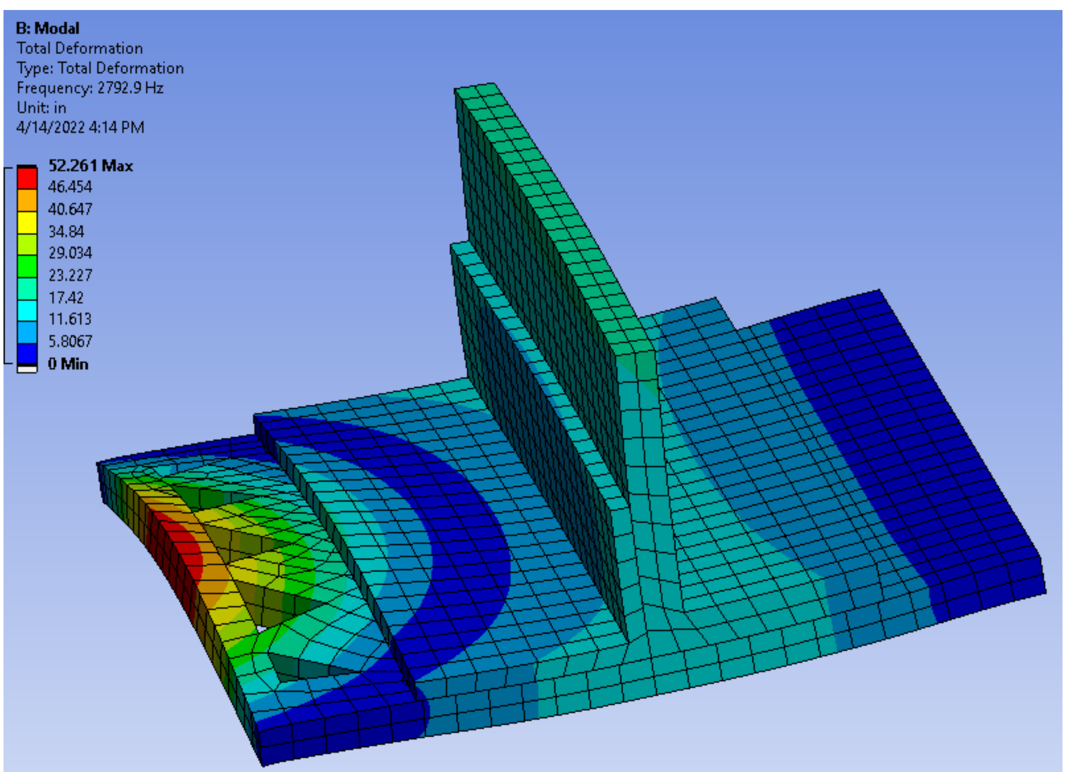

*Figure 29: Modal Analysis - Third Frequency, Mesh Size 0.21, and Quadratic Element Type.*

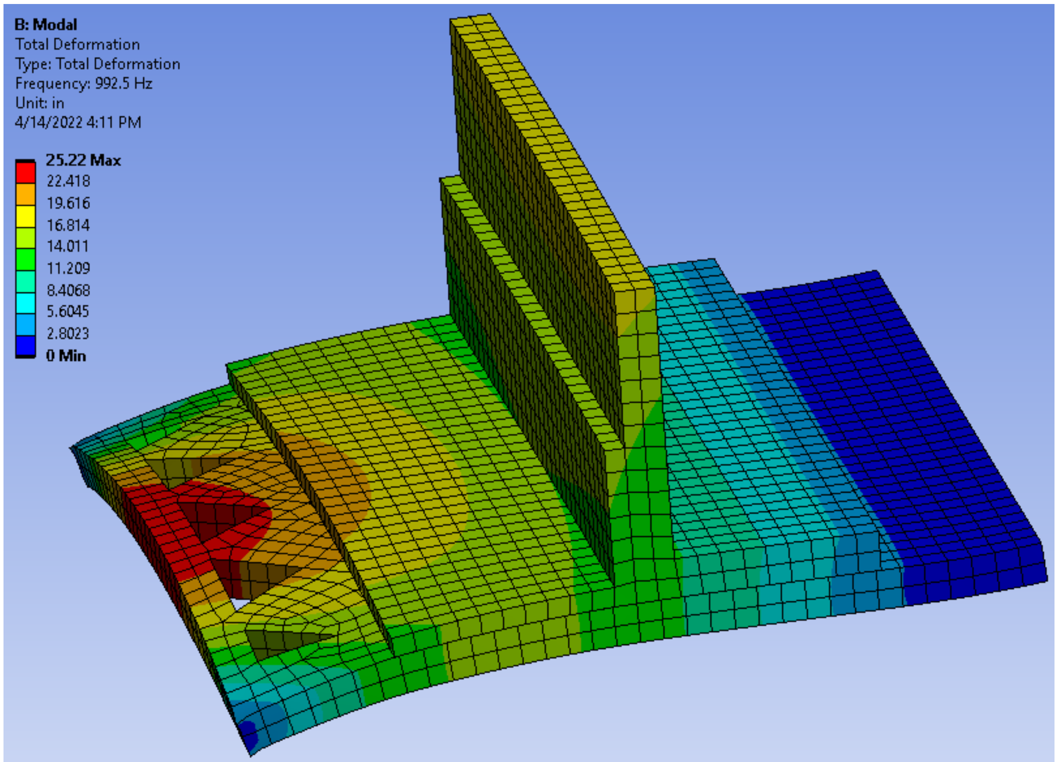


*Figure 30: Modal Analysis - First Frequency, Mesh Size 0.15, and Quadratic Element Type.*

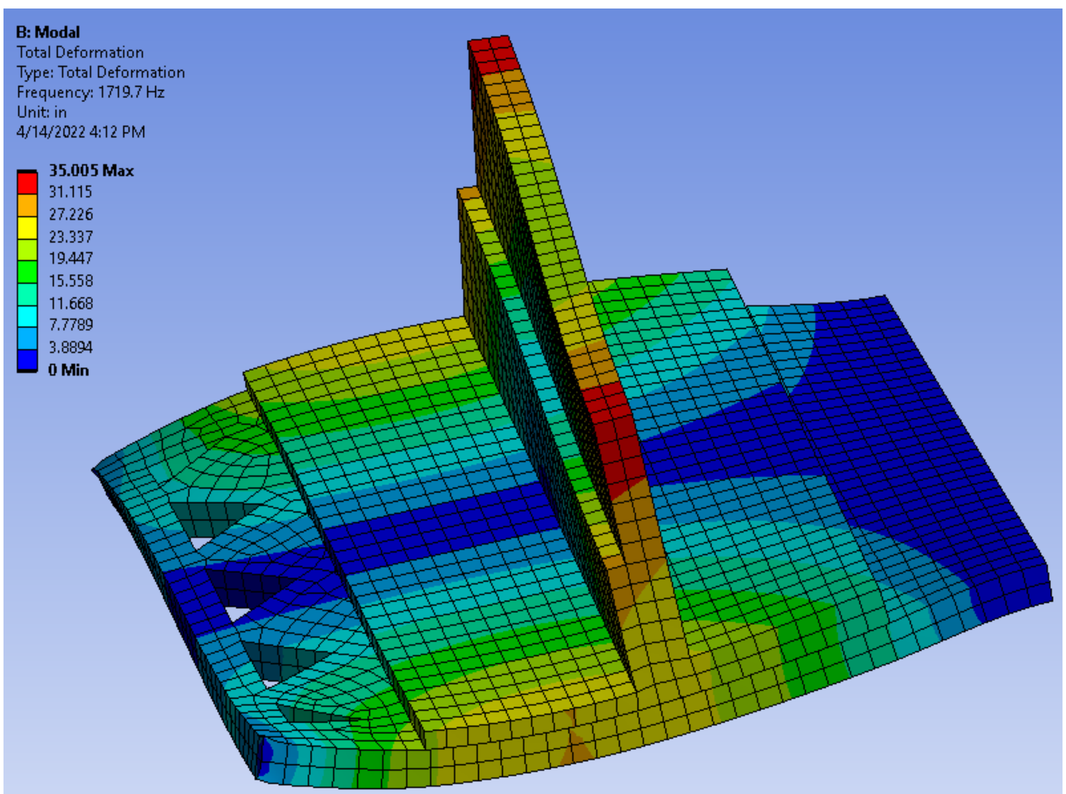


*Figure 31: Modal Analysis - Second Frequency, Mesh Size 0.15, and Quadratic Element Type.*

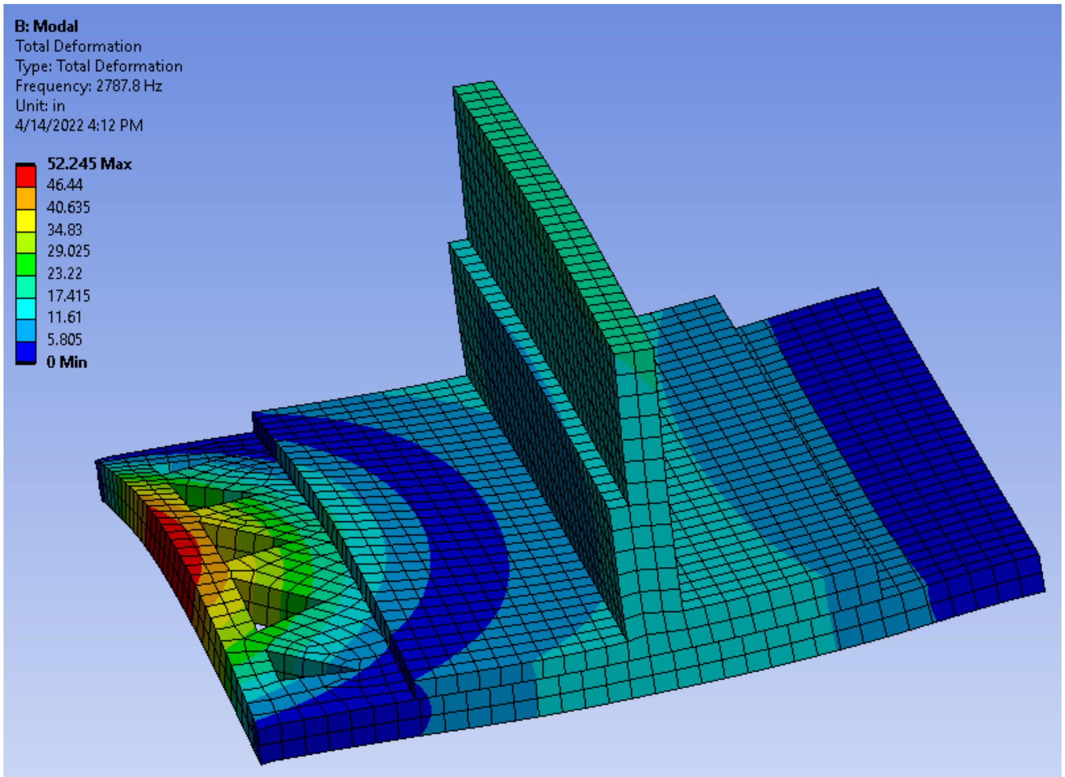


*Figure 32: Modal Analysis - Third Frequency, Mesh Size 0.15, and Quadratic Element Type.*

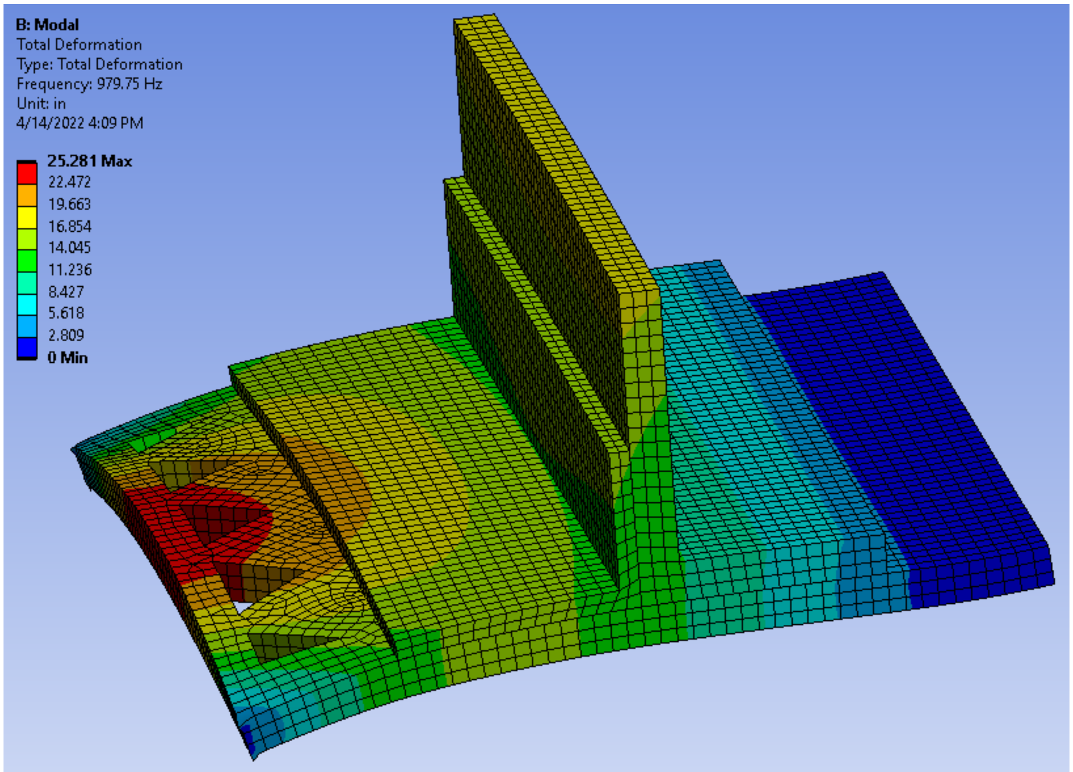


*Figure 33: Modal Analysis - First Frequency, Mesh Size 0.1, and Quadratic Element Type.*

