AE4630 Abaqus Project

By Chad Wrege and Alex Jarret

12/8/2022

Table of Contents

Abstract

In this study, Abaqus was used to create a model of a fuselage and analyze its behavior under various conditions. The problem being addressed was the need to optimize the fuselage for weight while ensuring that it meets requirements for deflection and stress. Abaqus was chosen for its range of features and capabilities for modeling and simulating complex structures. The modeling process included creating the geometry, defining material properties, and specifying external loads and constraints. The model was optimized for weight and the 3 lowest natural frequencies were found. The emergency impact scenario was simulated, and the FEA equations were solved using solvers in Abaqus. Assumptions and simplifications were made, and challenges and difficulties were addressed. The results showed improvements to the model and provided data on natural frequencies and behavior under the emergency impact scenario. The use of Abaqus was successful and provided valuable insights.

Introduction

The aim of this study was to optimize the design of a fuselage section for an aircraft that can meet specified requirements for load-bearing capacity while under material constraints with a 1.5 safety factor and minimizing weight. The fuselage section was designed to accommodate a 3 by 2 seating arrangement and was subjected to a range of loads including distributed loads, representing passengers and luggage, pressurization loads, torsional loads, and a surface traction representing the elevator load. The design of the floor beam and struts, as well as the fuselage panel, were optimized to ensure that the structure could withstand these loads. This paper presents both the design process and the final optimized design.

Design Parameters

The analyzed fuselage must accommodate a 3 by 2 seating arrangement, with 6 windows of the dimensions detailed below.

w	h	r
228.07	192.98	37.112

Due to the material limitations of AL-2024-T4 alloy with the following material properties: $E = 72\,GPa$, $v = 0.335$, and $\sigma_y = 320\,MPa$. With a 1.5 safety factor, the maximum stress allowed is 213.33 MPa. Maximum deflections are described by $\delta = \mathrm{L}/50 = 117.86$ mm.

The fuselage must be 5892.8 mm long with a diameter of 2946.4 mm and accommodate 8 rows of seats. Each seat must withstand a 2000 N load, each row a 8000 N load, and the back edge of the fuselage must be loaded with 80,000 N to simulate elevator downforce. A torsional load of 7366 N/m will be applied to the fuselage.

The analysis will take place at an altitude of 35,000 ft with an internal cabin pressure at 6,000 ft. The seats are 19 inches wide; the aisle is 21 inches wide, and the seat pitch is 30 inches. Lastly, the frame spacing on the fuselage may be no more than 0.5 m. This fuselage will be tested against a simulated 5 m/s ground impact.

Hand Calculations

The numeric hand calculations utilized MATLAB to ensure consistent calculations and to allow for optimization scripts. Using the given design parameters as inputs, rough optimizations would be done to estimate upper thicknesses of all the components.

Beam bending on the hollow shell was used to calculate an initial thickness of 294.64 mm, this result remains the same within 3 decimal points when using distributed vs point seat loads. The torsional load of 7366 N/m resulted in an initial thickness of 2.532×10^{-3} mm. Lastly, using the hoop stress equations and the pressure parameters, an initial thickness of 0.344182 was found. As conservative values, the largest value of 294.64 mm skin will be used as an initial condition for optimizing skin thickness.

The floor supports were designed to optimize stress distribution. As such the supports were mounted at the end of each seat row, at 1447.8 mm and 965.2 mm measured from the respective fuselage wall, and at equal arc lengths along the bottom of the fuselage.

