Instructions to Candidates

This is an individual piece of work.

Objective:

The aim of this coursework is for the student to analyse a practical engineering problem using a finite element software and be able to validate the numerical model using other research methods.

Assessed Learning Outcomes:

The expected learning outcomes are as follows:

- Understand the principles of the finite element method
- Set up suitable numerical models
- Validate computer models
- Establish links between analysis and design and enhance understanding of structural design
- Carry out various case studies to gain a deeper understanding of the structural behaviour

Background:

You are approached by a client to design a multi-story residential building. Based on the design brief, you have decided to use steel frame structures. A design proposal has been passed to you to carry out finite element analyses. The client has specified the following numerical analyses:

1. Determine the first six natural frequencies and the modal shapes of the frame structure with/without slabs.

2. Analyse the structural behaviour of the frame with slabs under given static loading.

3. Calculate the load capacity of the worst loaded beam when it is subjected to 500 °C temperature on one side, and ambient temperature on the other side. Justification of the boundary conditions will be necessary.

The design proposal and loading case are given in Table 1 and Fig. 1 (below). You are required to:

1. Set up a suitable numerical model in ANSYS, and justify your choices of the model, element types, mesh size and boundary conditions.
2. Validate your numerical model using the knowledge you have gained in previous years.
3. Carry out the various finite element analyses that the client has requested, and analyse the numerical data.
4. Comment on the models, their advantages and disadvantages to raise awareness of the limitations of the numerical results.

Design Proposals:

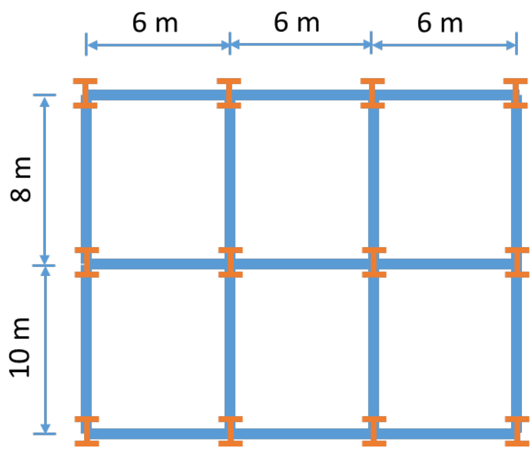


Fig. 1 Proposed floor plan

Table 1 Design parameters

Number of stories (floor height = 3m)	3
Column sizes	UC152X152X23
Beam sizes	UB127X76X13
Steel grade	S275
RC Slab thickness	125 mm
Floor loading	7 kN/m^2

Content

1. Introduction

As cities and communities continue to grow, there is an increasing demand for civil engineering structures. Steel is widely utilized in structures essential for maintaining the normal functioning of society, such as high-rise buildings, large-span structures, and offshore facilities. Steel structures are frequently utilized for several benefits such as their high strength-to-weight ratio, suitability for industrial applications, ease of transportation, and ease of construction (A.U.R. Dogar, 2020).

In this report, the numerical analysis of the 3-storey building is provided. ANSYS software is going to be utilized to set up a numerical model and get results. The basis of a design proposal given to the client is shown in figure1. For this multi-storey building, it is proposed that the steel structure with section properties, slab thickness, and floor height are as illustrated in Table 1, and beams and columns are specified in Fig 1.

The structural behavior of this steel frame is going to be investigated by determining the first six natural frequencies and the modal shapes of the frame structure with and without slabs. The following report also contains the analysis of the structural behavior of the frame with slabs under given static loading of 7 $^{KN}/_{m^2}$. Moreover, the load capacity of the worst loaded beam is determined when it is subjected to 500 degrees Celsius temperature on one side and ambient temperature on the other side.

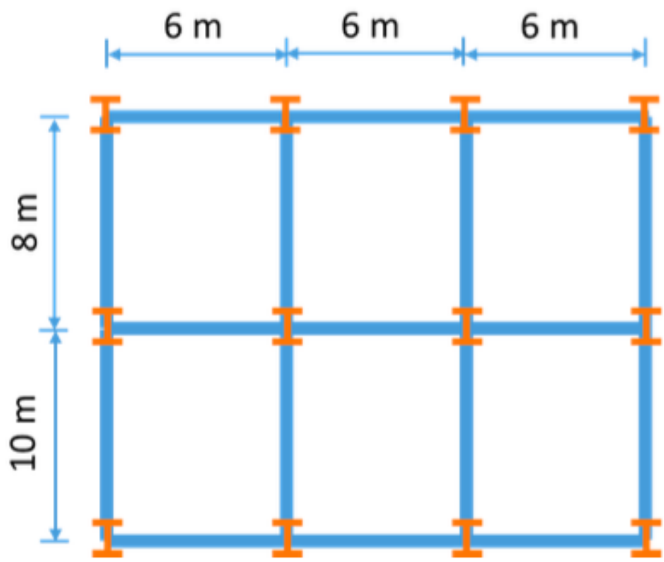


Figure 1. Proposed floor plan

Table1. Design Parameters

Number of stories	3
Floor height	3 m
Column sizes	UC 152 X 152 X 23
Beam sizes	UB 127 X 76 X 13
Steel grade	S275
RC slab thickness	125 mm
Floor loading	7 $^{KN}/_{m^2}$

1.1. Problem statement

The client has requested the following numerical analysis to check the structural stability as well as the structural fire safety of the multi-storey building.

1. Determining the first six natural frequencies and the modal shapes of the frame structure with/without slabs. As it is obvious, in modal analysis, we determine the natural frequency, mode shapes, and mode participation factors. The modal analysis enables the design to prevent resonant vibration or vibrate at a specific frequency and gives engineers an idea of how the design will respond to different dynamic load types (Kumar, et. Al., 2020).

2. Analyzing the structural behavior of the frame with slabs under given static loading of 7 $^{KN}/_{m^2}$ to better understand the structural stability.

3. Calculating the load capacity of the worst loaded beam when it is subjected to 500-degree Celsius temperature on one side, and ambient temperature on the other side which is going to be set to 22° C in Ansys. Justification of the boundary conditions will be provided.

1.2. Aims and Objectives

This project aims to investigate a 3-storey building with a steel frame structure using Ansys workbench 2023 R2 to create a 3D finite element model and be able to validate the numerical model using other research methods. To achieve these objectives, the outcomes mentioned below are expected.

- Understand the principles of the finite element method.
- Set up suitable numerical models.
- Validate computer models.
- Establish links between analysis and design and enhance understanding of

structural design.
- Carry out various case studies to gain a deeper understanding of the structural behavior.

2. Static Structural Analysis
2.1. Ansys model setup
2.1.1. Finite Element Modelling
As previously stated, the 3D finite element model of the residual building requested by the client has been modeled using ANSYS workbench 2023 R2 software. Nowadays ANSYS software is useful civil engineering software because of its compatible features in giving engineers the ability to not only design their desired structures but also ensure the safety and stability of the structures such as buildings, bridges, etc utilizing ANSYS software.

2.1.2. material properties
While using a static structural analysis system in Ansys, the material properties of both steel and reinforced concrete (RC) have been defined under the Engineering data tab. Steel will be used to define columns and beams, and reinforced concrete will be used to define slabs. Structural steel will be used for steel The structural steel has a yield strength of 275 MPa and a tangent modulus equal to 1% of its Young's modulus, which is 2 GPa. Also, under the engineering data tab and the engineering data source the design parameters of reinforced concrete slabs were specified. The detailed material properties and dimension used for beams, columns, and slabs are shown in Table 2.

Table 2. Structural Elements and Material properties:

Element	Material	Cross Section	Properties												
			Mass per meter	Depth of section	Width of section	Thickness		Root radius	Depth between fillets	Ratios for local buckling		Second moment of area (I)		Radius of gyration (i)	
						Of web	Of flange			flange	web	Axis	Axis	Axis	Axis
				D	B	t	T	r	d	B/2T	d/t	x-x	y-y	x-x	y-y
			kg/m	mm	mm	mm	mm	mm	mm			cm^4	cm^4	cm	cm
beam	S275 Steel	UB 127 X 76 X 13	13	127	76	4	7.6	7.6	96.6	5	24.2	473	56	5.35	1.84
column	S275 Steel	UC 152 X 152 X 23	23	152.4	152.4	5.8	6.8	7.6	123.6	11.2	21.3	1250	400	6.54	3.7

Element	Material	Cross section	Thickness
			t
			mm
Slab	Reinforced Concrete	Rectangular Surface	125

2.1.3. Geometry

According to Figure 1, the 2D-floor plan of the building was sketched using Space Claim and analyzed, as usual, using workbench. After sketching the beams and the columns, the cross sections for the beam were defined as UB 127 X 76 X 13 And the cross section for the column was defined as UC 152 X 152 X 23 according to the design parameters provided in Table 1. From a simple unit of creating beams and columns, the rest of the building was created using pattern features to create the first floor. As shown in Figure 2 the slabs were created by filling the four edges as a surface element separately to show that there is a connection between the beam and the slab. Then the first-floor geometry was patterned to create the second and third floors. Before proceeding to the workbench and going ahead with analysis there are two things that we need to make sure they are all right. First, we need to make sure that every element is connected so that when we use meshing for finite element analysis the building works as one unit. To do that, I made sure to combine all the structures in geometry. The second is to make sure the direction of the I-shape cross-section used for beams and columns is in the correct orientation. For instance, when a column bends it should bend in its major axis instead of minor. When it comes to the beams, the flange needs to face up to support the RC slabs, so the minor axis needs to be in the horizontal direction. All the wrong orientation was rotated at 90°.