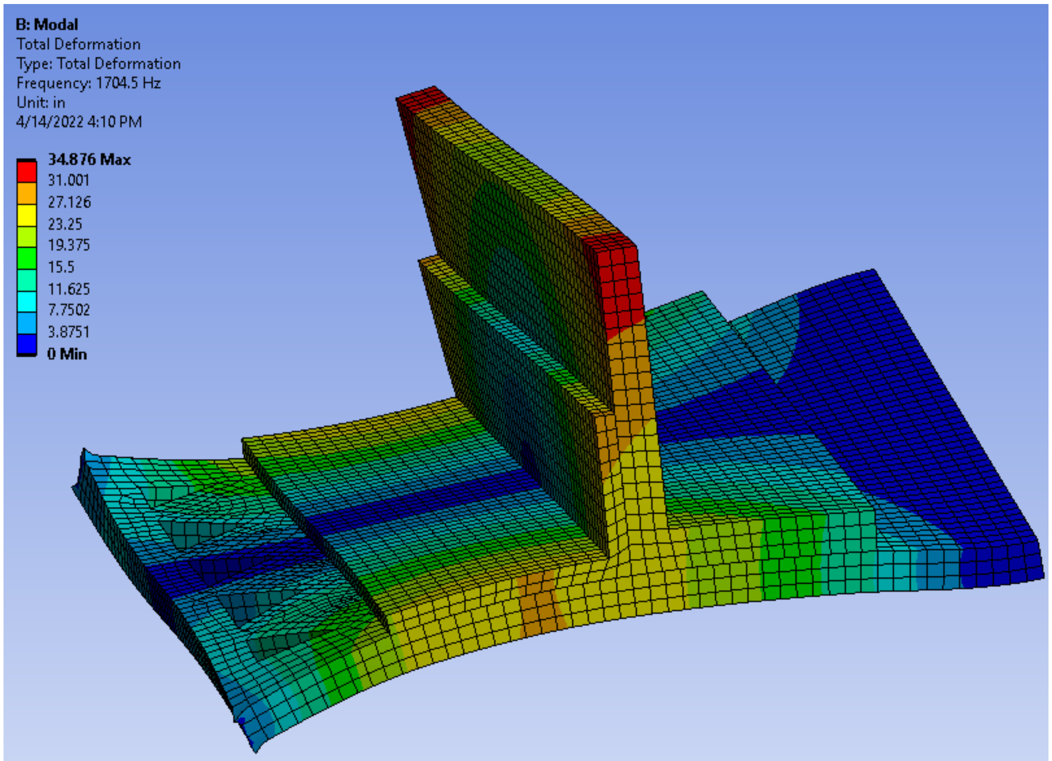


*Figure 34: Modal Analysis - Second Frequency, Mesh Size 0.1, and Quadratic Element Type.*

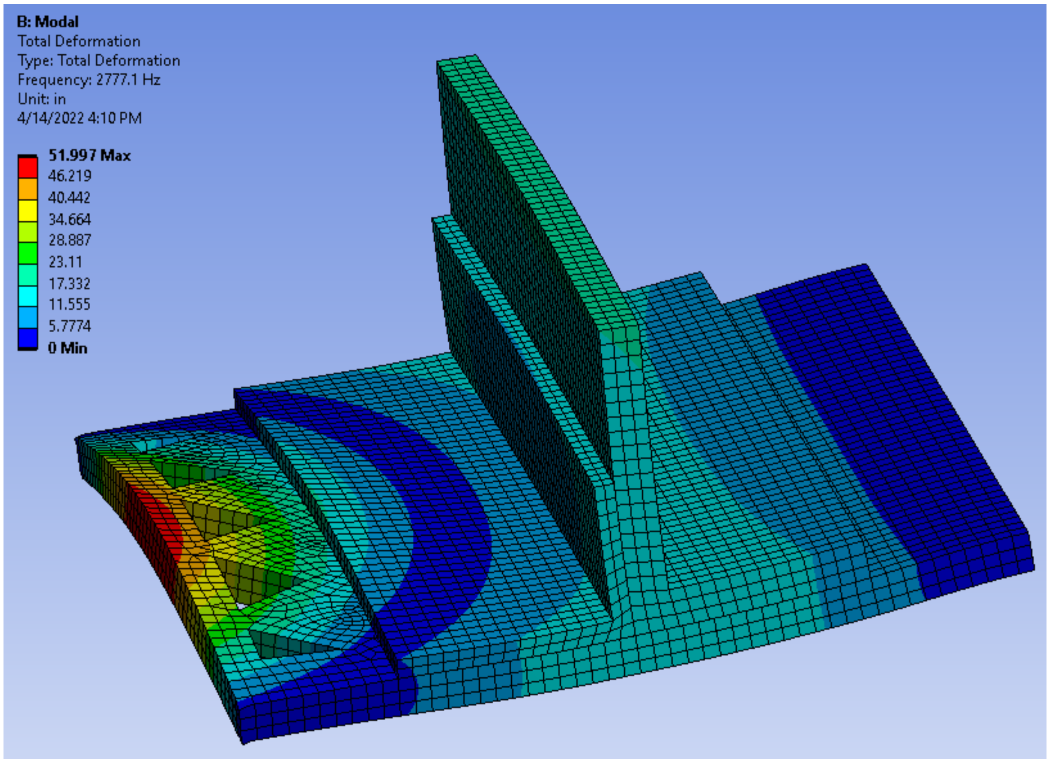


*Figure 35: Modal Analysis - Third Frequency, Mesh Size 0.1, and Quadratic Element Type.*

*Table 2: Comparing the First Three Natural Frequencies.*

| Mesh Size and Element Type | First Frequency – Deformation (in) | Second Frequency – Deformation (in) | Third Frequency – Deformation (in) |
|---|---|---|---|
| 0.21 and Linear Element | 25.69 | 35.26 | 50.97 |
| 0.15 and Linear Element | 25.20 | 35.28 | 52.05 |
| 0.1 and Linear Element | 25.19 | 35.22 | 52.77 |
| 0.21 and Quadratic Element | 25.13 | 35.09 | 52.26 |
| 0.15 and Quadratic Element | 25.22 | 35.00 | 52.24 |
| 0.1 and Quadratic Element | 25.28 | 34.87 | 52.00 |

The above table (Table 2) tabulates the maximum displacements form each natural frequency for each mesh size and element type. The first, second and third frequency displacements were recorded to be consistent for the given mesh sizes and element types.

## Structure 2

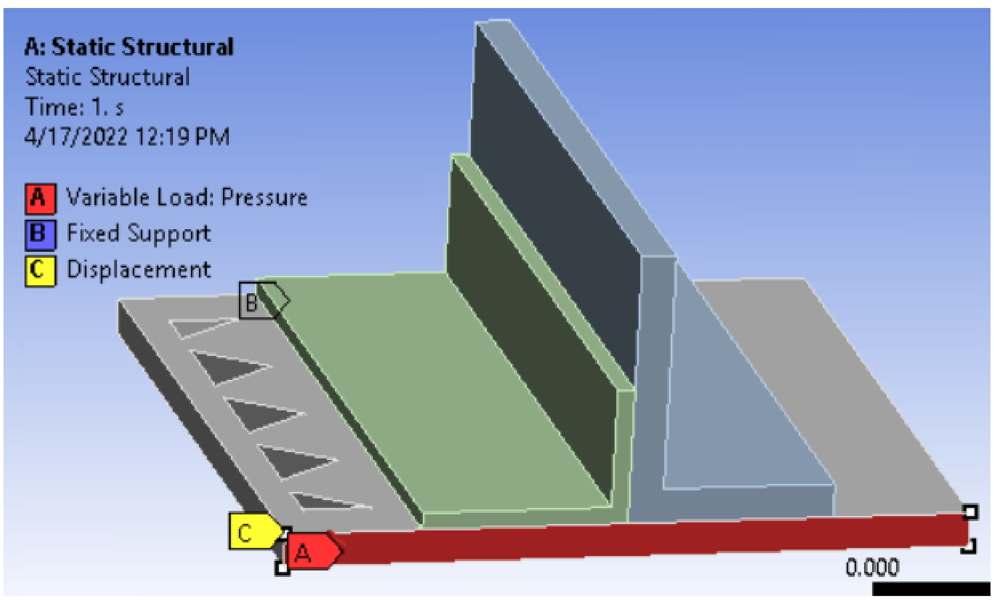


*Figure 36: Second Structure.*

## Static Analysis

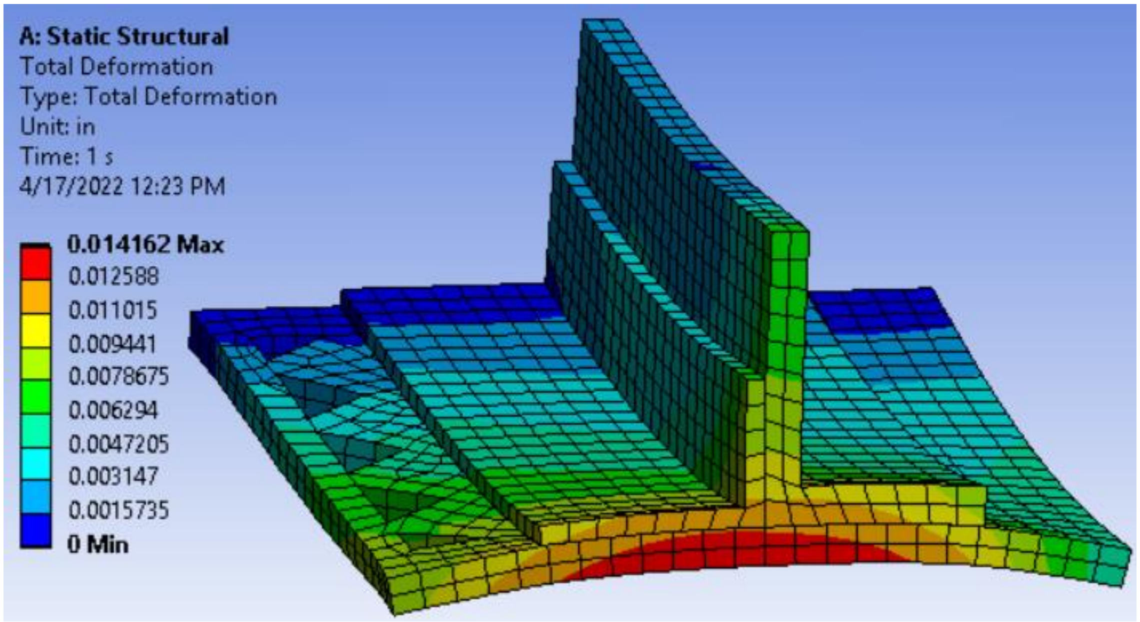


*Figure 37: Total Deformation - Mesh Size 0.21 and Linear Element Type.*

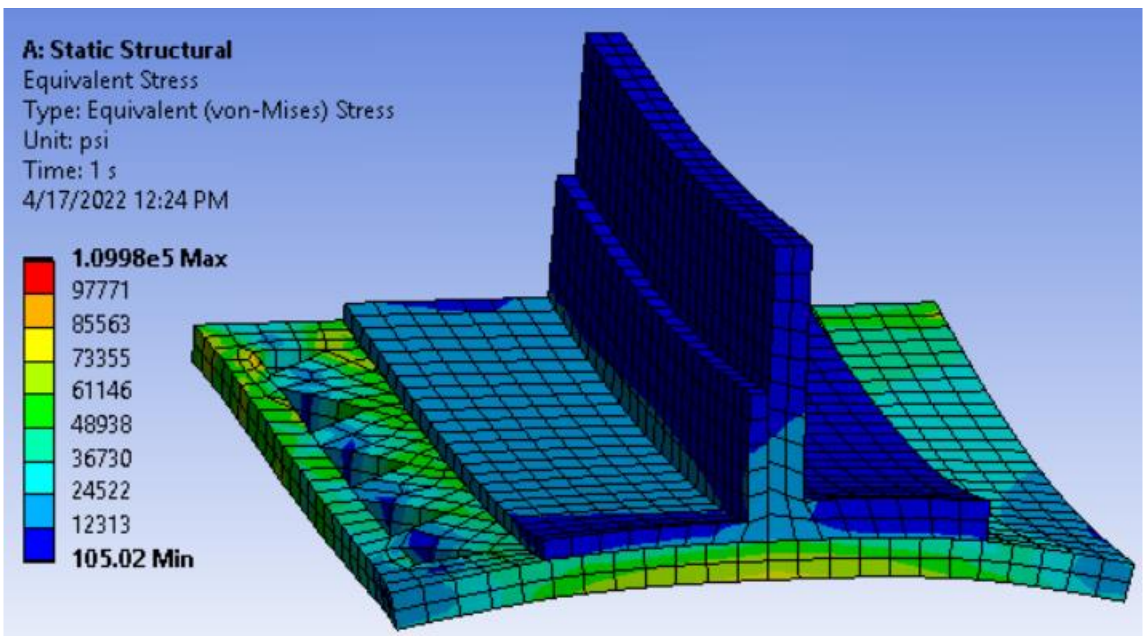


*Figure 38: Equivalent Von-Mises Stress - Mesh Size 0.21 and Linear Element Type.*