Equations

Uniform Loading

$$EI_{yy}u,_{zzzz} = p_x$$

$$EI_{yy}u,_{zzzz} = p_x z + C_1$$

$$EI_{yy}u,_{zzzz} = \frac{p_x z^2}{2} + C_1 z + C_2$$

$$EI_{yy}u,_{zzzz} = \frac{p_x z^3}{6} + \frac{C_1 z^2}{2} + C_2 z + C_3$$

$$EI_{yy}u,_{zzzz} = \frac{p_x z^4}{24} + \frac{C_1 z^3}{6} + \frac{C_2 z^2}{2} + C_3 z + C_4$$

Torsion

$$q = \frac{T}{2A}$$

$$\tau_{sz} \cdot t = q$$

Shear Cell

$$T = \sum T_i = 2\sum_{i=1}^{n} q_i A_i$$

$$\theta_1 = \theta_2 = \cdots = \theta_n$$

$$\theta = \frac{T}{GJ}$$

At junctions

$$\sum q_i = 0$$

$$q_{web} = q_{i+1} - q_i$$

Modeling Process

Using the conservative values from the hand calculations, the geometry for this model was created using a combination of shell extrusions, shell planes, and stringers. The skin was created using a shell extrusion and was given ample thickness, through the application of a section, to be optimized later. The bulkheads were created using a planer shell that was similarly given ample thickness, through the application of a section, to be optimized later. Stringers were created and were defined through the application of a section to create a circular beam with a thickness that was defined and optimized at a later point.

As outlined in the design parameters documentation, the fuselage would be made from AL-2024-T4 alloy. This gives the following material properties: $E = 72\,GPa$, $v = 0.335$, and $\sigma_y = 320\,MPa$. These values were applied to the entire model, with a 1.5 safety factor, using the material properties module of Abaqus.

Bulkheads

Bulkheads were created using an arbitrary planer shell with a width of 200mm. This was chosen because it seemed to fit with real life examples of bulkheads being used in planes of the same size. The bulkheads were then created with an outer diameter the same as the skin, with a width of 200mm.

Skin & Floor

The skin and floor were created using the same shell extrusion to add simplicity to our model. While this imposes a limitation on the actual design of the floor supports, this was the only way the floor supports could be easily and manageable added to the construction of the fuselage. The floor supports were created to maximize stress distribution as detailed in the hand calculation.

14 bulkheads were used in this model to fulfill the given design criteria of bulkhead spacing being no more than 0.5 m.

Windows

The windows were created in the fuselage model by using an extruded cut created from an offset datum plane. The windows were created so that they were evenly spaced apart and met the required design parameters. It was found through the stress analysis of our model that having the windows placed too high up created very high stress points at the corners of the windows even with very high material thickness. Therefore, the windows were added at a relatively low height.

Optimization Process

The following process was taken to optimize the structure to be as light as possible while maintaining the 213 MPa stress and 117 mm deflection limitations:

V1-V23*	N/A …
V24	Corrected Boundary Conditions
V25	Skin Optimization
V26	…
V27	…
V28	Bulkhead Optimization
V29	…
V30	…
V31	Floor Optimization
V32	… and Stringer Optimization
V33	…
V34	…
V35	…
V36	…
V37	3 Seat Support Optimization
V38	…
V39	…
V40	2 Seat Support Optimization
V41	…
V42	Stringer Optimization
V43	…
V44	Final Model

*A note on V1-V23, due to grossly incorrect boundary conditions these versions are rendered null. The back edge load was defined as a volumetrically based body load. This caused high stress points in the floor beams due to pinching. The stresses and deflections in V1-V23 were drastically overestimated compared to V44.

Skin

The optimization process of our model began with a focus on the areas where most of the mass and stiffness were located. The skin was the first section to be optimized, as it had the largest amount of mass and could be reduced in thickness without significantly impacting the torsional rigidity of the structure. This was also necessary because the skin is responsible for transferring a significant portion of the pressure loads to the bulkheads through the stringers.

Bulkheads

The bulkheads were the next section to be optimized, as they were the second heaviest part of the model and were a major contributor to its ability to withstand the required pressure.

Floor & Floor Supports

The floor and floor supports were the next sections to be optimized, using the same logic. The placement of the floor supports was determined by hand calculations to maximize force distribution. To this end the fuselage mounted edges of the floor supports were spaced at equal arc lengths and the floor mounted edges were placed at the end of each seat row.