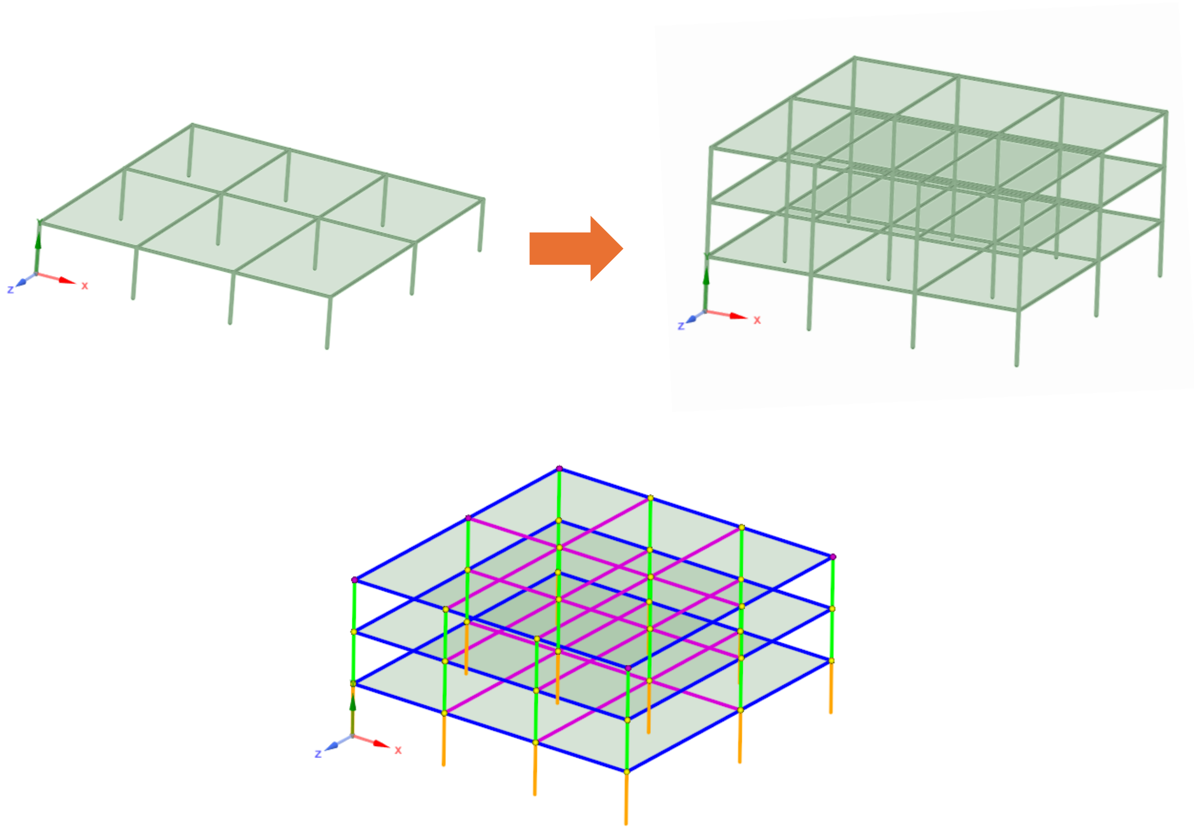


Figure 2. 3D model of the structural steel frame with slabs sketched and patterned in space claim

After updating the space claim model to the mechanical module the surfaces were selected, and a thickness of 125 mm was added. Then the material of beams and columns was assigned to be structural steels and slabs as reinforced concrete.

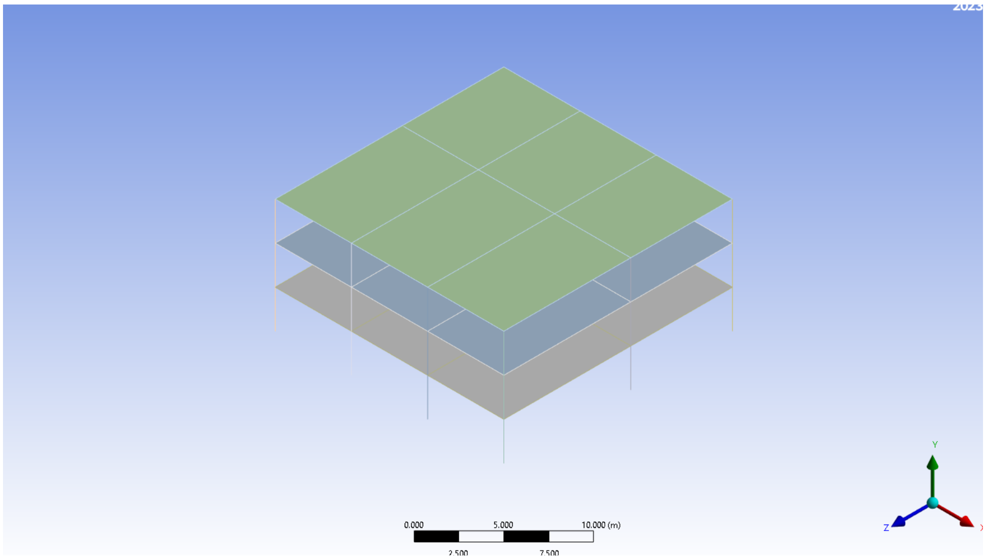


Figure 3. Updated space claim structure to the workbench

2.1.4. Boundary Conditions

2.1.4.1. Support

The boundary condition for this 3-storey building model was defined as fixed at the bottom of the ground floor column with the knowledge that all the buildings are expected to have foundations.

2.1.4.2. Loading

For the structural analysis, another boundary condition was specified which was the pressure or the static loading of 7 $^{KN}/_{m^2}$ applied on 3 surfaces. (combined slabs). This pressure for the proposed building was applied as -7000 pa in the Y direction to represent that it acts downwards.

2.2. Mesh Sensitivity Analysis

To get an appropriate mesh size the mesh sensitivity analysis was carried out to get more reliable and better data from the Ansys model. For this study, different mesh sizes from 5000 mm to 50 mm were tried to get the results for deformation, find the most converging one, and then stop the mesh size changing then proceed with the chosen one. In Table 3 You will be able to see the total displacement that has been achieved using each mesh. Moreover, the time that Ansys needs to run the model for the following mesh size can be found in the table which will affect our choice of mesh size.

Table 3. Mesh Sensitivity Analysis

Mesh size (mm)	Max deformation (mm)	Running time (s)
5000	52.874	1.23
2000	71.724	1.67
1000	81.683	2.33
500	83.248	4.26
250	86.455	13.55
100	90.346	67.23
50	91.323	264.67

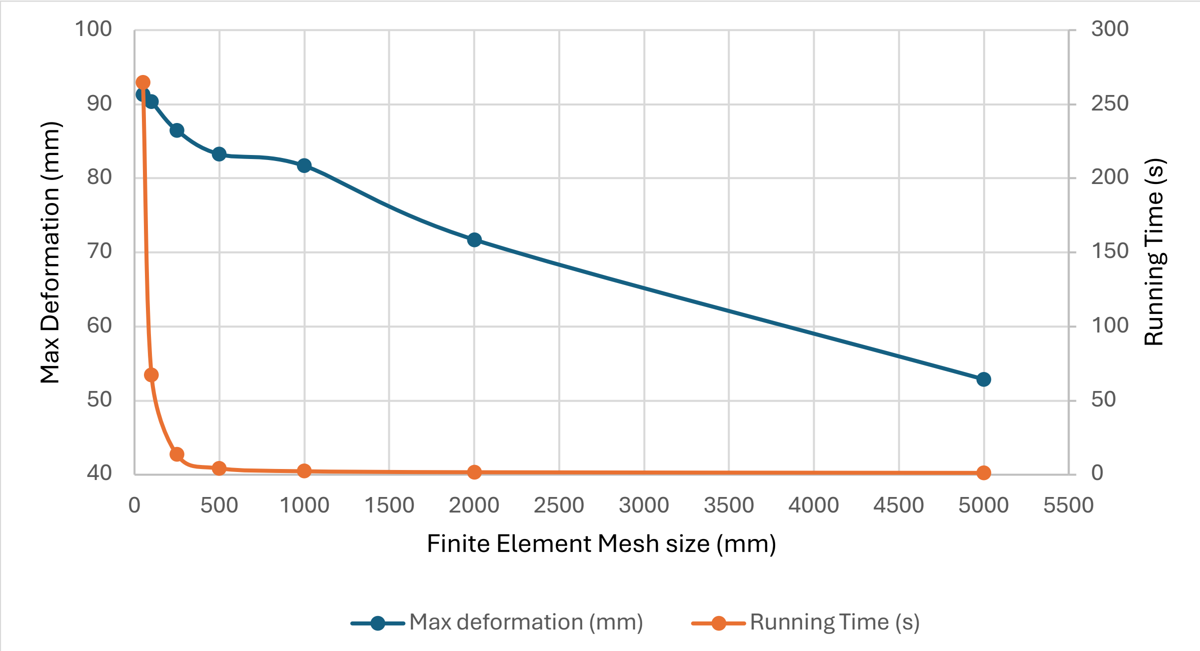


Figure 4. Mesh Sensitivity Analysis

According to the results being represented in Figure 4, it is obvious that, by decreasing the mesh size the deformation values become more accurate however the running time increases. According to the graph the most suitable mesh size in terms of accurate deformation results and low running time, for this model has been chosen for the rest of the modelling.

