Structural Analysis of Chimney Response to Pressure Wave Impacts

Overall Objective

The objective of this study is to determine the maximum total displacement of the chimney structure subjected to a transient analysis following a pressure wave impact. Furthermore, the study aims to assess the potential for structural collapse using a combination of modal analysis, mode superposition harmonic analysis, and time transient structural analysis.

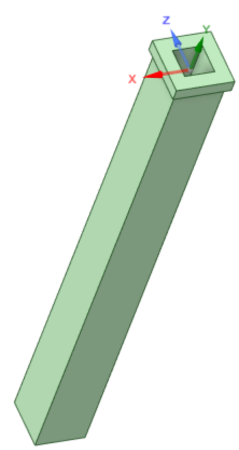
Assumptions

Thermal effects are not considered in this analysis.

It is assumed that minor cracking will occur within the structure, which reflects a structural stiffness damping (beta) of 3% at the dominant response frequency.

Geometry

The figure below illustrates the geometry of the chimney; dimensions are provided in the appendix for reference.

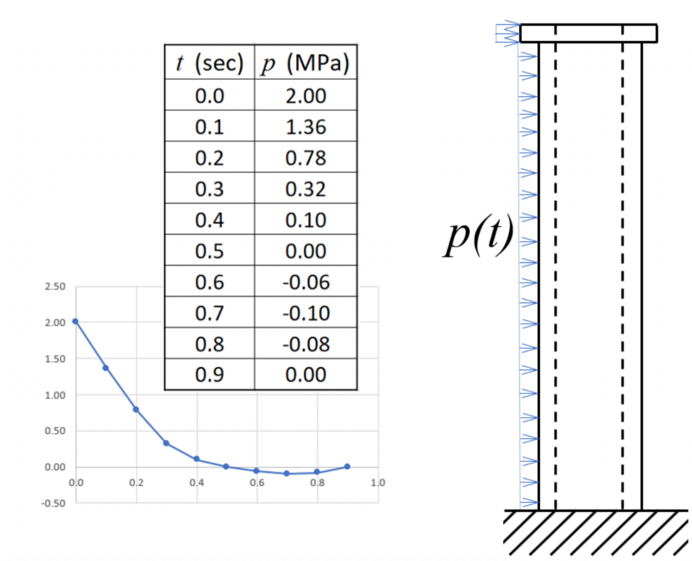


Material Data

Concrete specifications from the Engineering Data tool have been used for this structure, characterized by a modulus of elasticity (E) of 30,000 MPa, a Poisson's ratio of 0.18, and an ultimate tensile strength of 5 MPa.

Boundary Conditions

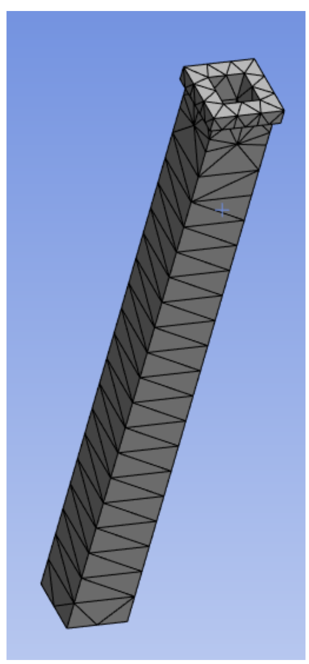
The base of the chimney is fixed. A time-dependent pressure load resulting from an explosion is applied to the two left surfaces of the chimney. The figure below illustrates these loads and their values over time.



t (sec)	p (MPa)
0.0	2.00
0.1	1.36
0.2	0.78
0.3	0.32
0.4	0.10
0.5	0.00
0.6	-0.06
0.7	-0.10
0.8	-0.08
0.9	0.00

Mesh convergence study

Following the guidelines in the problem statement, a default mesh size was utilized for the analysis to optimize time efficiency. Below figure shows the default mesh used in this analysis:



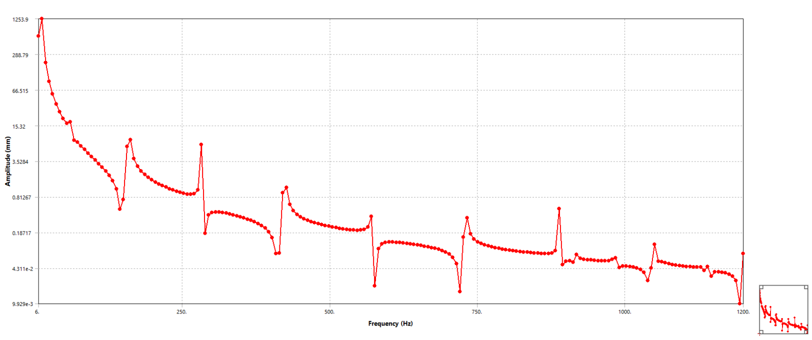
Results

For the modal analysis, 35 modes were used to capture over 90% of the Ratio of Effective Mass to Total Mass for both displacement and rotations across various axes. The table below summarizes the frequencies and deformations for the first 25 mode shapes:

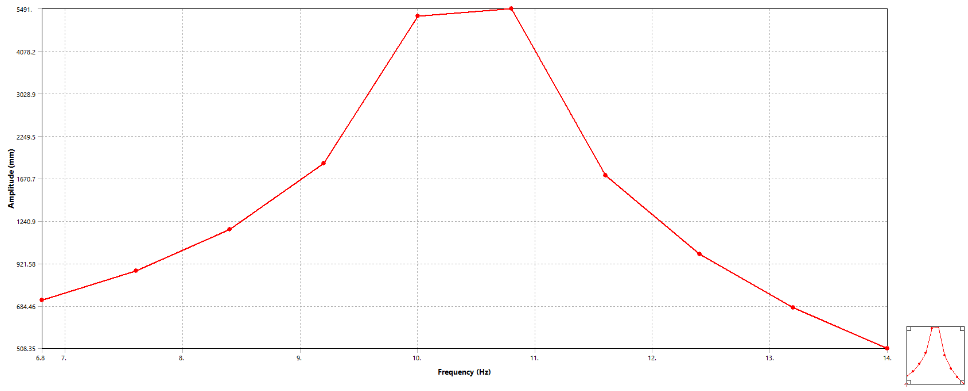
Tabular Data

	Mode	Frequency [Hz]
1	1.	10.418
2	2.	10.419
3	3.	61.512
4	4.	61.534
5	5.	92.934
6	6.	144.98
7	7.	158.73
8	8.	158.85
9	9.	278.27
10	10.	282.34
11	11.	282.59
12	12.	422.32
13	13.	423.
14	14.	434.7
15	15.	462.6
16	16.	572.83
17	17.	573.14
18	18.	646.92
19	19.	713.28
20	20.	723.63
21	21.	728.96
22	22.	729.07
23	23.	832.85
24	24.	887.61
25	25.	888.13

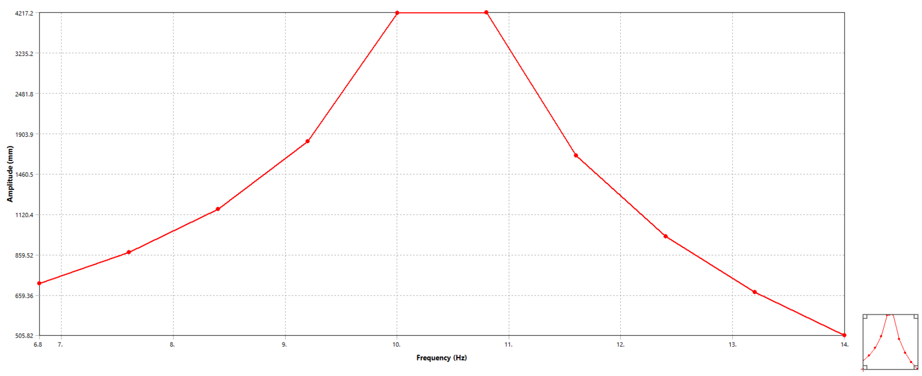
We conducted the harmonic analysis by applying a pressure of 2 MPa. The frequency range was set from a minimum of 0 Hz to a maximum of 1200 Hz (to capture all the first 35 modes), divided into 200 intervals for detailed analysis. Below, the results depict the frequency response deformation from this analysis:



Upon reviewing the table, it is evident that the maximum frequency response deformation occurs around 10.4 Hz. Consequently, we will refine our analysis by focusing on this frequency range and repeating the calculations for greater accuracy.