Stringers

The stringers were the final section to be optimized, as they are critical for maintaining the stress and deflection levels of the model. The optimization process concluded with version 44 of the model.

Frequency Analysis

The natural frequency of our fuselage was found by performing a modal analysis. This analysis involves solving the eigenvalue problem for the structure to determine the frequency at which it will vibrate without external excitation. This analysis was done with no loads applied to the structure. The eigenvalue solution was then computed using the finite element method, and the natural frequencies and corresponding mode shapes of the structure were obtained. These results could then be used to evaluate the stability and dynamic response of the structure under various loading conditions.

Buckling Analysis

The buckling analysis of our fuselage was found via nonlinear analysis that was performed by Abaqus using the finite element method to solve for the critical buckling loads and corresponding mode shapes of the structure. The results of the buckling analysis can be visualized and post-processed to evaluate the stability and deformations of the structure under different loading conditions.

Impact Scenario

An emergency impact scenario was then applied to our fuselage as if it were impacting the ground at 5 m/s. An assumption about the impact was that the impact was evenly distributed over the front of the fuselage. The impact was also assumed to take place over a period of one second. The impact was then assumed to be F=ma where m is the mass of the fuselage which was 2315 Kg and a is the acceleration that the fuselage experiences at impact which was 5 m/s^2. This was done by applying a concentrated force of 11580 N to a reference point that was kinematically coupled to the outside edge of the front of the fuselage.

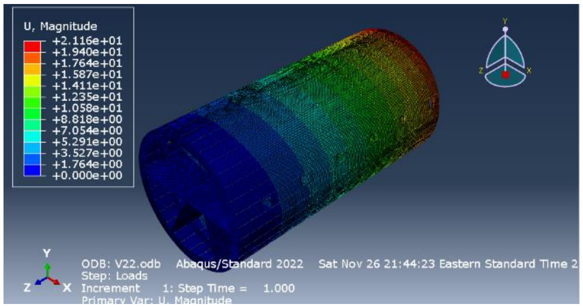

Figure 1: V22 Deformation Expanded View

Results

Starting with V22, the initial model failed neither stress nor displacement; however, it was extremely heavy at 7.67×10^9 units. This was to be expected due to the initial conditions being pre-optimized by the hand calculations. V22 can be viewed in Figure 2: V22 Displacement and Figure 3: V22 Stress.

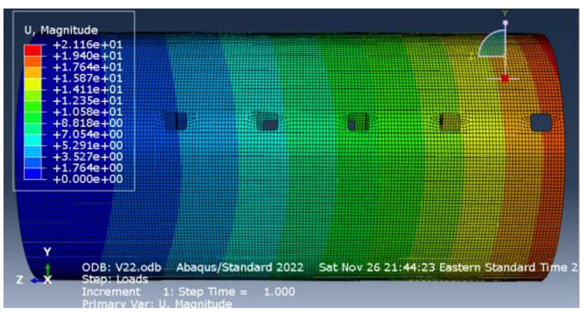

Figure 2: V22 Displacement

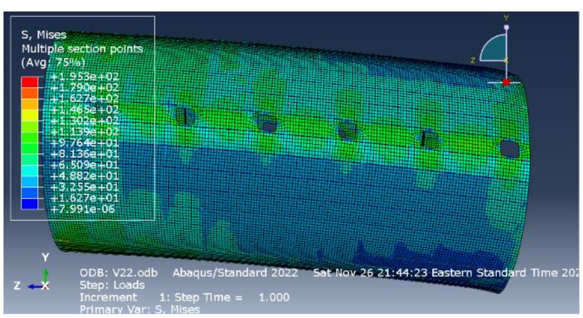

Figure 3: V22 Stress

While this model meets the design criteria, further weight optimization is possible. Strategic optimizations of the fuselage would begin from components that contributed the most weight to the least.