Moreover, in order to get a better quality of the mesh by maintaining the same shape of the finite elements, face meshing was generated using the quadratic method. In Figure 5 The mesh size generated for the structure is visible.

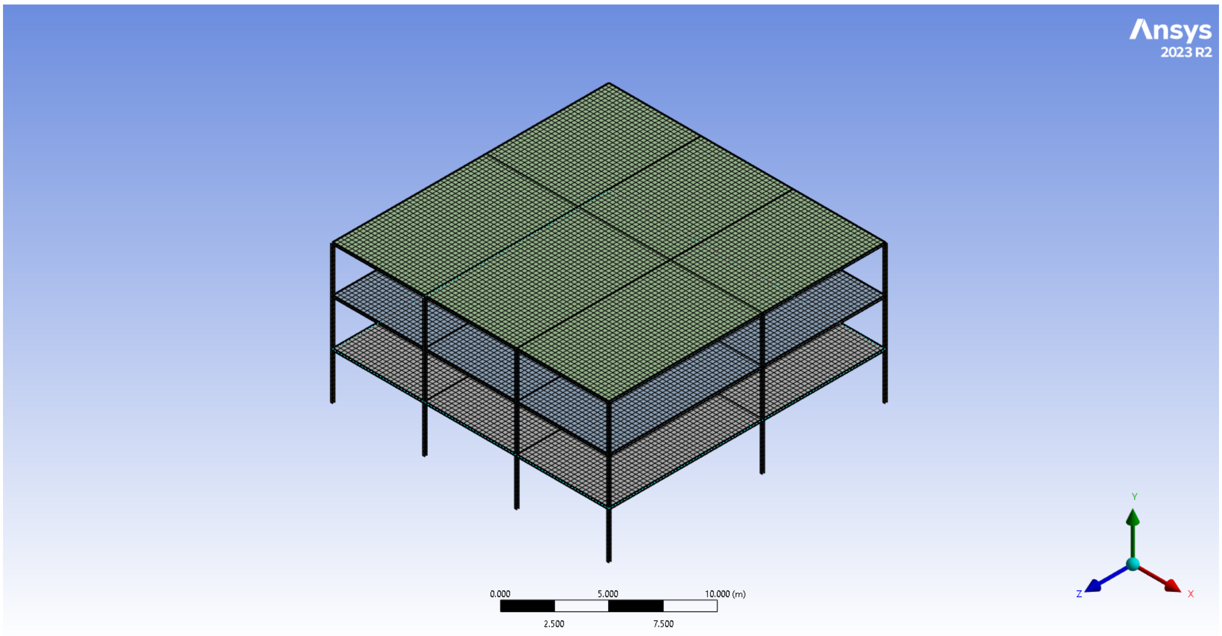


Figure 5. The mesh size generated for the structure

2.3. Results and Discussion

In this section, according to the client's requested analysis, the FE model of the steel structure building was investigated to determine the first six natural frequencies with slabs and without slabs, and then the following results were obtained. Moreover, the results of the static structural behavior of the building under given static loading of 7 $KN/_{m^2}$ will be highlighted.

2.3.1. Modal analysis without Slabs

The first six natural frequencies of the designed steel frame structure without slabs were studied to determine the critical frequencies during external loading. The investigation of a model's deformation mode shapes serves as a visual representation of how a structure will deform at its natural frequencies. As is obvious in Table 4, the first natural frequency is 1.2359 Hz, and the sixth frequency is 2.5049 Hz. The frequencies around 1.7 Hz should be avoided for the presented structure. If not, it will cause structural deformation. Figures 6 to 11 show the results of the deformation mode shapes of the model with their corresponding natural frequencies.

Table 4. Results for six natural frequencies for steel frame structure without slabs

Mode	Frequency (Hz)	Max Deformation (mm)
1	1.2359	0.88969
2	2.2041	0.73678
3	2.2258	1.0843
4	2.3871	0.75893
5	2.4719	0.90948
6	2.5049	1.733

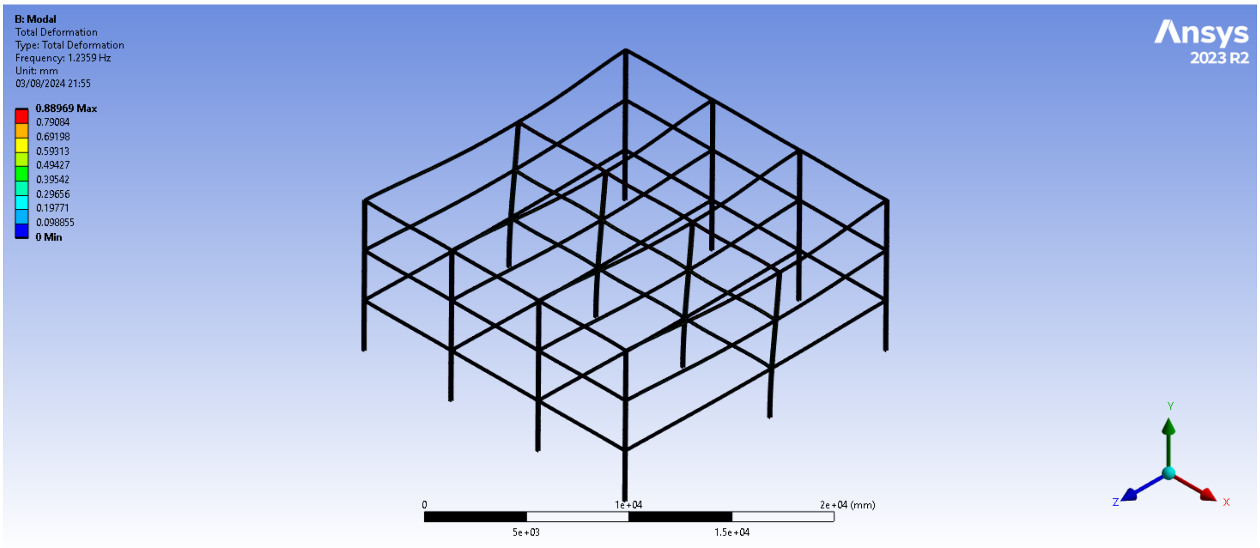


Figure 6. Total deformation of the structure without slabs at a natural frequency of 1.2359 Hz

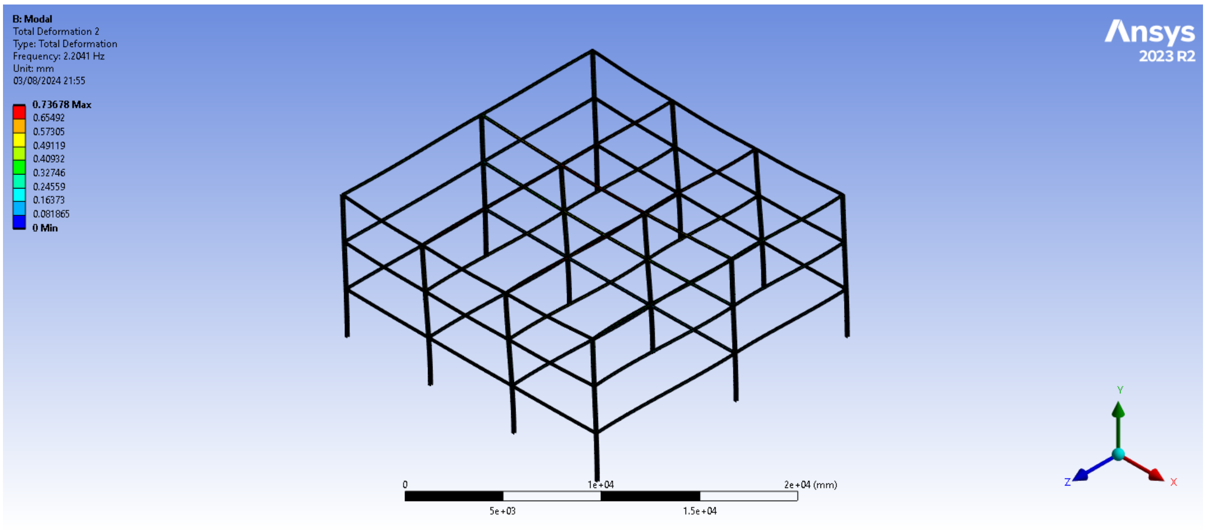


Figure 7. Total deformation of the structure without slabs at a natural frequency of 2.2041 Hz

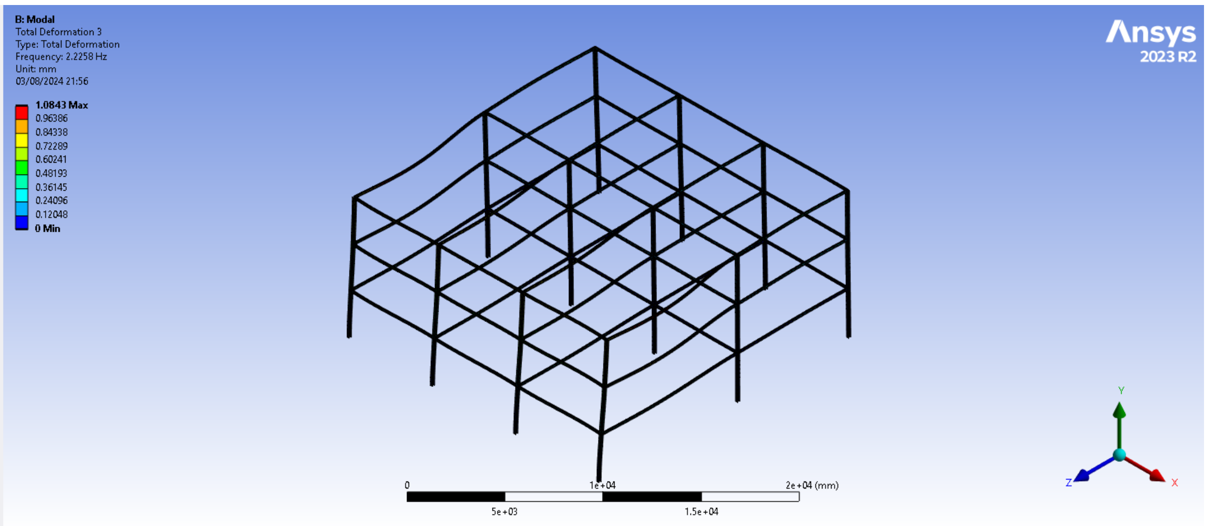


Figure 8. Total deformation of the structure without slabs at a natural frequency of 2.2258 Hz