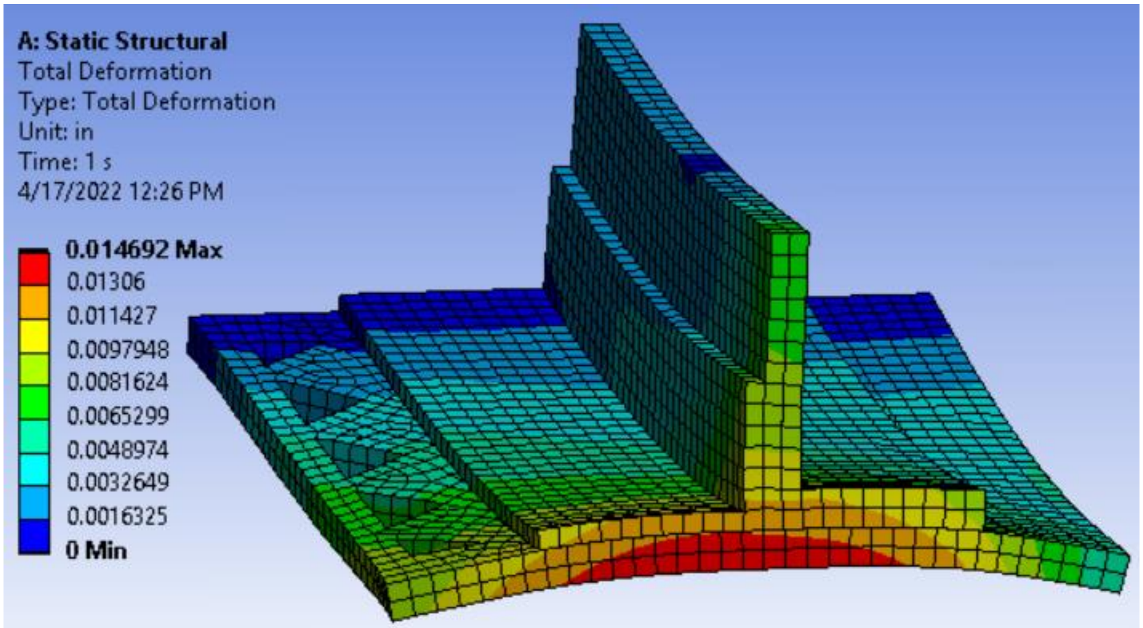


*Figure 39: Total Deformation - Mesh Size 0.15 and Linear Element Type.*

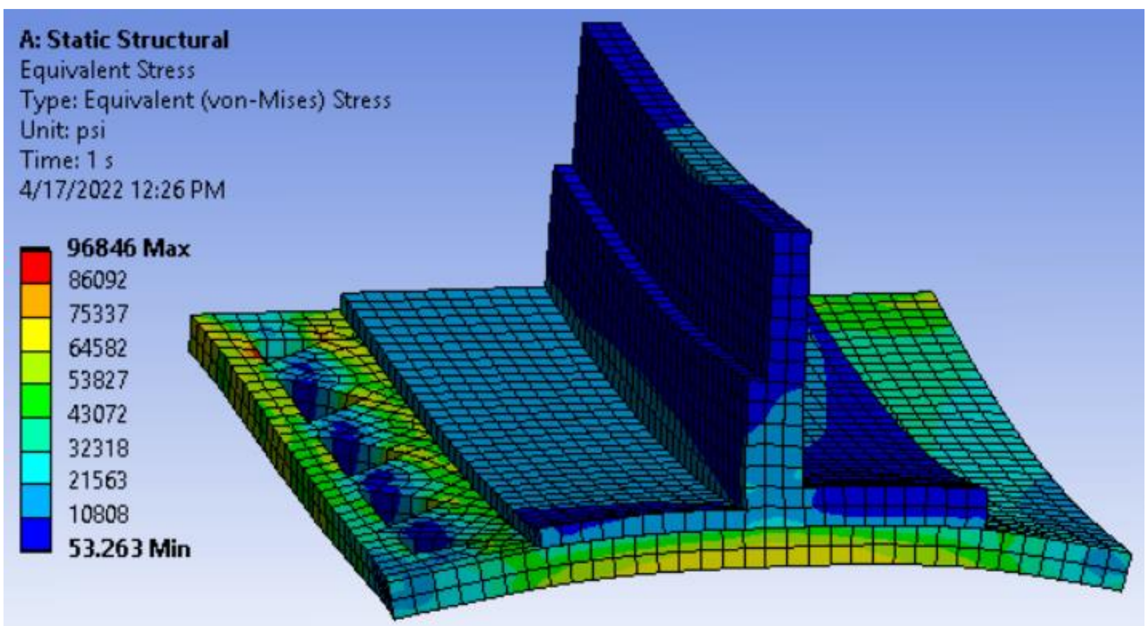


*Figure 40: Equivalent Von-Mises Stress - Mesh Size 0.15 and Linear Element Type.*

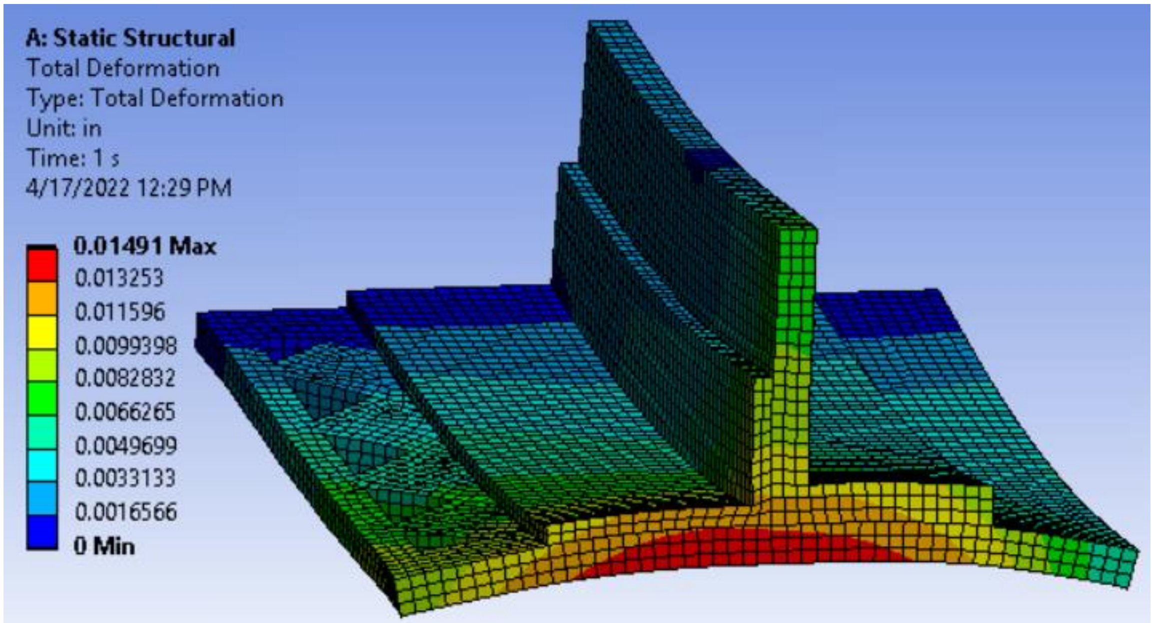


*Figure 41: Total Deformation - Mesh Size 0.1 and Linear Element Type.*

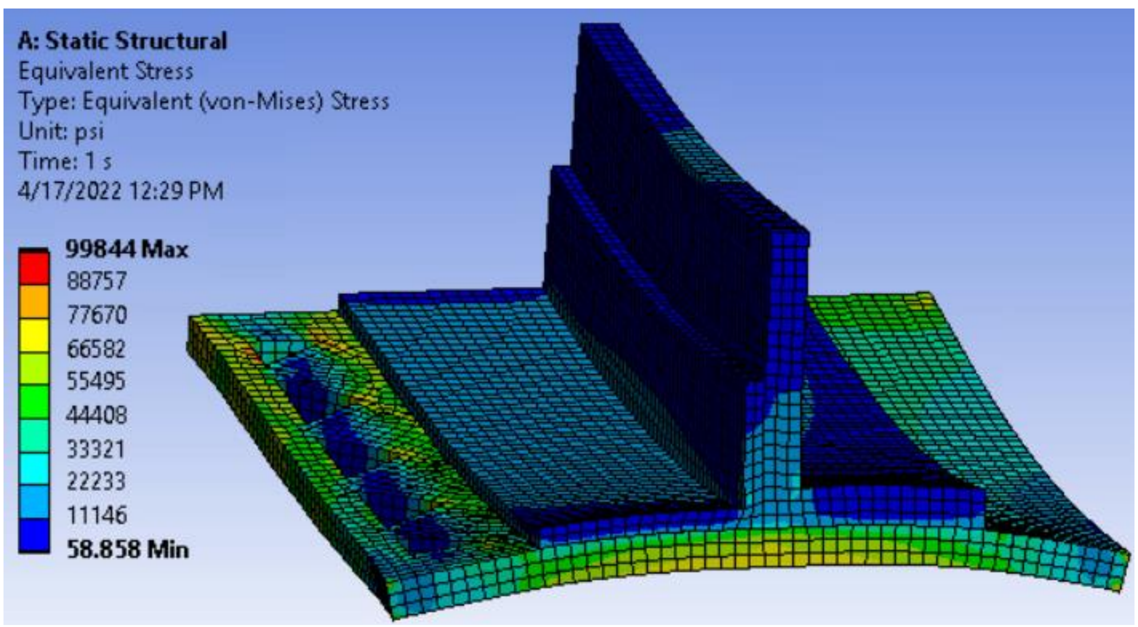


*Figure 42: Equivalent Von-Mises Stress - Mesh Size 0.1 and Linear Element Type.*

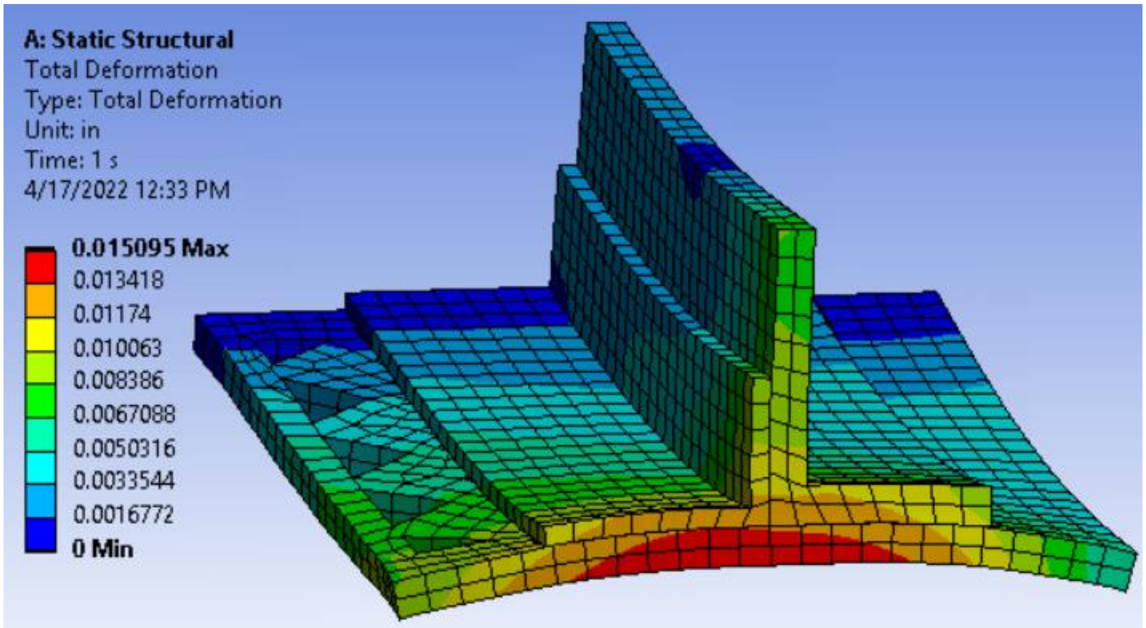


*Figure 43: Total Deformation - Mesh Size 0.21 and Quadratic Element Type.*

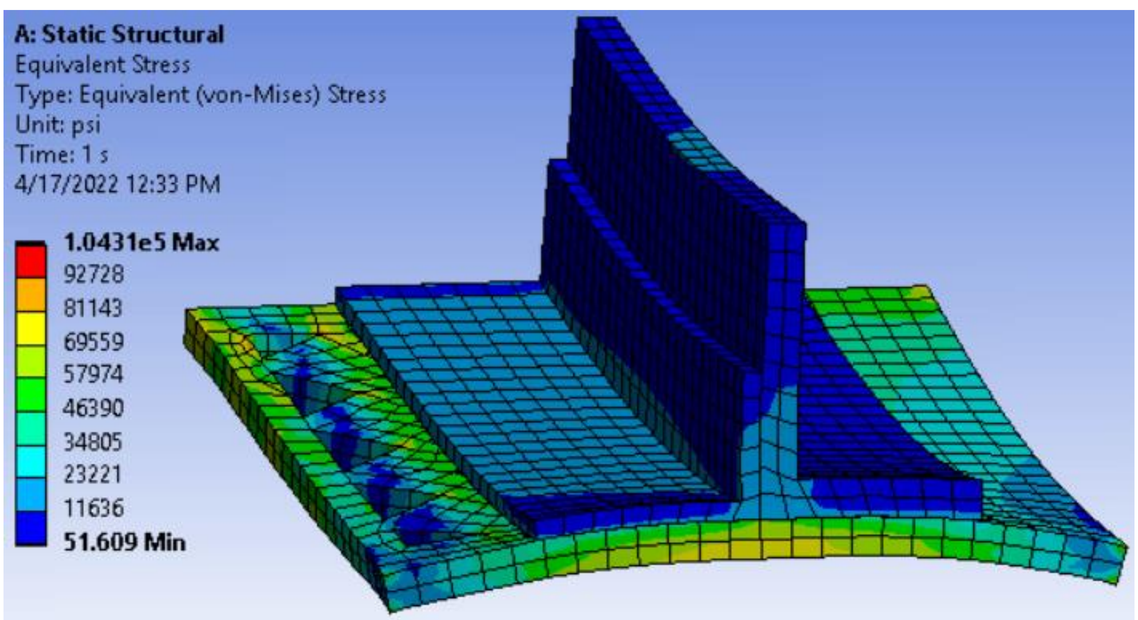


*Figure 44: Equivalent Von-Mises Stress - Mesh Size 0.21 and Quadratic Element Type.*