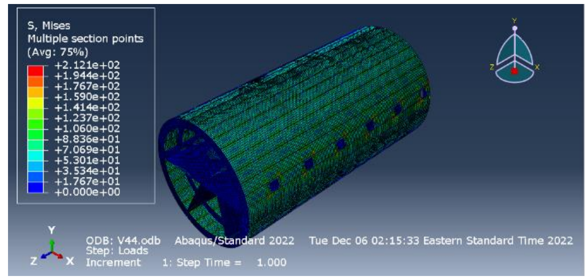

Figure 4: V22 Stress Expanded View

Ver	Disp (mm)	Stress (MPa)	Weight (Units)
V22	21.16	195.3	7.67E+09
V23	20.61	133.4	7.67E+09
V24	20.91	134.1	4.37E+09
V25	3.86	101.41	4.10E+09
V26	2.51	83.7	3.94E+09
V27	5	135.9	3.88E+09
V28	5.52	136.8	3.66E+09
V29	6.056	138.9	3.55E+09
V30	7.027	138.6	3.43E+09
V31	14.07	219.4	2.63E+09
V32	11.97	193.5	2.78E+09
V33	12.15	197.8	2.09E+09
V34	12.3	199.5	1.74E+09
V35	12.45	199.4	1.57E+09
V36	64.7	249	1.48E+09
V37	12.58	199.4	1.38E+09
V38	12.66	199.2	1.30E+09
V39	12.72	199.1	1.26E+09
V40	12.02	200	1.00E+09
V41	14.29	201	9.19E+08
V42	15.54	228.6	7.69E+08
V43	14.63	217.6	8.25E+08
V44	14.46	212.1	8.55E+08

Table 1: Deformation and Stress Results by Version

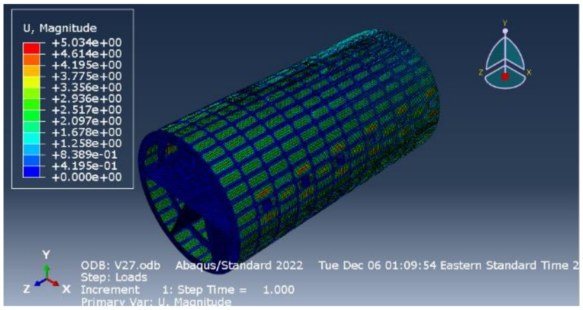

Figure 5: V27 Displacement Expanded View

V27 Skin Optimization

Skin optimization completed. This is readily apparent information from Table 1: Deformation and Stress Results by Version, between V26 and V27 there is a significant increase in stress and deformation.

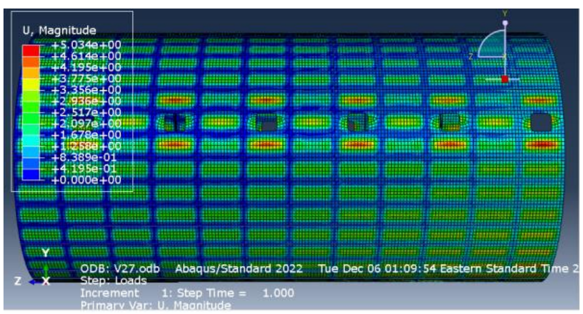

Figure 6: V27 Displacement

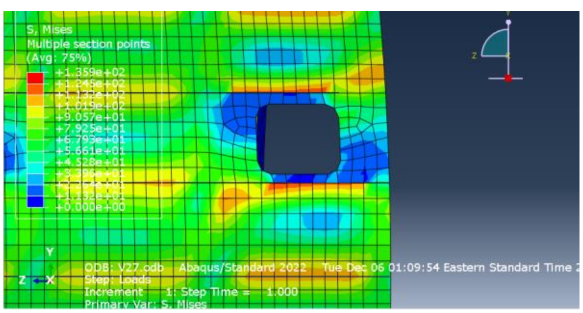

Figure 7: V27 Stress

Figure 9: Stress per Version details the change from V26 to V27, with the decrease in skin thickness, there is a significant jump in maximum stress.