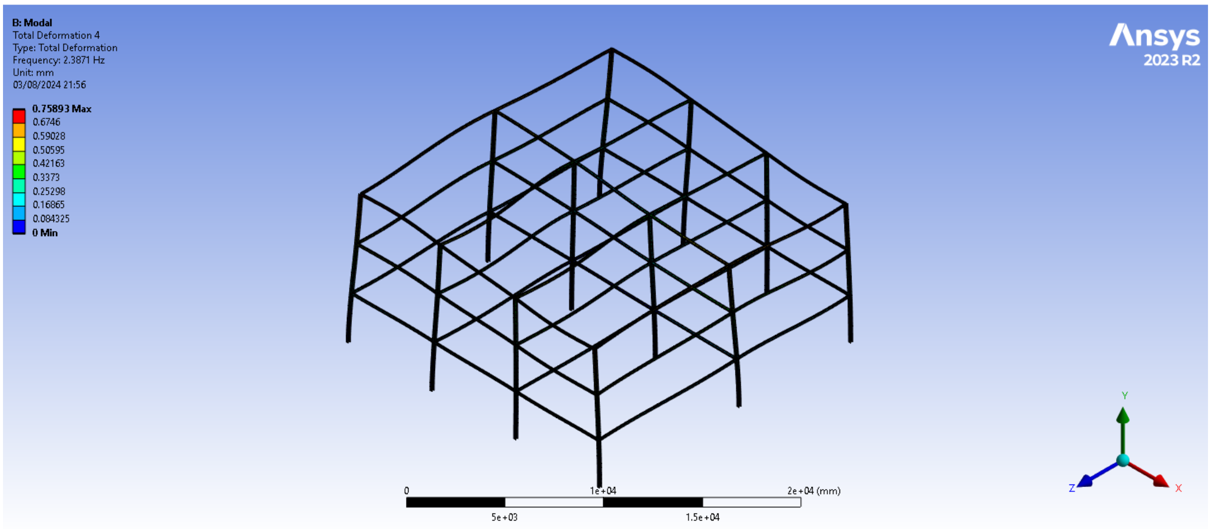


Figure 9. Total deformation of the structure without slabs at a natural frequency of 2.3871 Hz

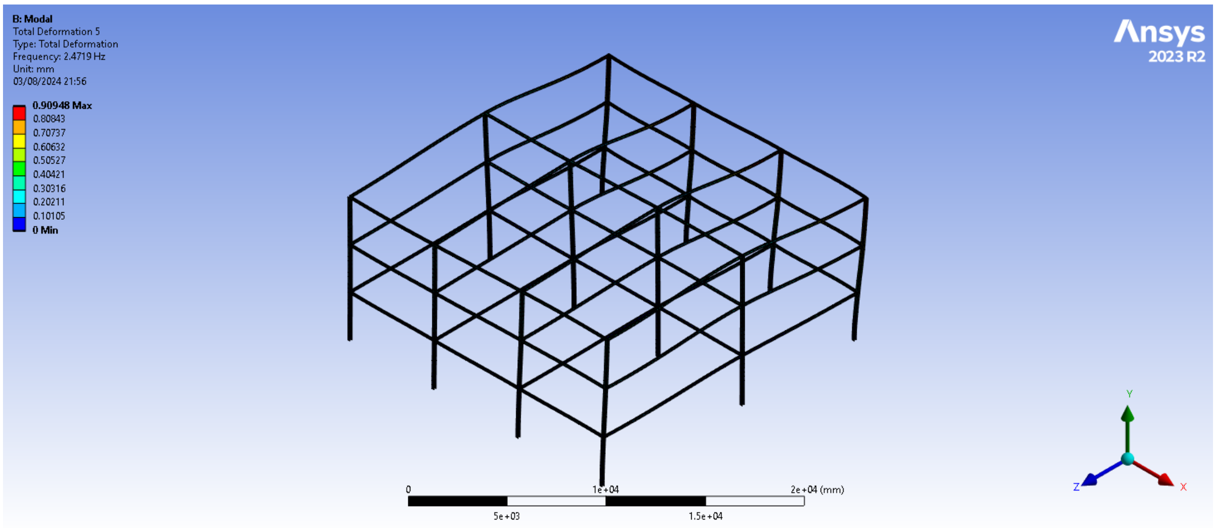


Figure 10. Total deformation of the structure without slabs at a natural frequency of 2.4719 Hz

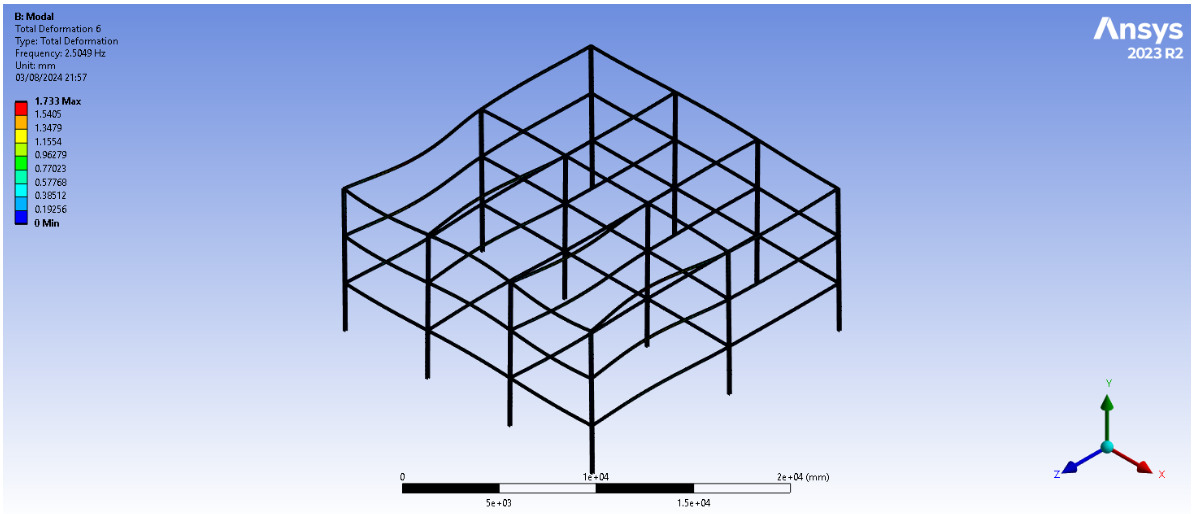


Figure 11. Total deformation of the structure without slabs at a natural frequency of 2.5049 Hz

Based on Table 4 and the mode shapes depicted in Figures 6 to 11, it's evident that the structure's deformation is not consistent with increasing frequencies. However, by looking deeply at the data collected from mode shapes, it is understood that the highest frequency corresponds to the highest deformation.

2.3.2. Modal analysis with Slabs

As stated in the previous heading, similar to that, the first six natural frequencies of the designed steel frame structure with slabs were studied to determine the critical frequencies during external loading. Table 5 shows the results of max deformation versus frequency in six natural mode shapes. The frequencies range from 0.37062 Hz to 1.9631 Hz. Comparing the deformation of the structure with and without slabs, the deformation is lower with slabs. For example, the deformation at the sixth natural frequency without slabs (which is the highest frequency) is the same as the deformation at the fourth natural frequency without slabs. This shows that the slabs' presence decreases the range of the deformation.