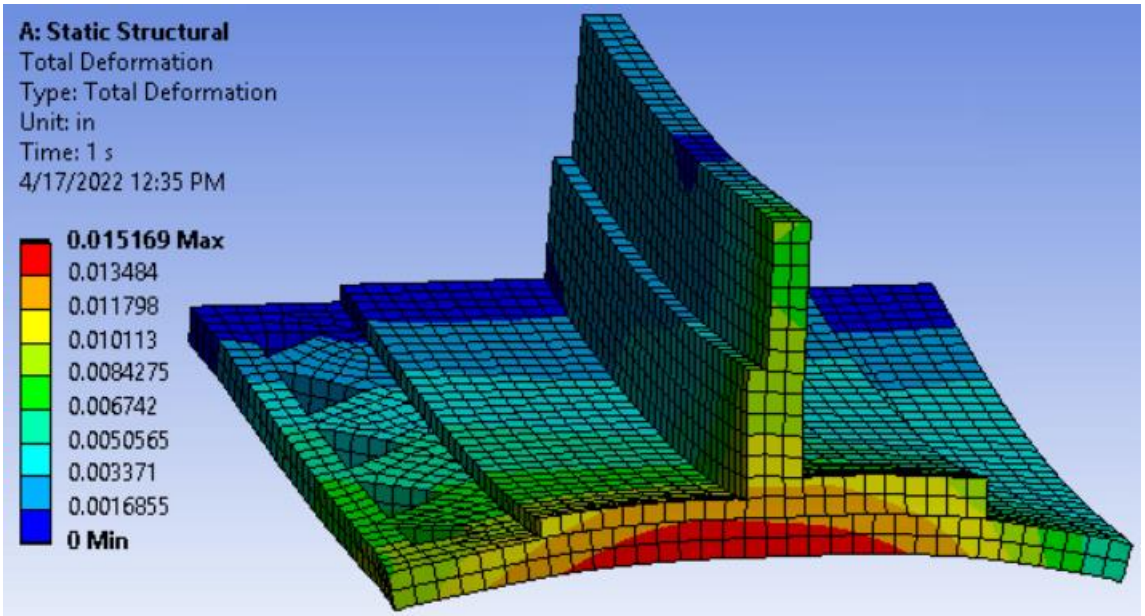


*Figure 45: Total Deformation - Mesh Size 0.15 and Quadratic Element Type.*

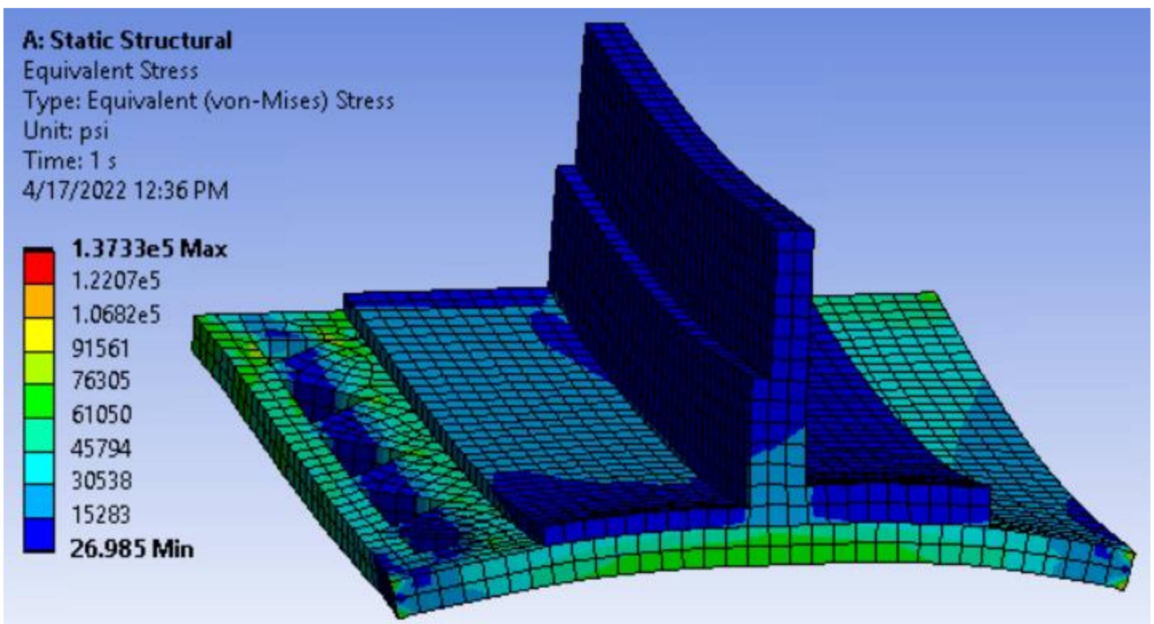


*Figure 46: Equivalent Von-Mises Stress - Mesh Size 0.15 and Quadratic Element Type.*

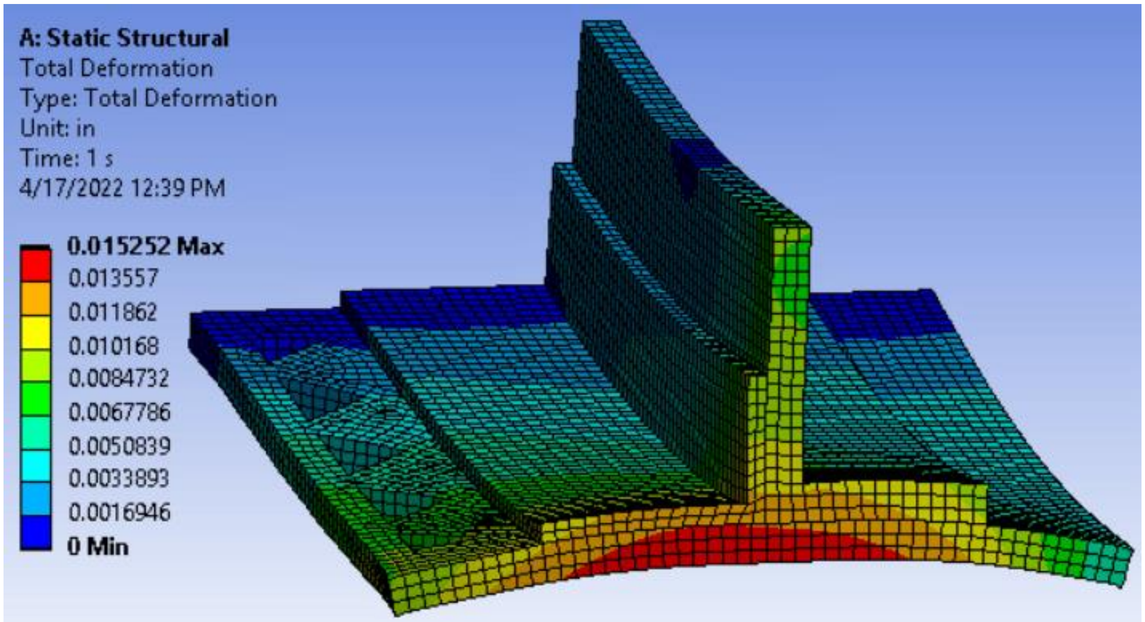


*Figure 47: Total Deformation - Mesh Size 0.1 and Quadratic Element Type.*

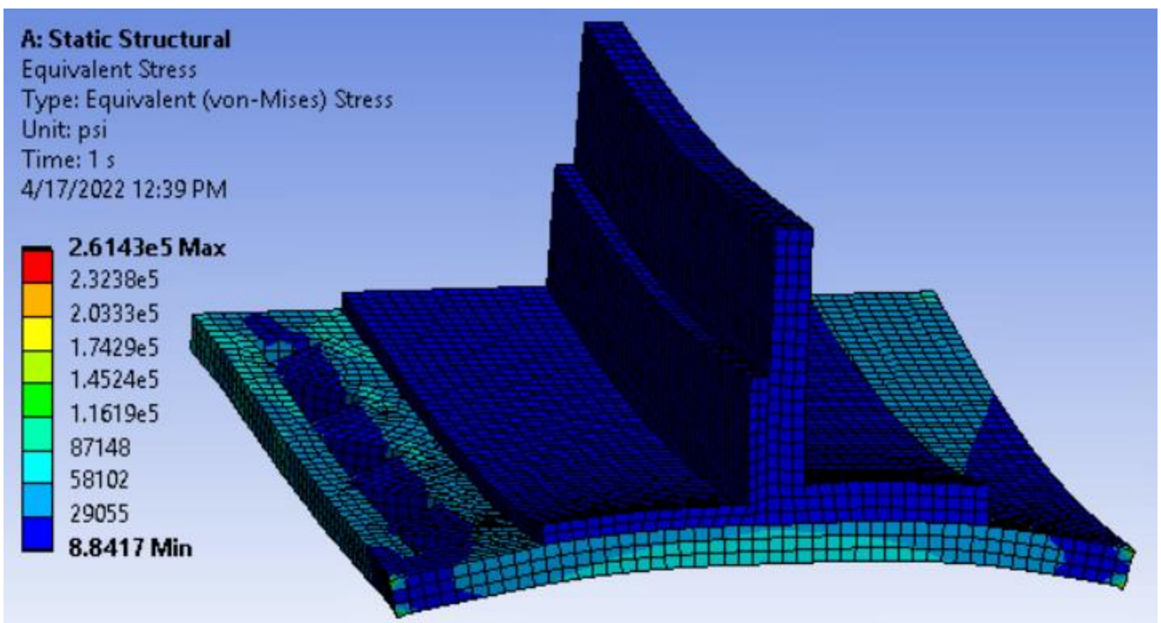


*Figure 48: Equivalent Von-Mises Stress - Mesh Size 0.1 and Quadratic Element Type.*

*Table 3: Convergence Test for Structure 2.*

| Mesh size and Element Type | Maximum Equivalent Von-Mises Stress |
|---|---|
| 0.21 and Linear Element | $1.1*10^5$ Psi |
| 0.15 and Linear Element | $0.96*10^5$ Psi |
| 0.1 and Linear Element | $0.99*10^5$ Psi |
| 0.21 and Quadratic Element | $1.04*10^5$ Psi |
| 0.15 and Quadratic Element | $1.4*10^5$ Psi |
| 0.1 and Quadratic Element | $2.6*10^5$ Psi |

As hypostasized before doing the analysis and as seen in the above table (Table 3), the maximum stress values approach its theoretical values as the mesh sizes are refined. For a linear element the stress converges with decreasing the mesh size. The maximum stress for a linear element converges to $1*10^5$ Psi. The variation in stress values for a given mesh size for the two-element types is because a quadratic element considers nodes in between the two nodes of a linear element. Therefore, giving a more accurate result.