When incorporating a structural stiffness damping of 3% (beta damping) into the harmonic analysis, the resulting plot is as shown below:



The settings for the damping are detailed in the appendix.

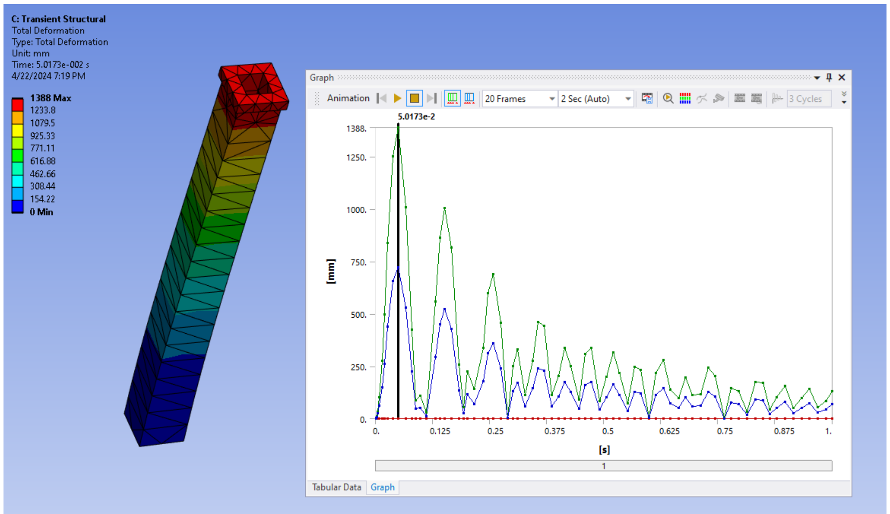
We now proceed with the transient structural analysis, incorporating a structural stiffness (beta) damping of 3%.

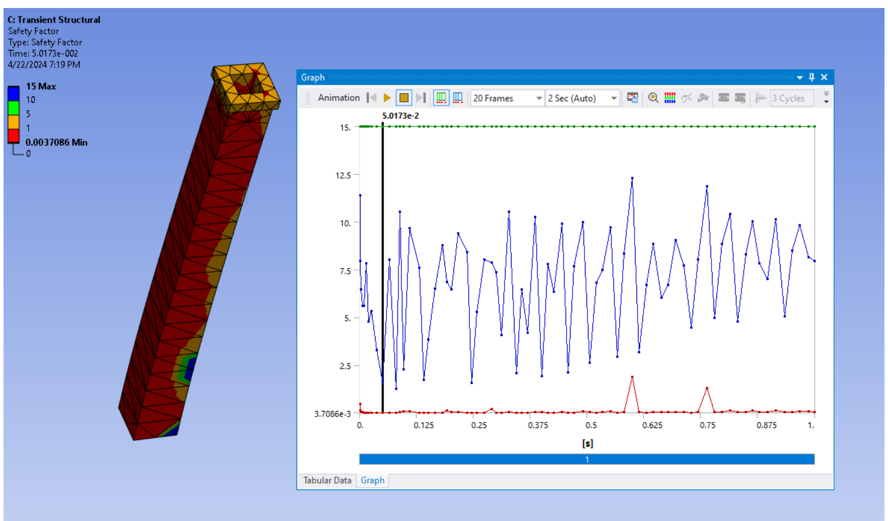
To ensure convergence in the transient analysis, we have recalculated using various time steps. The results, showing maximum total deformation and minimum safety factor for these time steps, are presented in the table below.