Table 5. Results for six natural frequencies for steel frame structure with slabs

Mode	Frequency (Hz)	Max Deformation (mm)
1	0.37062	0.075841
2	0.61889	0.85311
3	0.68392	0.13854
4	1.1871	0.081919
5	1.866	0.079865
6	1.9631	0.79072

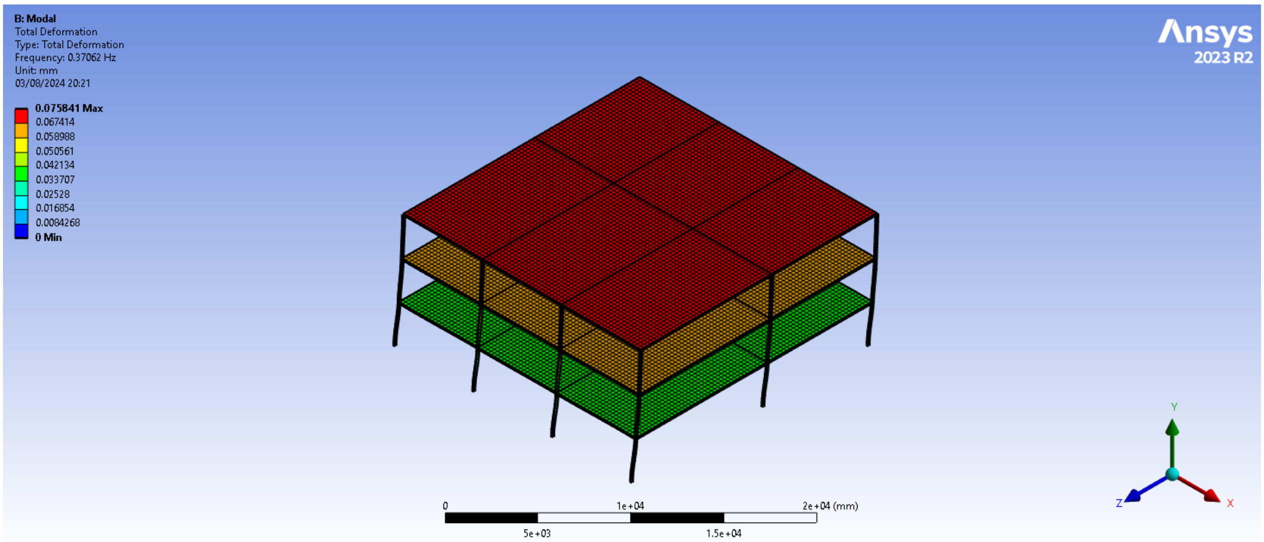


Figure 12. Total deformation of the structure with slabs at a natural frequency of 0.37062 Hz

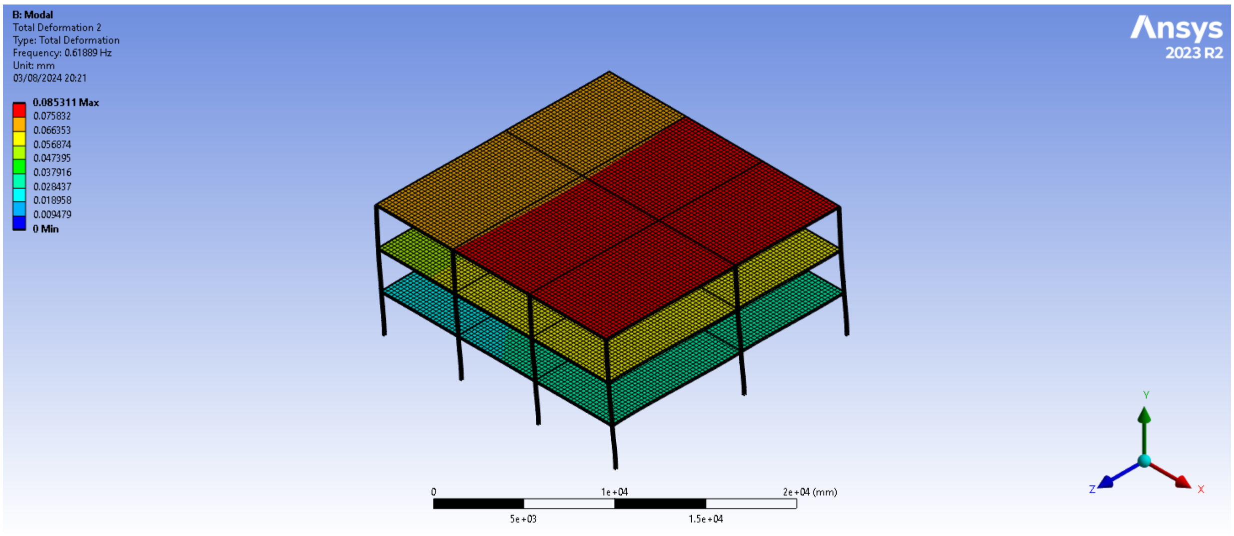


Figure 13. Total deformation of the structure with slabs at a natural frequency of 0.61889 Hz

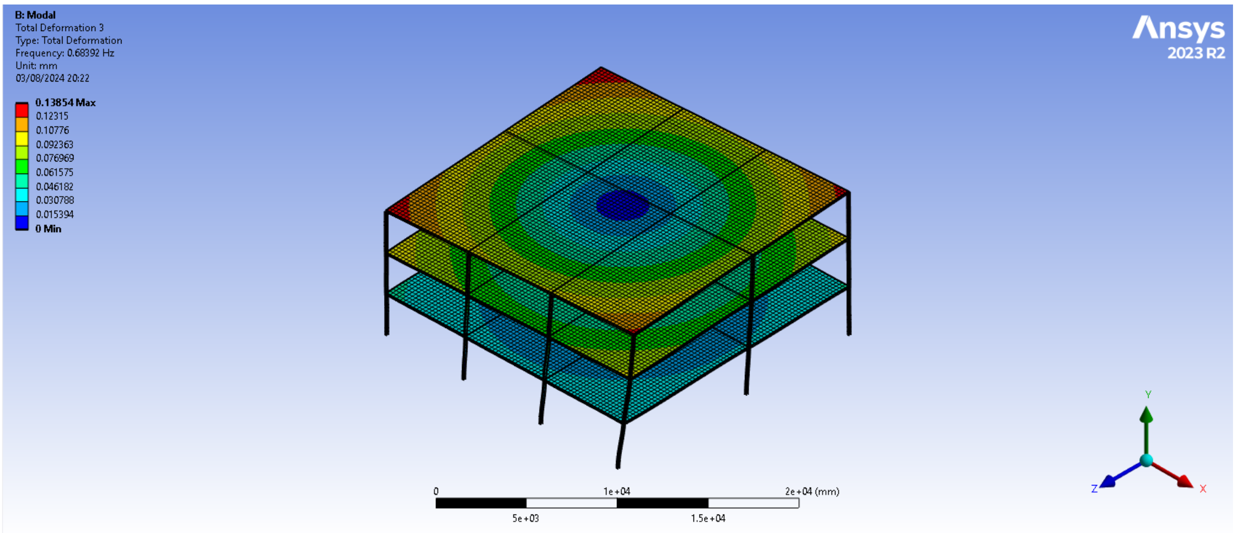


Figure 14. Total deformation of the structure with slabs at a natural frequency of 0.68392 Hz

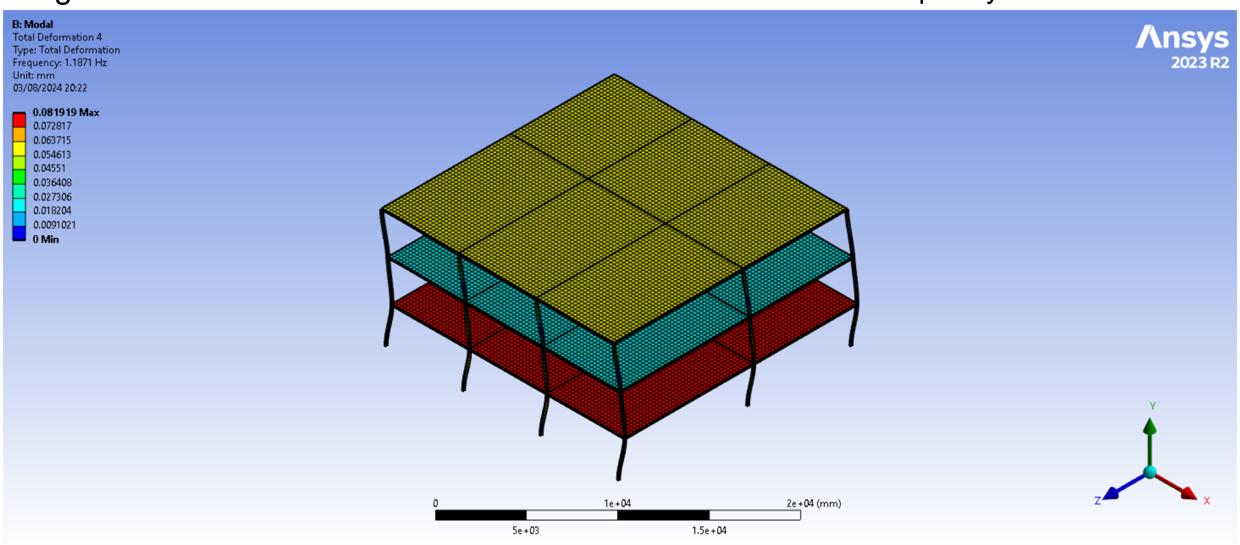


Figure 15. Total deformation of the structure with slabs at a natural frequency of 1.1871Hz

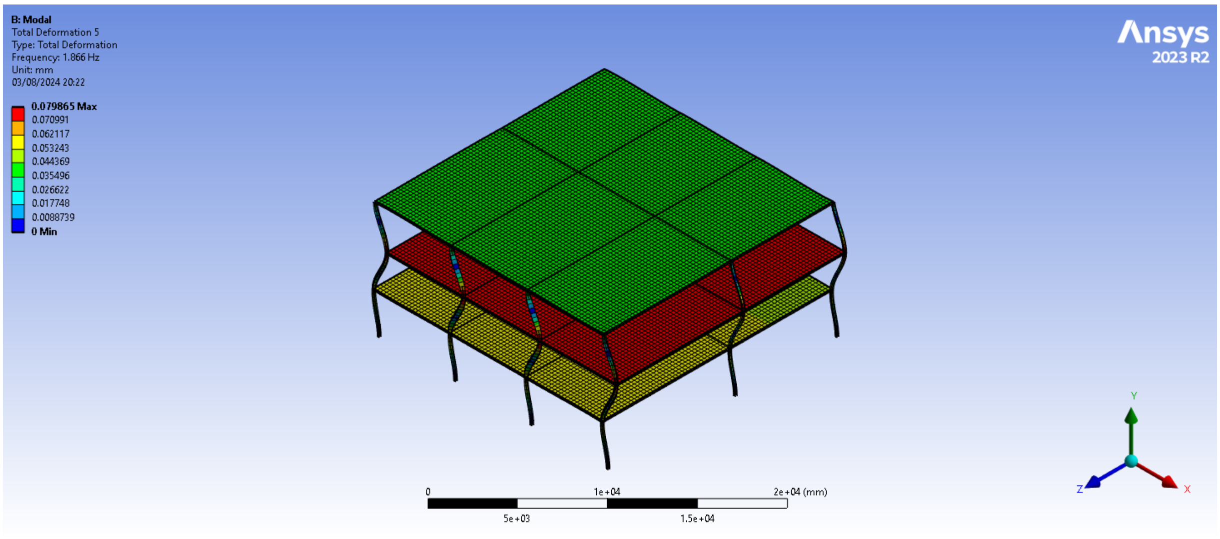


Figure 16. Total deformation of the structure with slabs at a natural frequency of 1.866 Hz