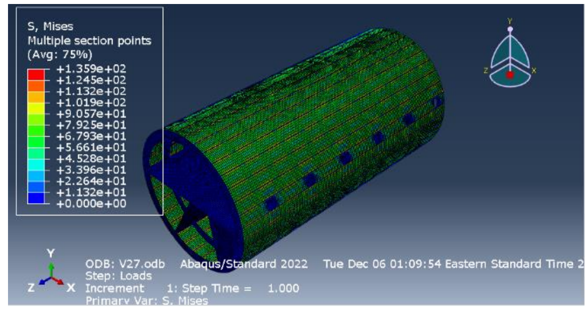

Figure 8: V27 Stress Expanded View

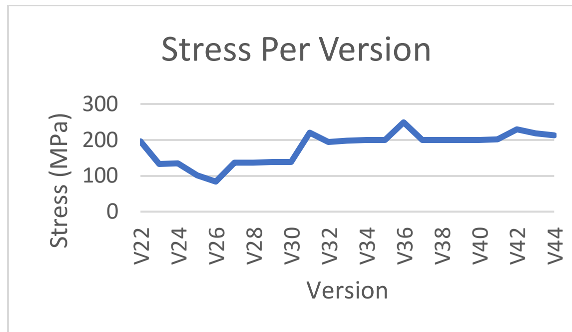

Figure 9: Stress per Version

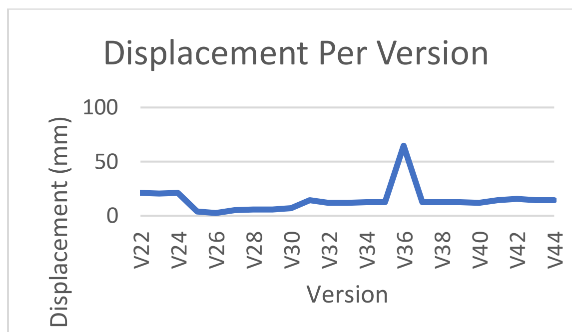

Figure 10: Displacement per Version

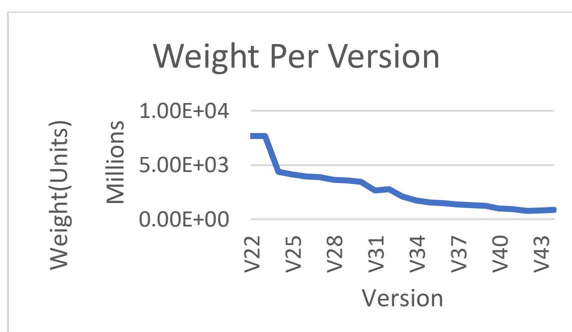

Figure 11: Weight per Version

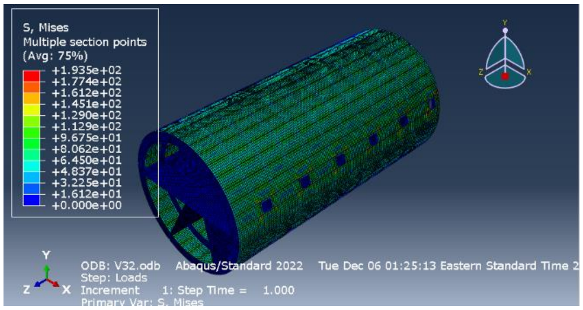

Figure 12: V32 Stress Expanded View

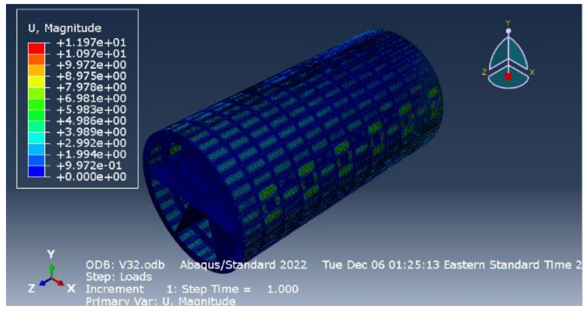

Figure 15: V32 Displacement Expanded View

V32 First Stringers Optimization

It is clear from Figure 14: V32 Stress that stress builds up around the corners of windows. Lowering the windows was a major part in reducing said stress. This optimization contained the first stringer optimization. Due to the need of further floor optimizations, this had little effect on the overall model.