## Modal Analysis

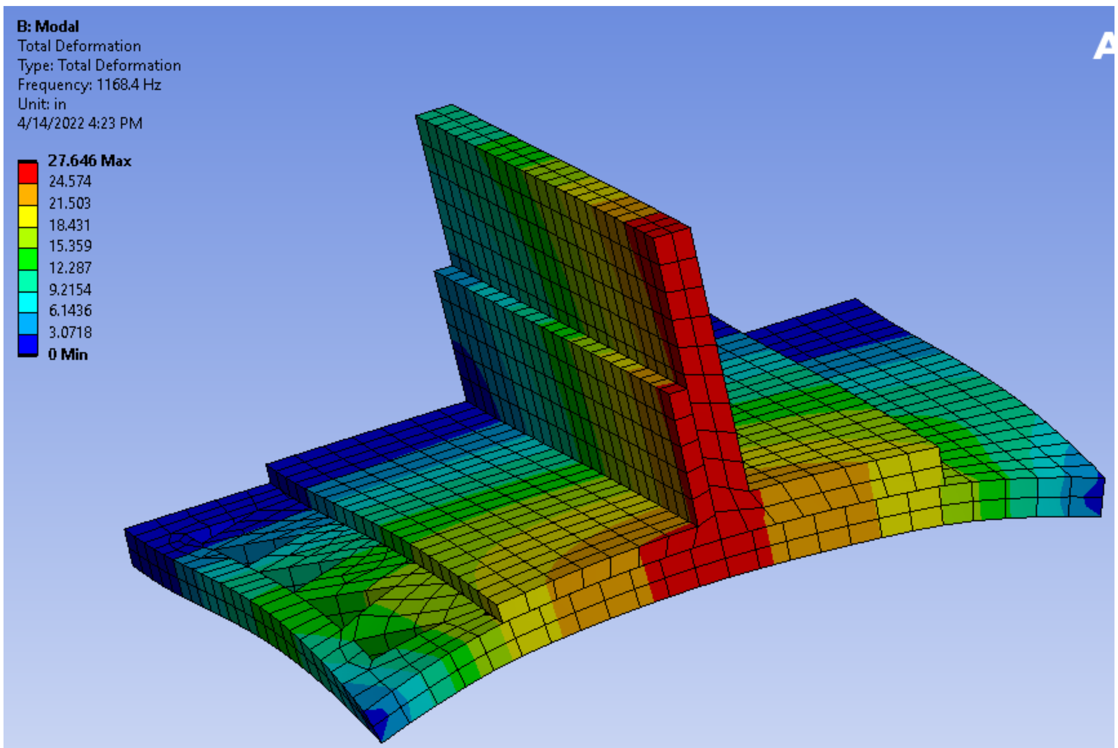


*Figure 49: Modal Analysis - First Frequency, Mesh Size 0.21, and Linear Element Type.*

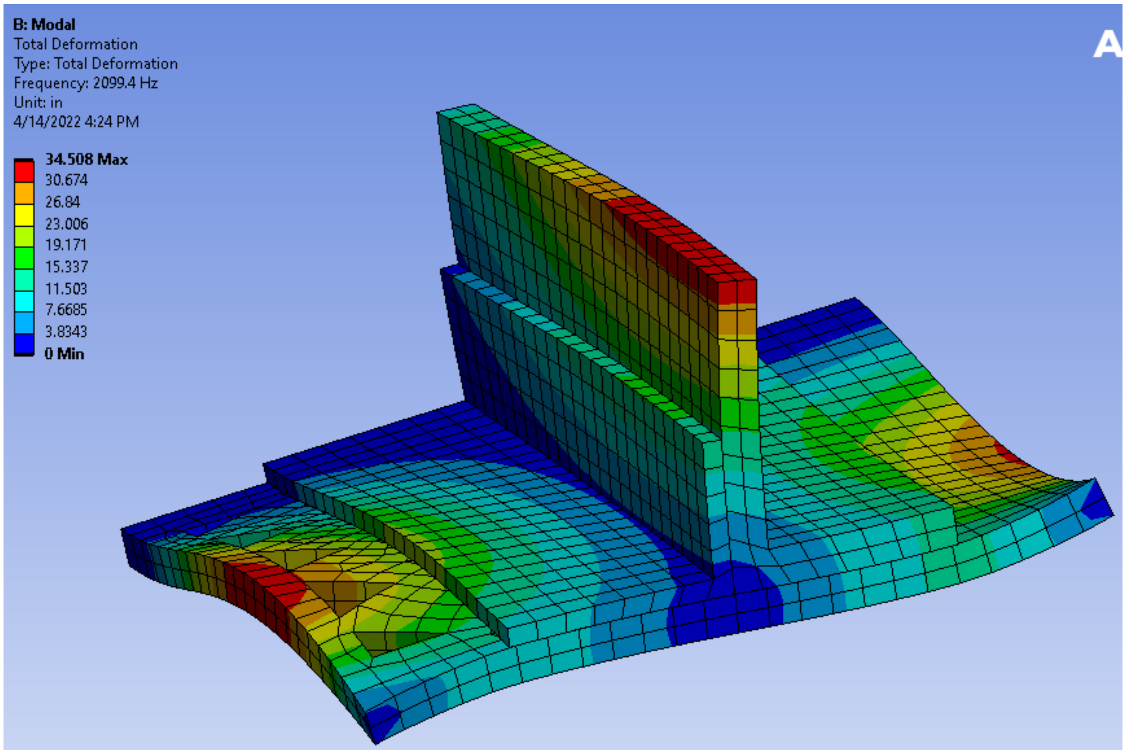


*Figure 50: Modal Analysis - Second Frequency, Mesh Size 0.21, and Linear Element Type.*

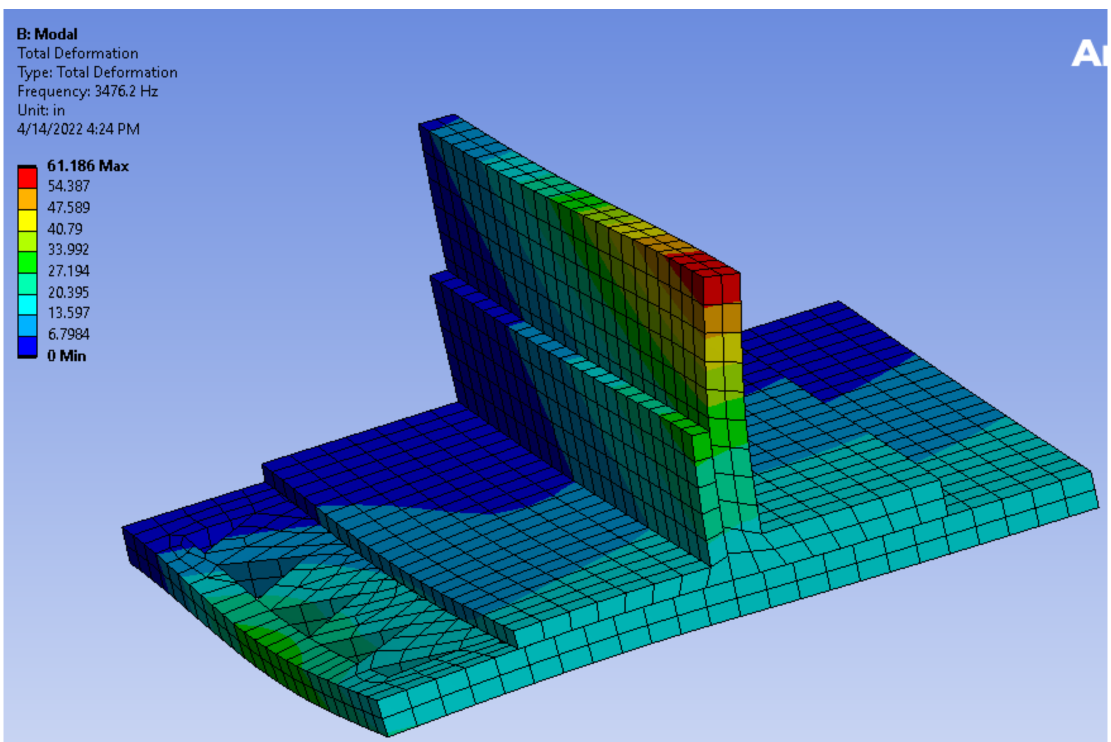


*Figure 51: Modal Analysis - Third Frequency, Mesh Size 0.21, and Linear Element Type.*

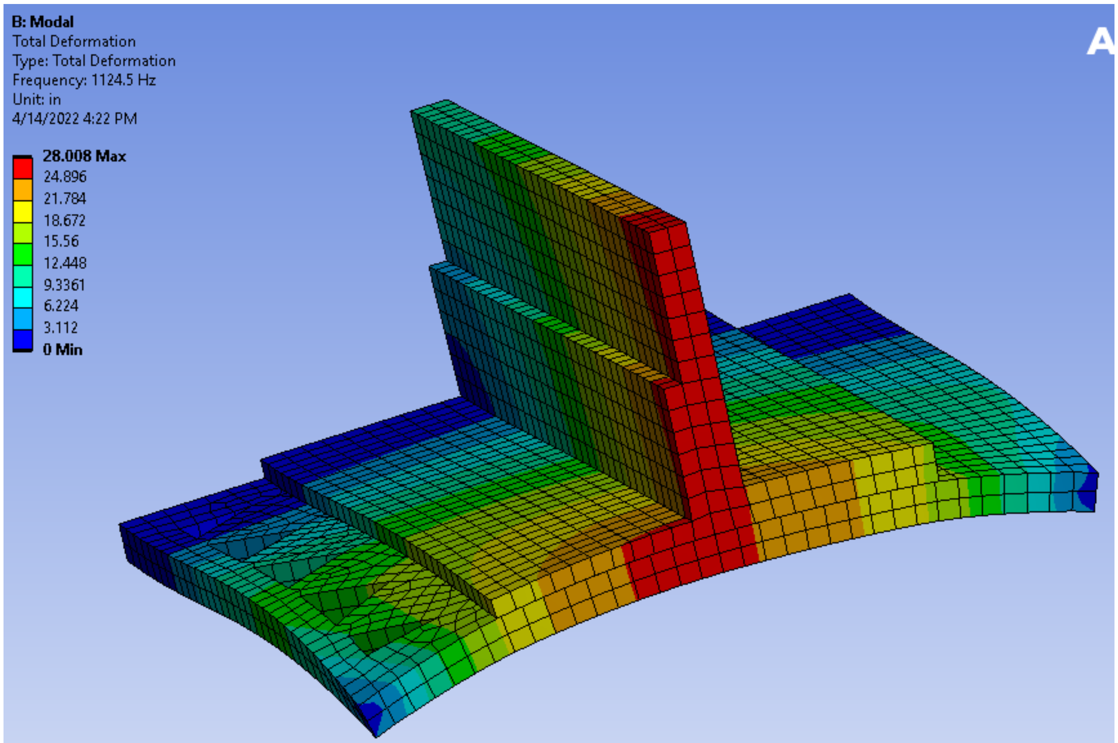


*Figure 52: Modal Analysis - First Frequency, Mesh Size 0.15, and Linear Element Type.*

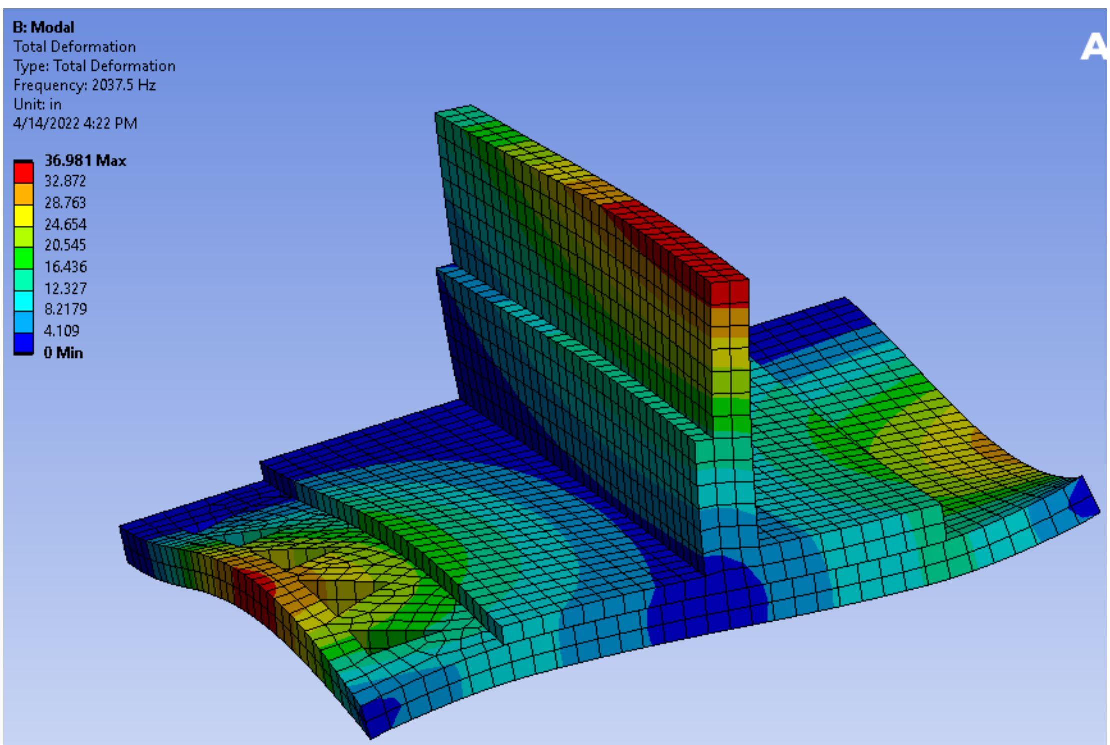


*Figure 53: Modal Analysis - Second Frequency, Mesh Size 0.15, and Linear Element Type.*