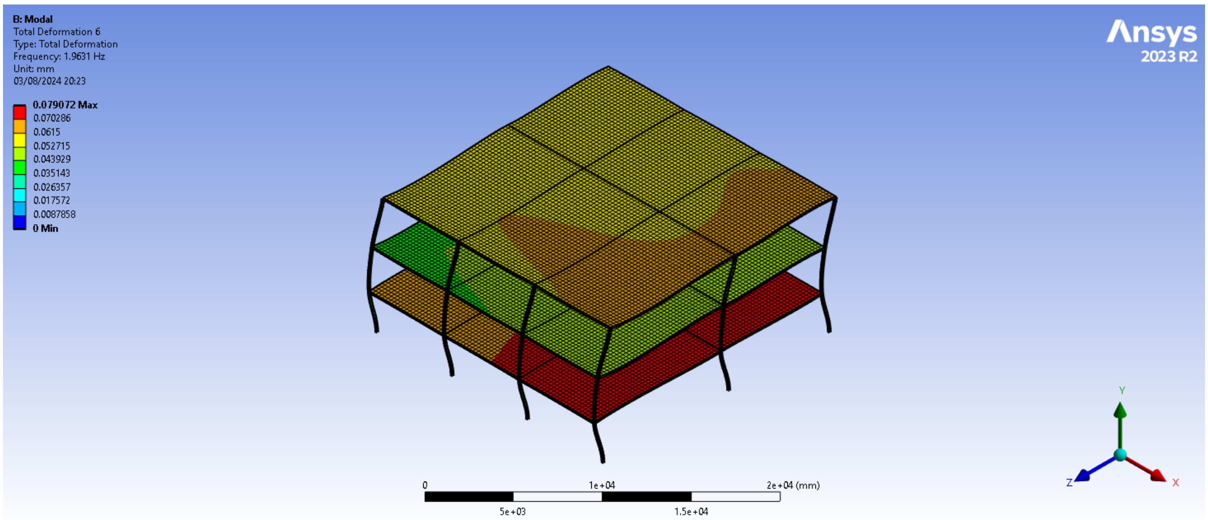
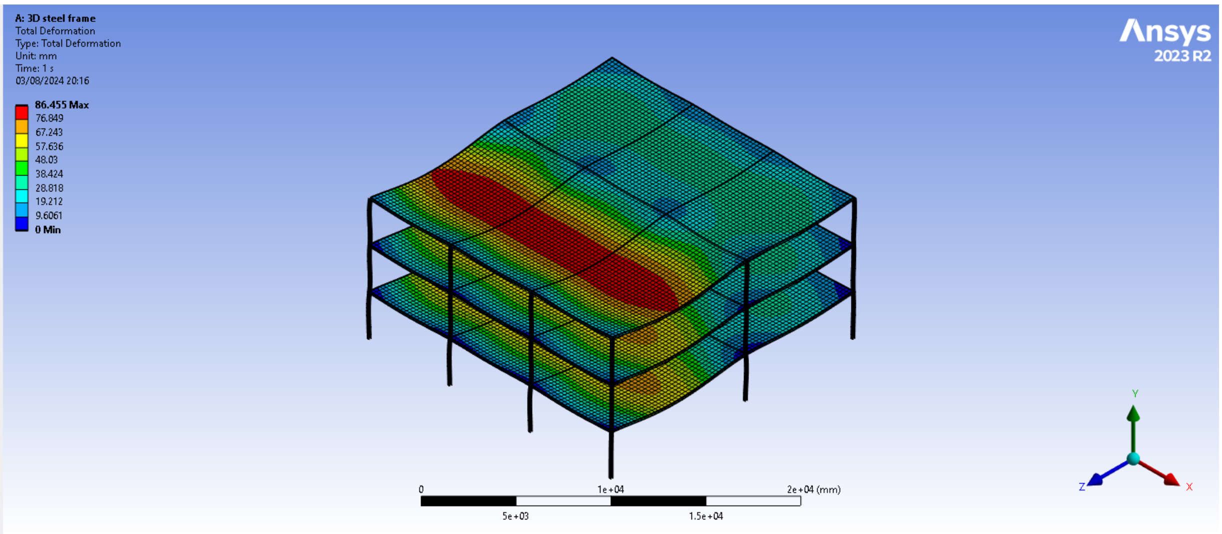


Figure 17. Total deformation of the structure with slabs at a natural frequency of 1.9631 Hz

2.3.3. Static Loading Analysis

The static structural analysis has been done on the model to determine the behavior of the building under determined loading which is 7 $KN/_{m^2}$. As it is obvious in Figure 18, the maximum deformation is 86.455 mm. This means that when the structure is subjected to the 7 $KN/_{m^2}$ external loading, it will max deform 86.5 mm. Based on Figure 18, the max deformation happens at the slabs with a higher span length of 10 meters which is reliable.



Tabular Data

	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	1.	0.	86.455	36.068

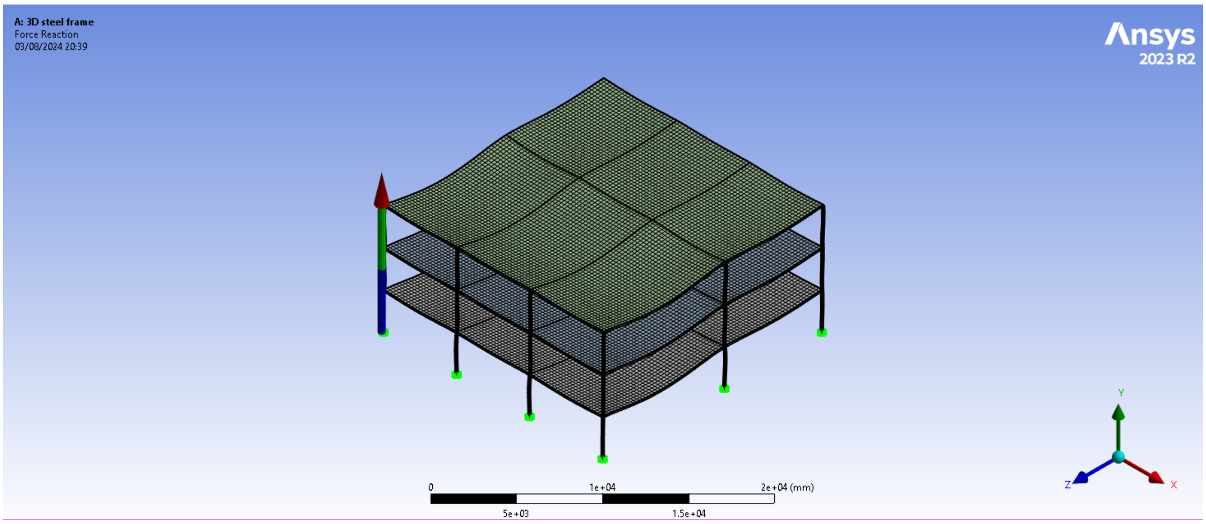
Figure 18. Total Deformation of the building under 7 $^{KN}/_{m^2}$ Static Loading

2.3.3.1. Validation

To validate Finite element Analysis, reaction force was added under the solution in Ansys. In order to establish that the model was set correctly and the results obtained were credible. According to Newton's third law of motion, for every action in nature, there is an equal and opposite reaction (Newton, 2022). In analyzing the steel frame structure with slabs, the pressure applied on the model was 7 $^{KN}/_{m^2}$ so the pressure was converted to a force as calculated below to determine the total applied force on the model.

Width of building plan = 6+6+6= 18m.
Length of building plan = 18m.
Number of the storeys = 3
Total Force = 3 x 18 x 18 x 7 = 6804 KN.

Therefore, the total applied force of 6804 KN downwards in the vertical direction should be equal to the sum of the Reaction forces obtained at the supports that were assigned to the model upwards. The reaction forces considered were the ones in the Y direction since the pressure was applied in the Y direction. According to Figure 19 the total reaction force for the recommended design was 6804 KN which is equal to the total applied force of 6804 KN which means the results were successfully validated.