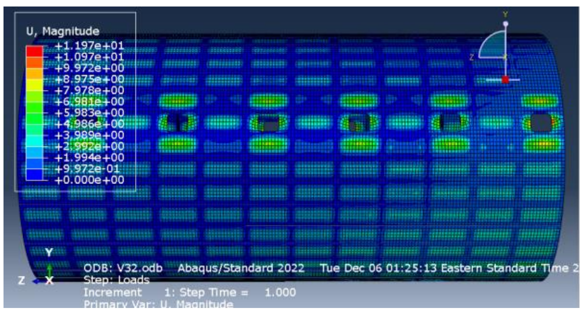

Figure 13: V32 Displacement

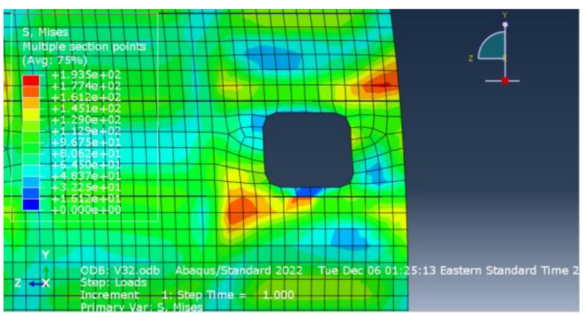

Figure 14: V32 Stress

V36 Displacement Spike

As is seen on Figure 10: Displacement per Version there is a displacement spike at V36. This is due to floor thickness being dropped to 5 mm, far below the optimized result of 10 mm. The deformation may be seen below in Figure 16: V36 Deformation.

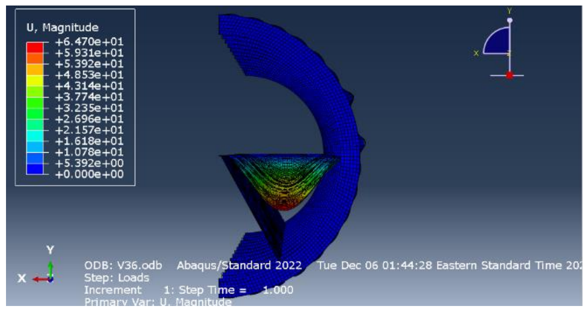

Figure 16: V36 Deformation

V44 Final Model

The final model has been optimized to a weight of 8.55×10^8 with a max displacement of 14.46 mm and a stress of 212.1 MPa. This is an order of magnitude less than the initial fuselage condition.