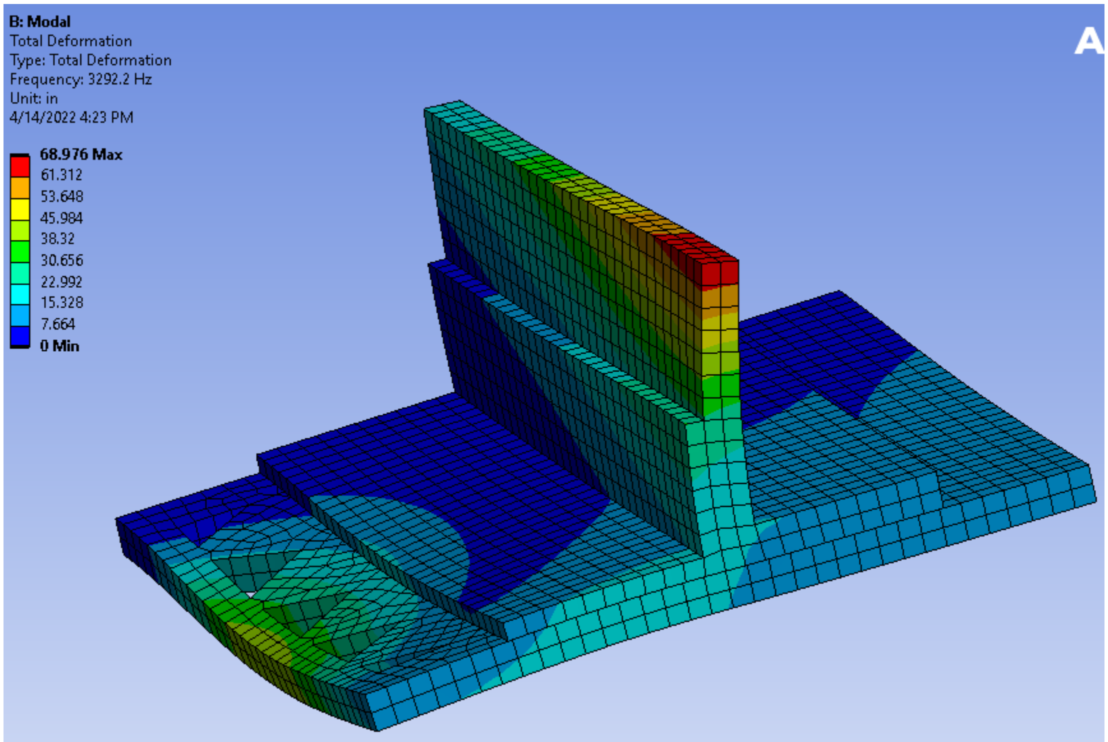


*Figure 54: Modal Analysis - Third Frequency, Mesh Size 0.15, and Linear Element Type.*

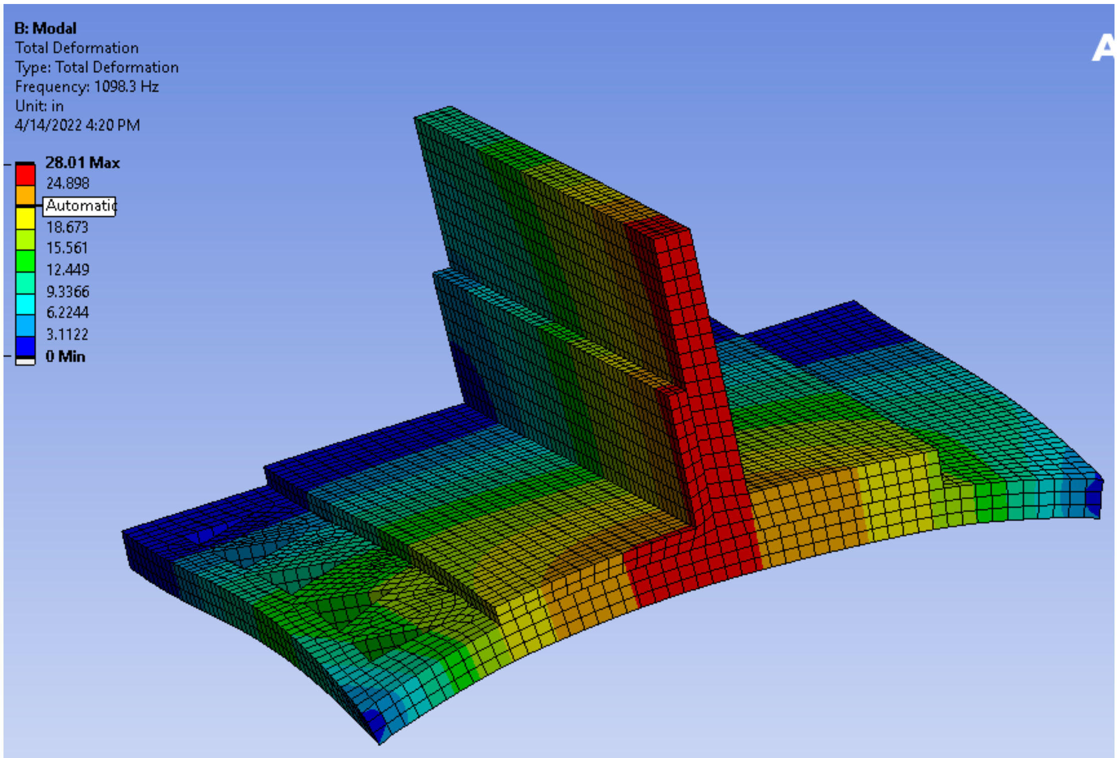


*Figure 55: Modal Analysis - First Frequency, Mesh Size 0.1, and Linear Element Type.*

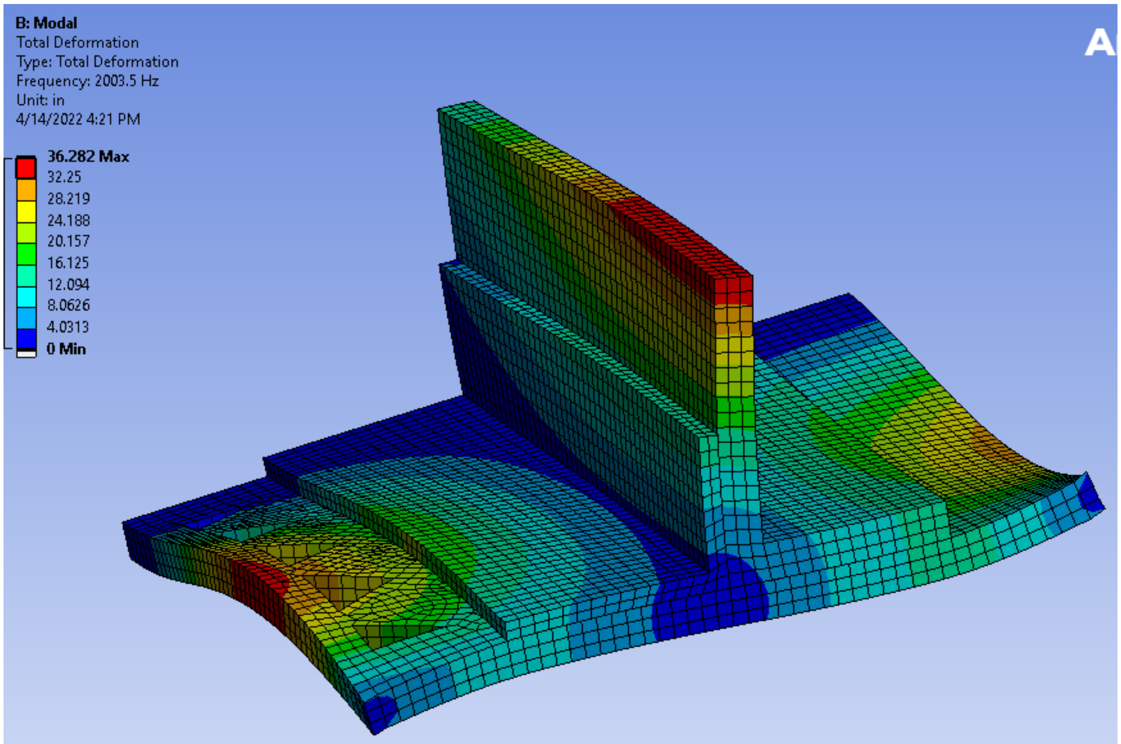


*Figure 56: Modal Analysis - Second Frequency, Mesh Size 0.1, and Linear Element Type.*

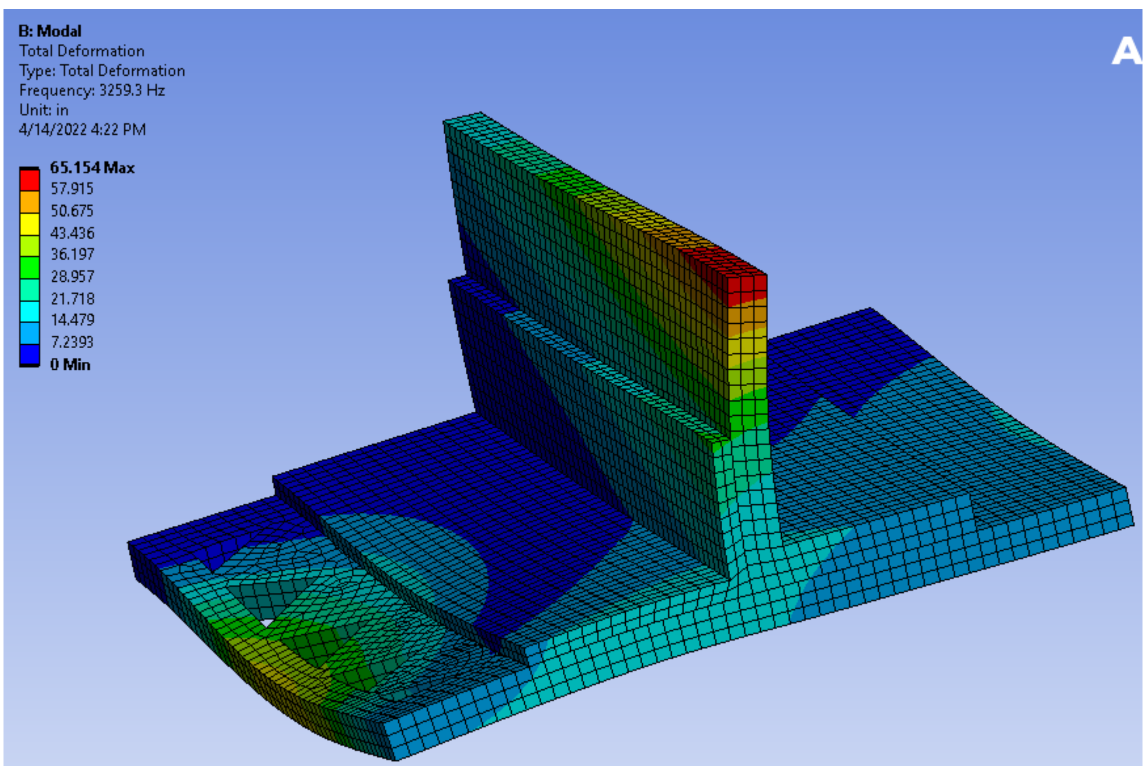


*Figure 57: Modal Analysis - Third Frequency, Mesh Size 0.1, and Linear Element Type.*

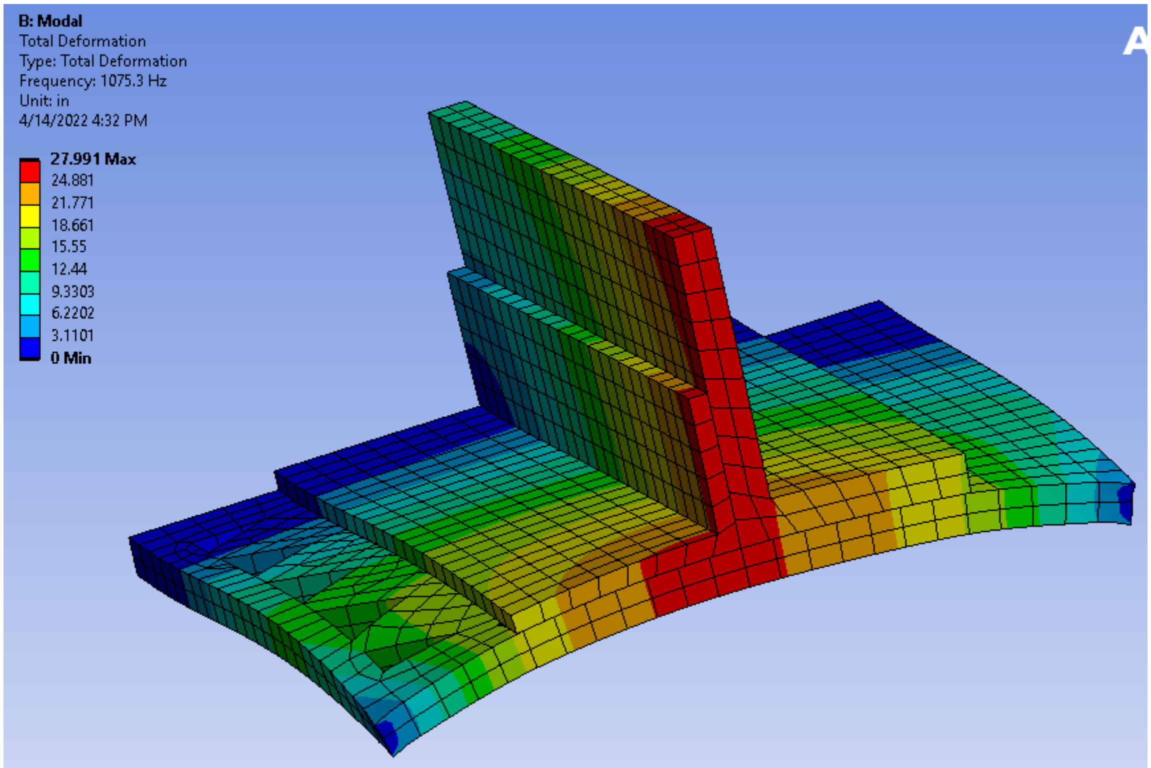

*Figure 58: Modal Analysis - First Frequency, Mesh Size 0.21, and Quadratic Element Type.*