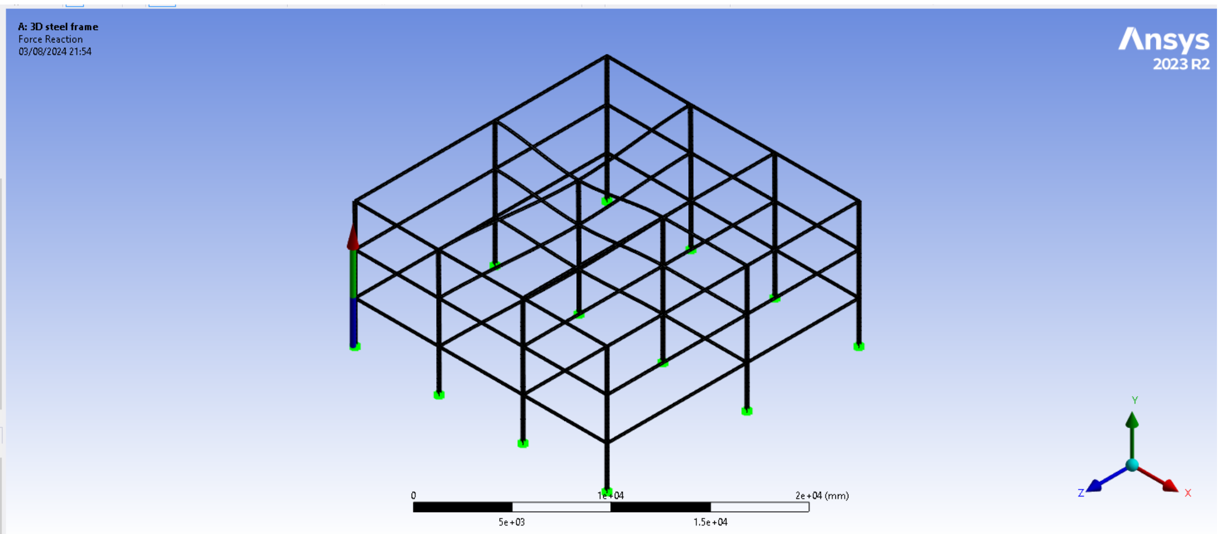


Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1.	1.6484e-005	6.804e+006	-5.0729e-005	6.804e+006

Figure 19. Force Reaction of the structure supports under 7 $^{KN}/_{m^2}$ Static Loading

Also, the validation has been done for the structural steel without slabs as well. As the surface bodies has suppressed with in the model to represent the without slab condition, the pressure of 7 $^{KN}/_{m^2}$ was suppressed as well. So to get the total deformation data to determine first six natural frequencies there was a need to add a force to the structure which was defined on a single node and equal to 1 N. by getting the reaction force results at this stage the answers were again validated according to Newton's law. As it is obvious from the Figure 20 the force reaction in Y direction shows 1N which is exactly equal to the one that applied to the structure. That means the structural model without slabs was validated successfully.



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1.	7.2988e-015	1.	9.7674e-014	1.

Figure 20. Force Reaction of the structure supports under 1N Static Loading

3. Thermal-Mechanical Analysis

3.1. Ansys model setup

3.1.1. Finite Element Modelling

As previously mentioned in the Static Structural Analysis, the Ansys 2023 R2 has been utilized to model the building model. According to Ansys software's capability to analyze and give results in structural fire safety, the load capacity of the worst loaded beam is going to be analyzed under thermal mechanical analysis in Ansys. According to an engineering point of view, numbers 1 and 2 beams located on the first floor of this multi-story building, shown in Figure 21, are considered as the worst loaded beams but number 1 because of its high length span, its location in the middle of the first-floor plan of the building, and following comparative calculations, has been chosen as the worst loaded beam.

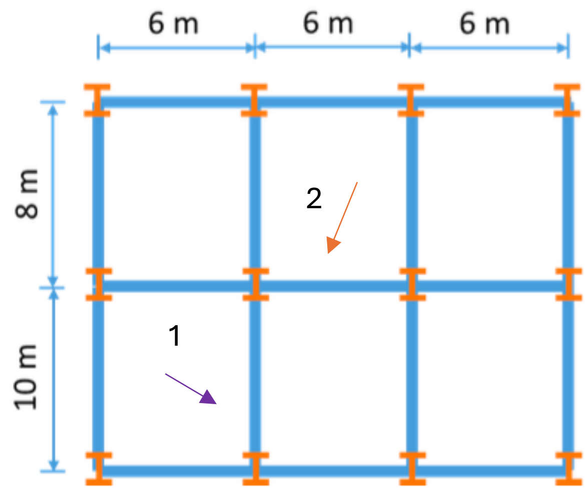


Figure 21. The worst loaded beam location on the first floor of the structure

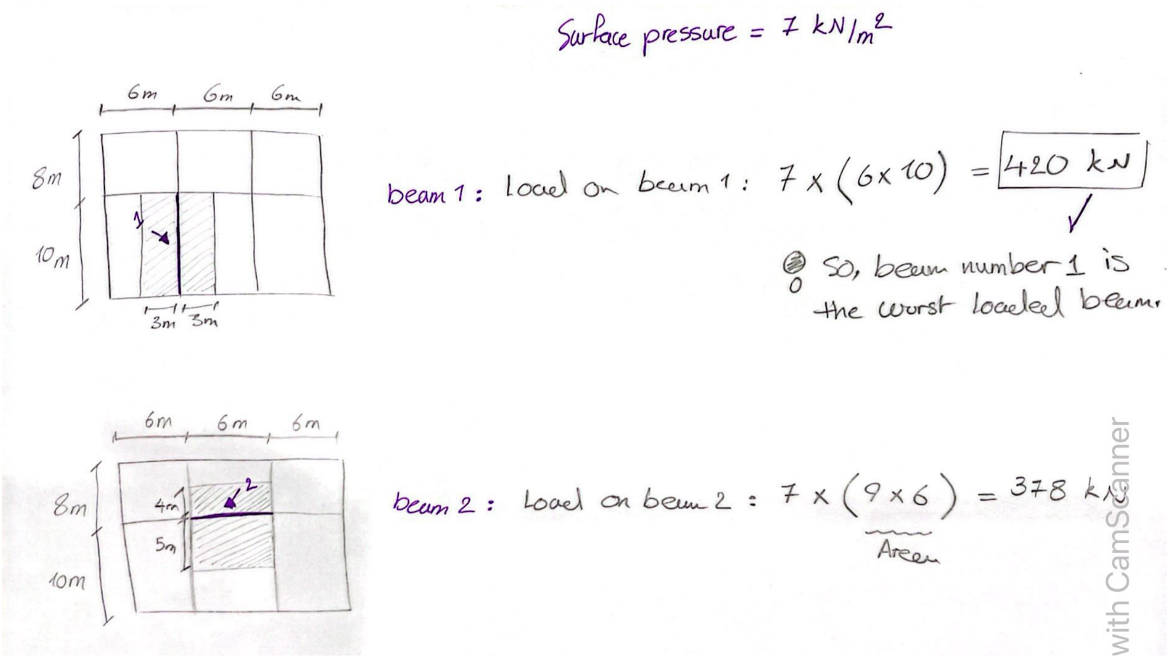


Figure 22. Hand calculation of the worst loaded beam selection

To do the static structural analysis, transient thermal analysis and steady-state analysis have been conducted to understand the overtime trend and steady state of the temperature distribution while there is heat underneath the beam.

3.1.2. Material Properties

While doing Transient Thermal Analysis and Steady-State Thermal Analysis, the material properties are kept to the default steel material properties in Ansys.
However, while doing the thermal-mechanical coupled analysis, engineering data was defined as linked to transient thermal analysis. At this stage to continue with the static structural analysis, first, engineering data such as strain stress curve and material properties need to be defined for the steel material under high temperature.

According to the given data for different temperatures in BSEN1993-1-2:2005 Eurocode 3, steel structures fire design, Young's modulus, and plastic behavior have been defined for structural steel(S275) as summarized in Table 6. Also, the Poisson's ratio for all the temperatures is going to be defined as 0.3. According to Figure 23, Young's modulus of the material given the temperature variation is defined in Ansys's engineering data tab and graph obtained.

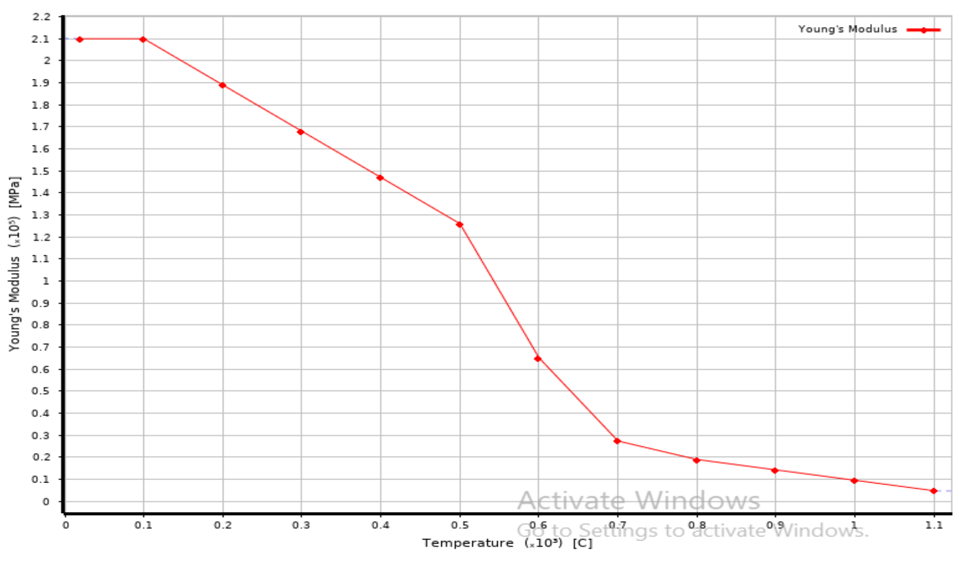


Figure 23. Young's Modulus of S275 Steel Temperature Reductions

The plastic behavior was defined by adding multi-linear isotropic hardening to the structural steel material properties. In each temperature, we need to define the relation between plastic strain and stress according to Table 6 and from 20 degrees to 500 degrees. the steel behaviour given temperatures was simplified to 3 stages given the data from EN1993-1-2:2005. Figure 24 was obtained by adding the plastic strain, proportional limit stress, and effective yield strength to the isotropic multi-linear data to the engineering data in Ansys from 20 degrees to 500 degrees. Plastic strain is obtained by deducing the proportional limit strain from 2 percent.