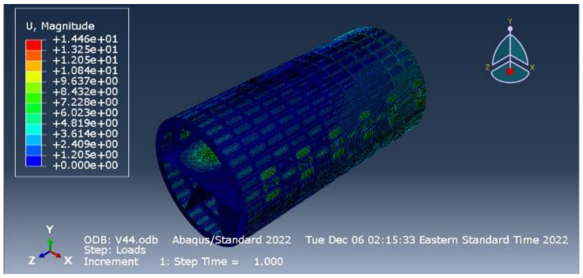

Figure 17: V44 Displacement Expanded View

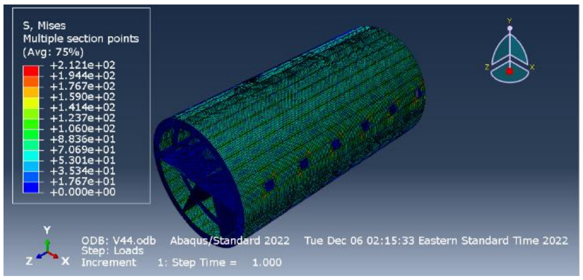

Figure 18: V44 Stress Expanded Veiw

Figure 19: V44 Displacement

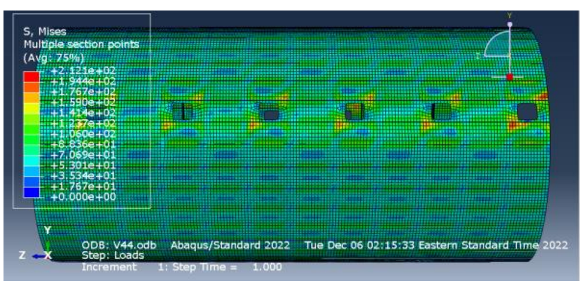

Figure 20: V44 Stress

Frequency Analysis:

The three lowest frequencies were as follows:

Mode	Frequency
1	6.29520E-05
2	7.84025E-05
3	9.15800E-05

Table 2: Frequency Analysis

Visually this may be viewed in the figures on the next page. This describes the frequency at which describe the harmonics at which catastrophic decoherence occur.

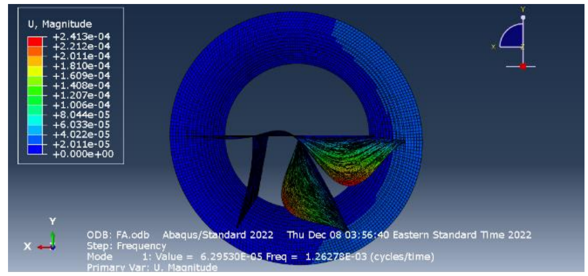

Figure 21: Frequency Mode 1

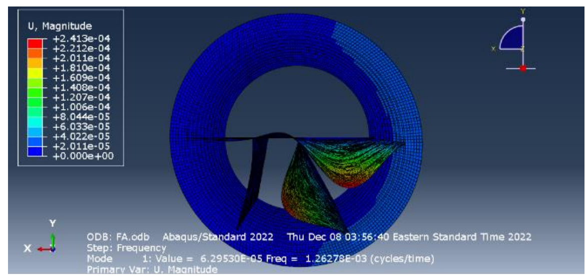

Figure 22: Frequency Mode 2

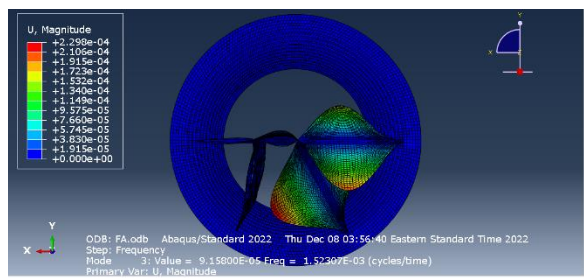

Figure 23: Frequency Mode 3

Buckling Analysis:

The buckling analysis module in Abaqus is a tool that allows users to evaluate the stability of structures under different loading conditions. This is done by solving the eigenvalue problem for the structure to determine the critical buckling loads and corresponding mode shapes of the structure. The buckling analysis module can be used to evaluate the behavior of structures under various types of loading, such as compression, bending, or torsion, and can be used to determine the buckling strength of the structure and the corresponding deformations.

Mode	Eigenvalue
1	-181.74
2	-183.39
3	-185.07
4	-186.52
5	-186.82
6	-188.29
7	-188.39
8	-188.83
9	-190.18
10	-191.16

Table 3: Eigenvalues

The buckling analysis was done with a fuselage that had roughly 10 times the thickness of the optimized fuselage on every section. The buckling analysis would not converge unless this was done, and this was deemed the only solution. The negative magnitudes are relevant as they mean the opposite applied loads create instability in our structure. These opposite loads are entirely possible for our structure. Therefore, it is assumed that there are limitations existing based on our boundary conditions, the way the loads were applied, or the way the geometry of the part was created.