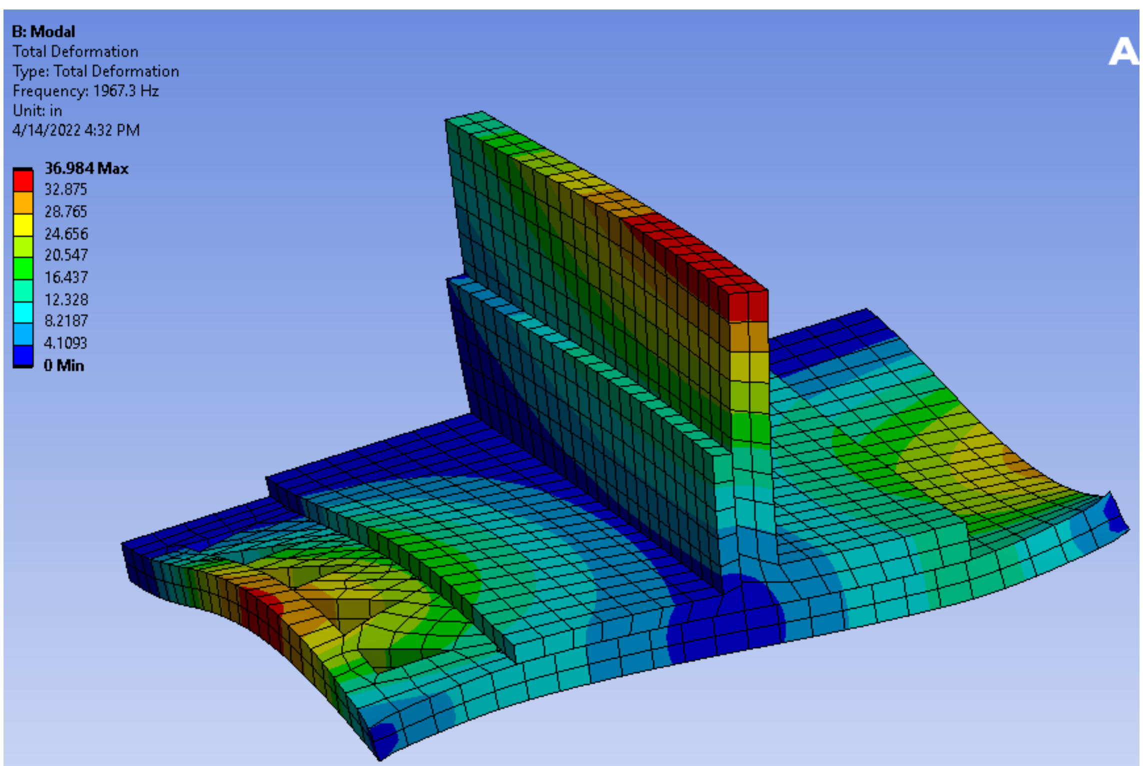

*Figure 59: Modal Analysis - Second Frequency, Mesh Size 0.21, and Quadratic Element Type.*

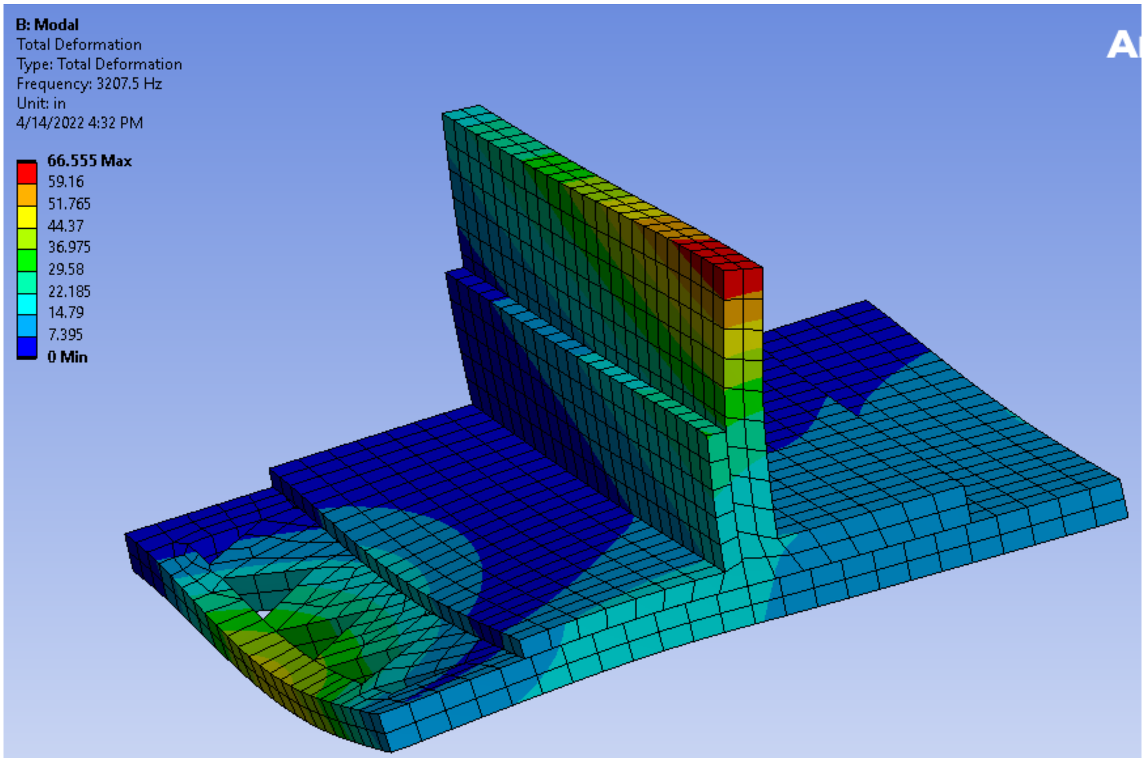


*Figure 60: Modal Analysis - Third Frequency, Mesh Size 0.21, and Quadratic Element Type.*

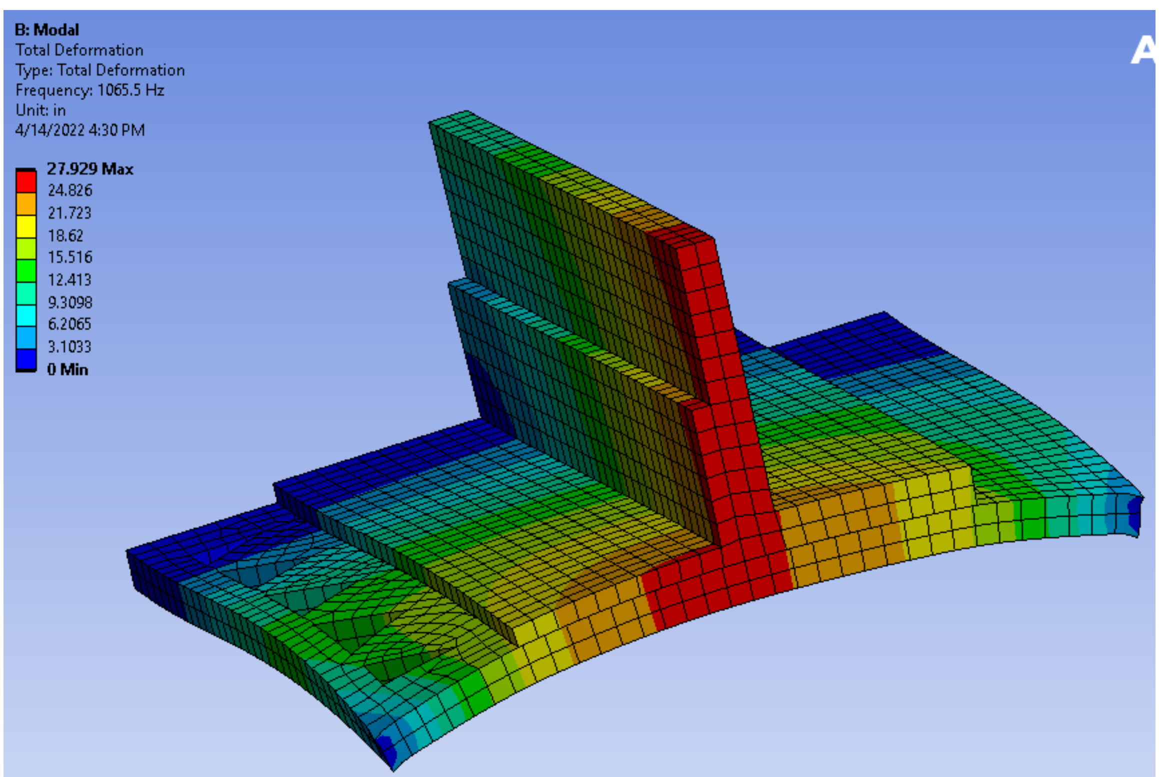


*Figure 61: Modal Analysis - First Frequency, Mesh Size 0.15, and Quadratic Element Type.*

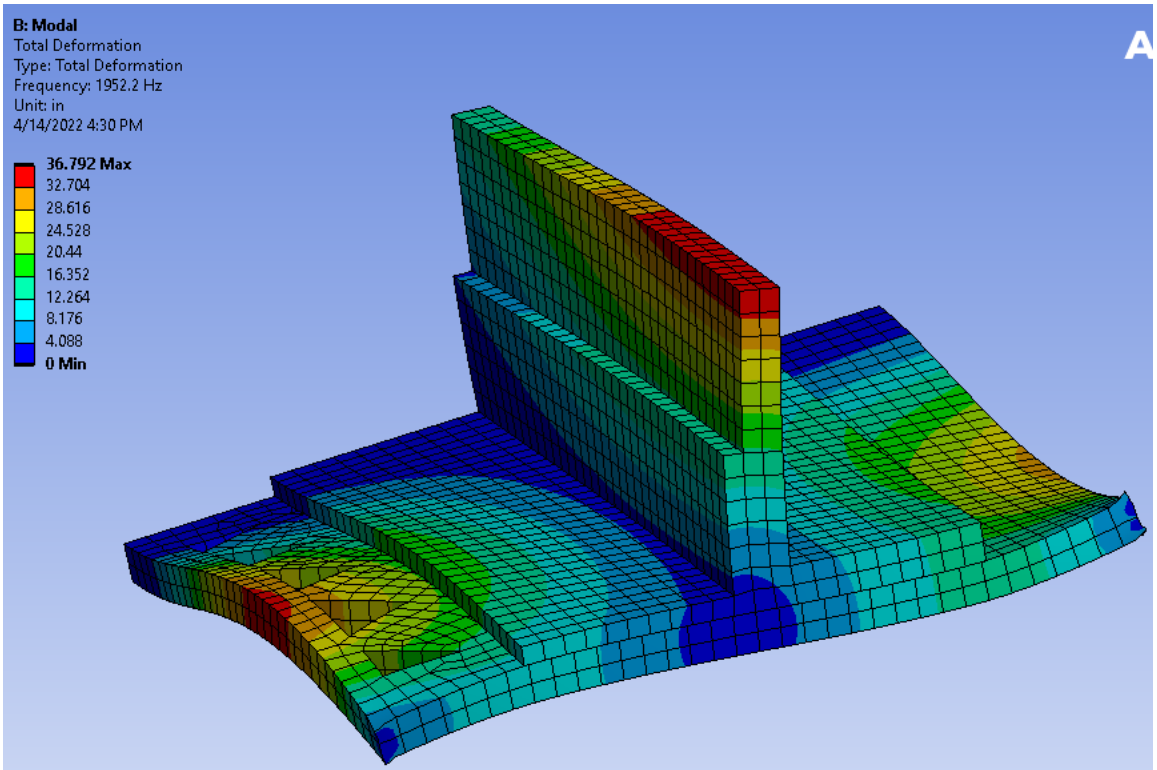

*Figure 62: Modal Analysis - Second Frequency, Mesh Size 0.15, and Quadratic Element Type.*