Time step (s)	0.01	0.005	0.0025	0.001
Maximum deformation (mm)	1330	1359	1366	1388
Minimum safety factor	0.0038	0.0037	0.0038	0.0037

The table above indicates that convergence is achieved with a time step of 0.001 seconds, as the difference in maximum deformation between time steps of 0.0025 and 0.001 is less than 1.6%.

The results displayed in the figure below were obtained using a minimum time step of 0.001 seconds:





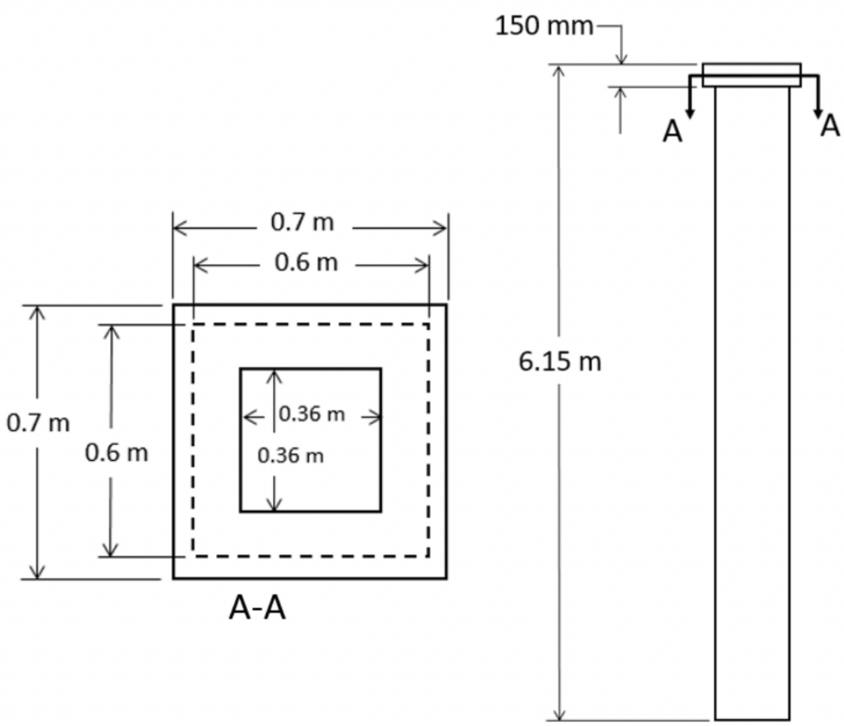
The safety factor analysis using the stress tool indicates that the structure fails under the applied load, as the minimum safety factor is much less than 1.

Conclusion

This study determined the maximum displacement of a chimney structure under a pressure wave and evaluated its likelihood of collapse using advanced analytical techniques. Despite achieving convergence, the low safety factors indicate that the chimney will fail under the simulated conditions. It is recommended to review the design and consider increasing the damping to enhance structural resilience and safety.

Appendix

The figure below displays the dimensions of the chimney as used in this report.



Settings for damping:

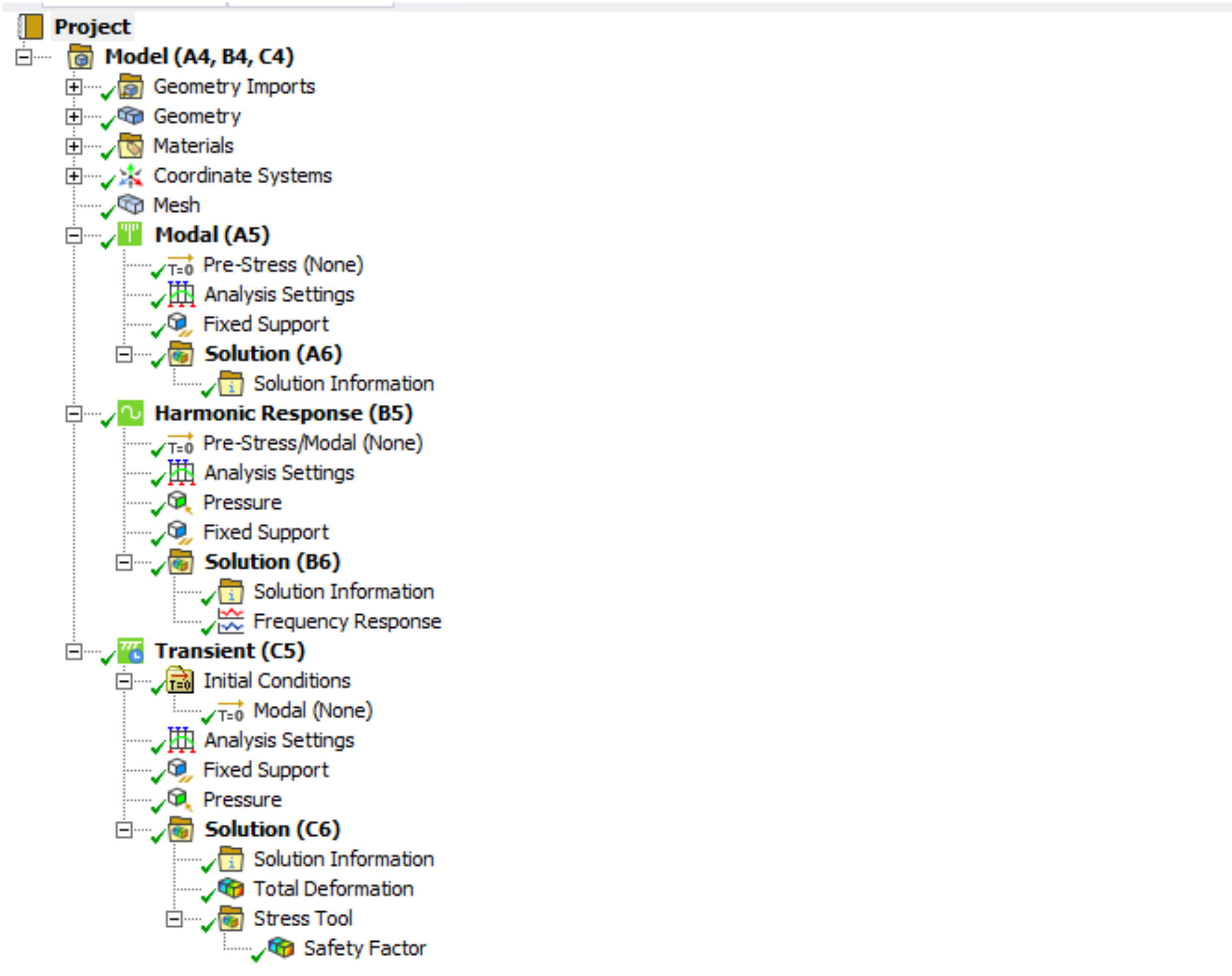
Damping Controls	
Stiffness Coefficient Define By	Damping vs Frequency
-- Frequency	10.418 Hz
-- Damping Ratio	3.e-002
Stiffness Coefficient	9.1662e-004
Mass Coefficient	0.

Settings for stress tool:

Details of "Stress Tool"

Definition	
Theory	Max Tensile Stress
Stress Limit Type	Tensile Ultimate Per Material

Structure of the project in Ansys Mechanical:


Project
Model (A4, B4, C4)
Geometry Imports
Geometry
Materials
Coordinate Systems
Mesh
Modal (A5)
Pre-Stress (None)
Analysis Settings
Fixed Support
Solution (A6)
Solution Information
Harmonic Response (B5)
Pre-Stress/Modal (None)
Analysis Settings
Pressure
Fixed Support
Solution (B6)
Solution Information
Frequency Response
Transient (C5)
Initial Conditions
Modal (None)
Analysis Settings
Fixed Support
Pressure
Solution (C6)
Solution Information
Total Deformation
Stress Tool
Safety Factor