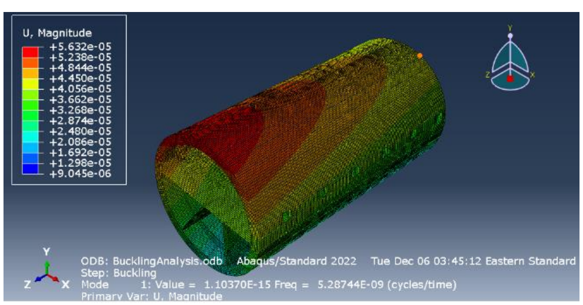

Figure 24: V44 Buckling Analysis

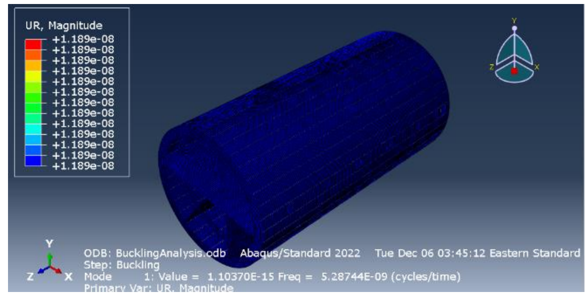

Figure 25: V44 Buckling Analysis UR

Emergency Impact:

5 m/s emergency impact resulted in a maximum displacement value of 11.36 mm and a maximum stress of 196.6 MPa. These results are under our required stress and displacement values including the safety factor. Therefore, there is evidence to conclude that the fuselage would survive an impact directly distributed on the front end of the fuselage. This conclusion has obvious limitations since this analysis only works if the aircraft were to crash falling straight own and if it were to evenly distribute the pressure from this crash, which would not happen in the real world. Therefore, this conclusion must be taken with caution.

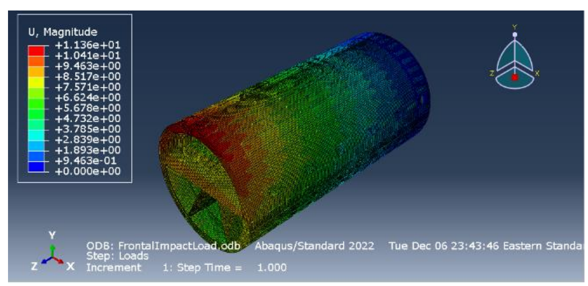

Figure 26: V44 Impact Displacement

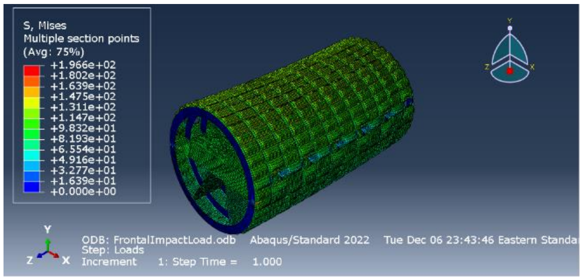

Figure 27: V44 Impact Analysis Stress

Conclusion:

In conclusion, this study aimed to use Abaqus to create a model of a fuselage and analyze its behavior under different loading conditions. The modeling process included the creation of the geometry, specification of material properties, and definition of external loads and constraints. The model was optimized for weight while still meeting the requirements for deflection and stress. The natural frequencies of the model were found, and a buckling analysis was performed. The emergency impact scenario was also simulated. The results of the optimization, natural frequency analysis, buckling analysis, and emergency impact scenario were discussed, and relevant data output was included. Overall, the use of Abaqus proved to be an effective tool for analyzing the behavior of the fuselage under various loading conditions.