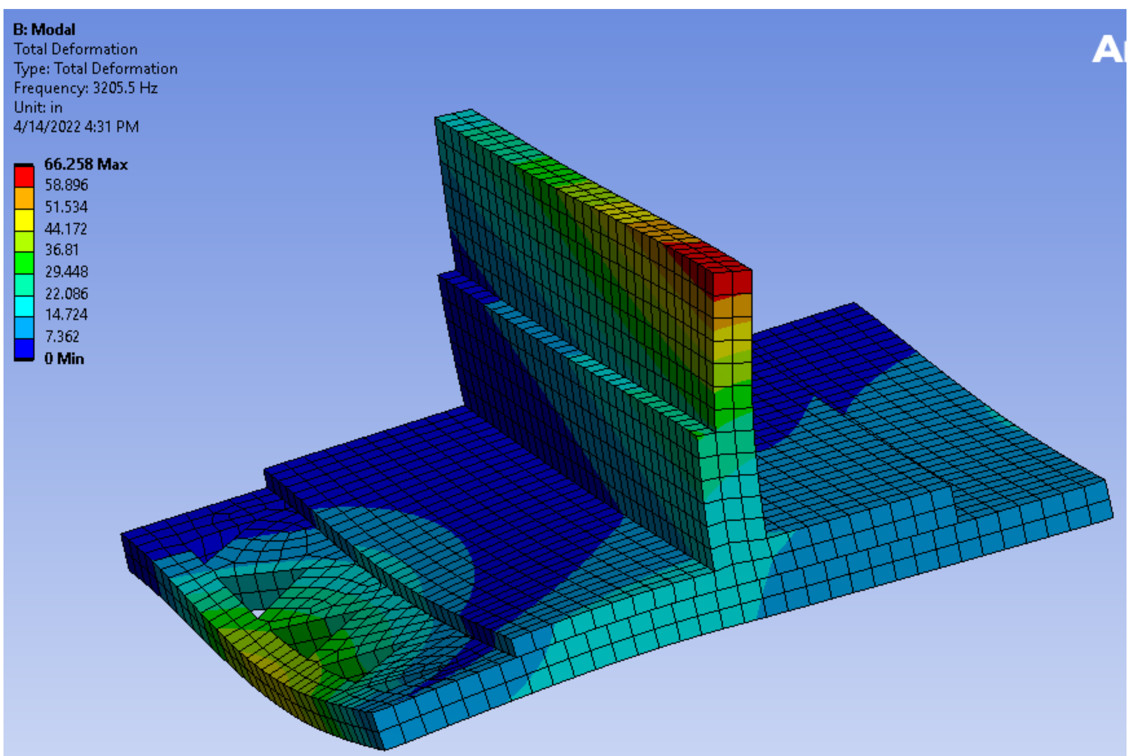

*Figure 63: Modal Analysis - Third Frequency, Mesh Size 0.15, and Quadratic Element Type.*

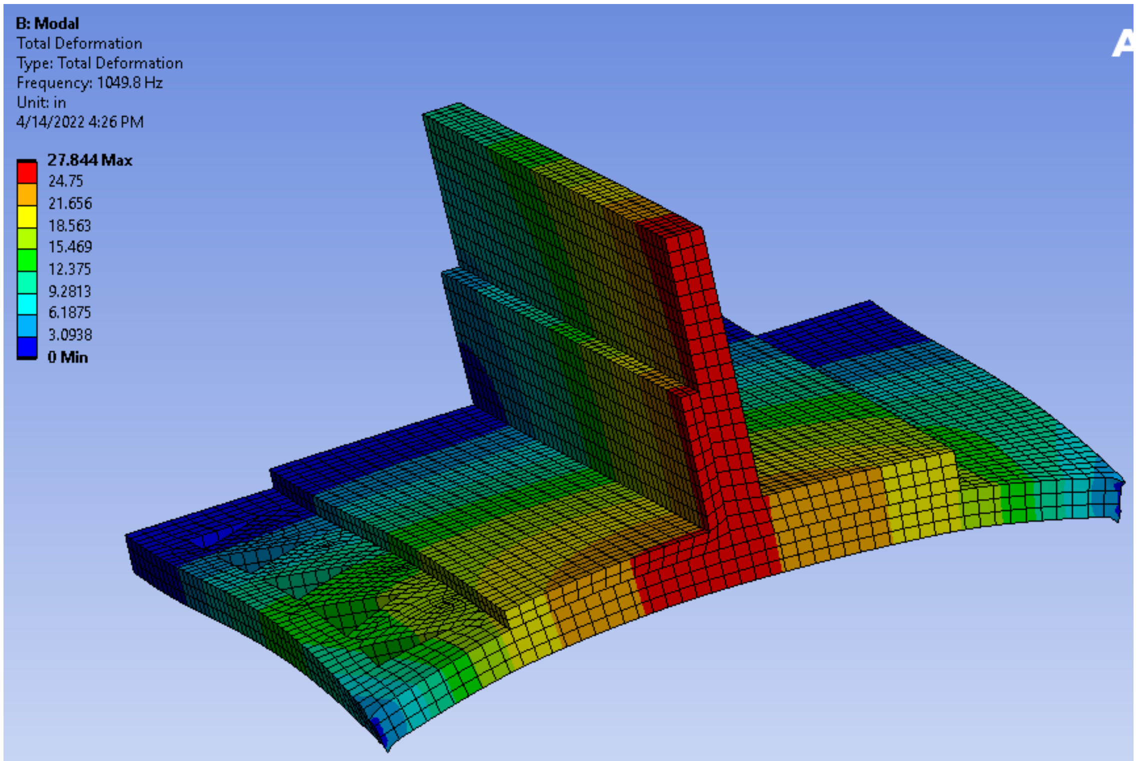


*Figure 64: Modal Analysis - First Frequency, Mesh Size 0.1, and Quadratic Element Type.*

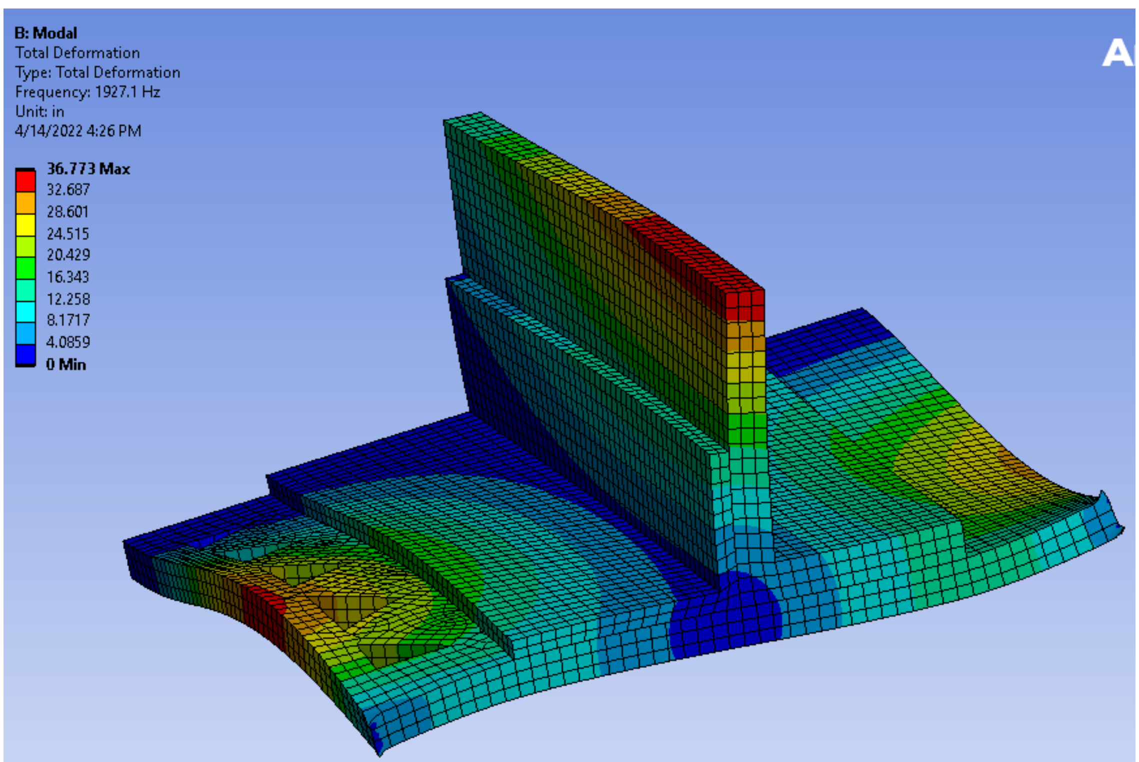


*Figure 65: Modal Analysis - Second Frequency, Mesh Size 0.1, and Quadratic Element Type.*

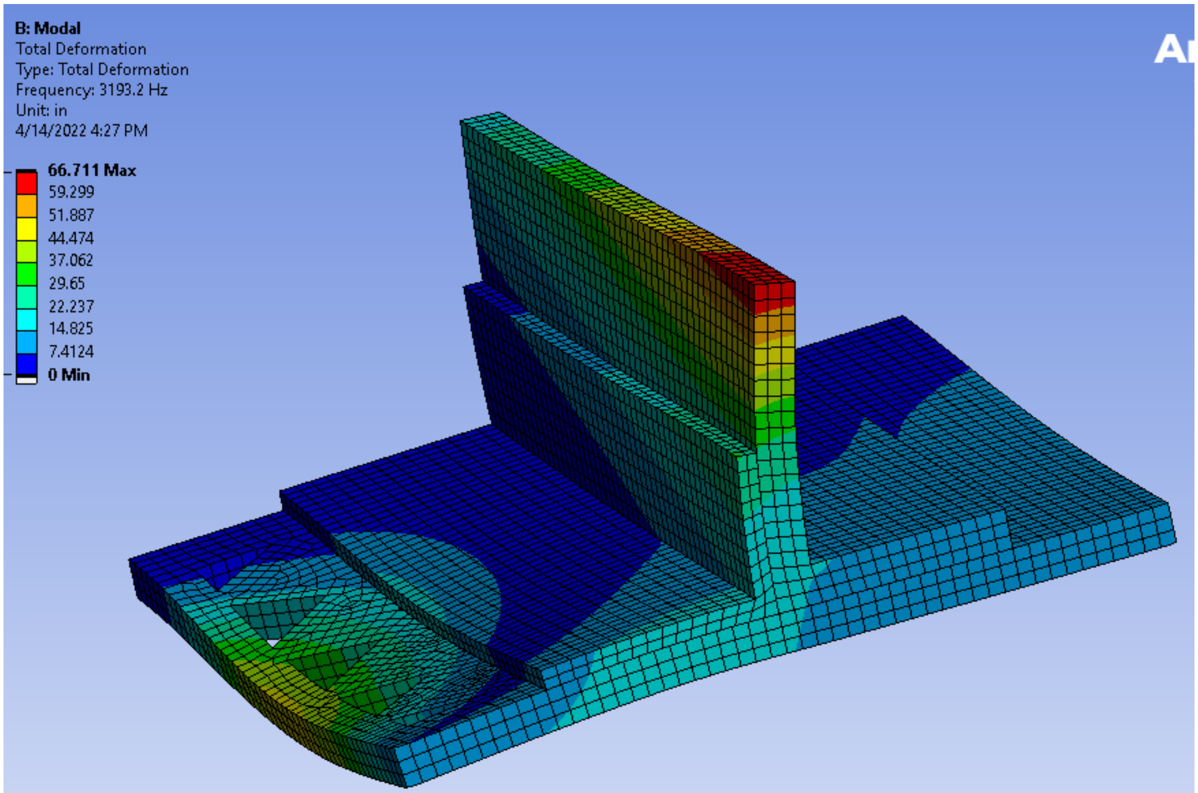


*Figure 66: Modal Analysis - Third Frequency, Mesh Size 0.1, and Quadratic Element Type.*

*Table 4: Comparing the First Three Natural Frequencies.*

| Mesh Size and Element Type | First Frequency | Second Frequency | Third Frequency |
|---|---|---|---|
| 0.21 and Linear Element | 27.65 | 34.51 | 61.18 |
| 0.15 and Linear Element | 28.00 | 36.98 | 68.97 |
| 0.1 and Linear Element | 28.00 | 36.28 | 65.15 |
| 0.21 and Quadratic Element | 28.00 | 36.98 | 66.55 |
| 0.15 and Quadratic Element | 27.95 | 36.79 | 66.26 |
| 0.1 and Quadratic Element | 27.84 | 36.77 | 66.72 |

The above table (Table 4) tabulates the maximum displacements form each natural frequency for each mesh size and element type. The first, second and third frequency displacements were recorded to be consistent for the given mesh sizes and element types.

## Conclusion

For the final project, we were required to conduct a static structural analysis and a modal analysis for the given aircraft structural component. Each analysis was performed for two structures with different boundary conditions. For each structure, two element types were used to compare the results of static and modal analysis. The first element type was the linear solid and the second element type was the quadratic solid. A convergence test is also done along side by comparing three mesh sizes for each structure and element type. It was recorded that the quadratic solid element gave more accurate results for deformation and stress compared to the actual theoretical value. The stress and deformation graphs were recorded for the static structural analysis of each structure. The figures are included in the results section above. For the static analysis of both structures and element types, it was recorded that the smaller mesh sizes gave the highest stress and deformation values. This was expected as a part of the convergence test before doing the analysis because a more refined mesh gives a more accurate result for any analysis. The maximum stress and deformation were recorded to occur at the point of maximum load applied for both structures. For the modal analysis the first three natural frequencies are compared for each structure and element type. It was recorded that refining the mesh size did not have a very significant influence on the natural frequency deformation as it was clear for the static analysis. The deformations are compared in tables 2 and 4. In conclusion, the task was successfully completed by applying the knowledge of static analysis and modal analysis learned throughout the course. Through the course of this project a designing software (CATIA V5) and an engineering simulation software (ANSYS Workbench) were used in tandem to design CAD the given model and conduct the required analysis for the structure.