Table 6. S275 Steel Material properties under high-temperature BSEN 1993-1-2:2005

Temperature	Effective yield strength fy (Mpa)	proportional limit strength fp (Mpa)	Elastic modulus Ea (MPa)	proportional limit strain εp	Plastic strain	Tangent modulus
20	275	275	210000	0.0013	0.019	0
100	275	275	210000	0.0013	0.019	0
200	275	222	189000	0.0012	0.019	2819
300	275	169	168000	0.0010	0.019	5602
400	275	116	147000	0.0008	0.019	8301
500	215	99	126000	0.0008	0.019	6011
600	129	50	65100	0.0008	0.019	4145
700	63	21	27300	0.0008	0.019	2215
800	30	14	18900	0.0007	0.019	856
900	17	10	14175	0.0007	0.019	321
1000	11	7	9450	0.0007	0.019	214
1100	6	3	4725	0.0007	0.019	107
1200	0	0	0			

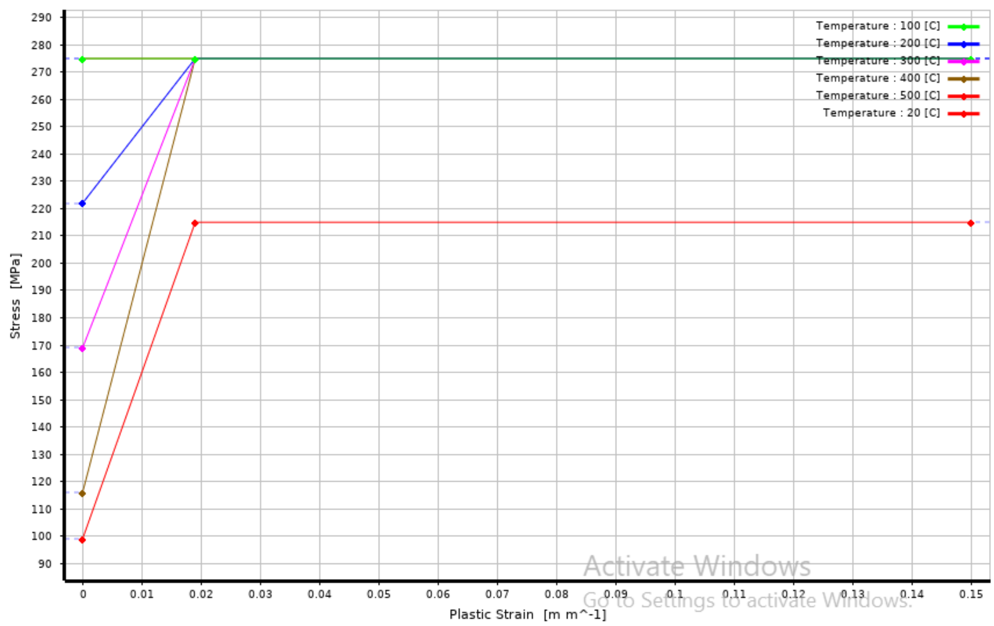


Figure 24. Stress-Strain of S275 Steel Given Temperature Reductions According to EN 1993-1-2:2005.

3.1.3. Geometry

To model the amended beam, which is the I section, the section properties, and dimensions of the UB 127 x 76 x 13 section are used according to the British steel document shown in Table 2. After sketching the 3D shape of I section beam in the geometry tab of Ansys, the beam was extruded 10000 millimeters to represent the exact dimension of the beam in the building. Then the solid body of the beam was created.

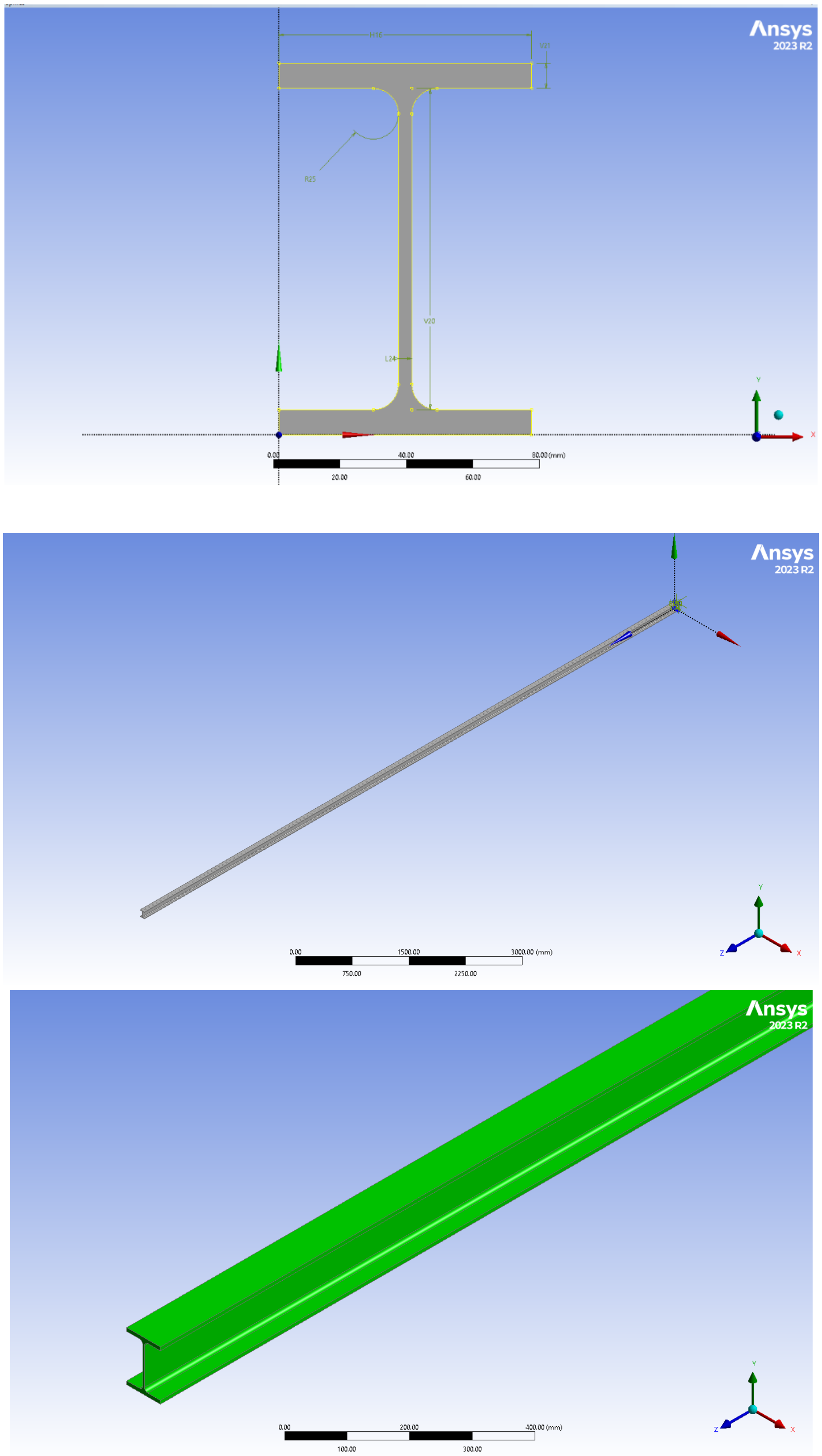


Figure 25. Geometry of the worst loaded beam

3.1.4. Boundary Condition

3.1.4.1. support

The boundary condition was defined as fixed support on one end and roller support on the other end to reflect the reality of the beam while it is in the building. The boundary conditions selection was based on the recommended supports for beams in (Sijia Liu, et. Al.,2024), and (H. Amrous, et. Al., 2023). For roller support, the X and Y components were set to 0 mm displacement and the Z component was set free.

3.1.4.2. Transient Thermal Analysis

In order to do the thermal analysis other boundary conditions have been defined for the beam. The transient thermal analysis is time-dependent and shows how the temperature distributes over time.

3.1.4.3. Steady-State Thermal Analysis

The next and desired thermal analysis is Steady-State Thermal Analysis which only looks at the static condition. In thermal analysis, the only bottom surface of the flange is going to be exposed to the heat source at 500°C temperature, and the rest of the surfaces of the I section beam are going to be at ambient temperature (22°C) and being exposed to a convection current of stagnant air to simulate how the beam would be in reality.

3.1.4.4. Thermal Mechanical Coupled Analysis

To do the thermal-mechanical coupled analysis, to see the steel beam load capacity under high temperature new static structural dragged on to the thermal analysis in Ansys. All the modal and the solutions are shared in the new structural analysis. After updating the modal, the setup tab will load to proceed with the rest of the modelling. Then the load is imported from steady-state thermal to the static structure analysis as the Thermal load as shown in Figure 26. In order to get results at the solution, total deformation and equivalent stress were added.

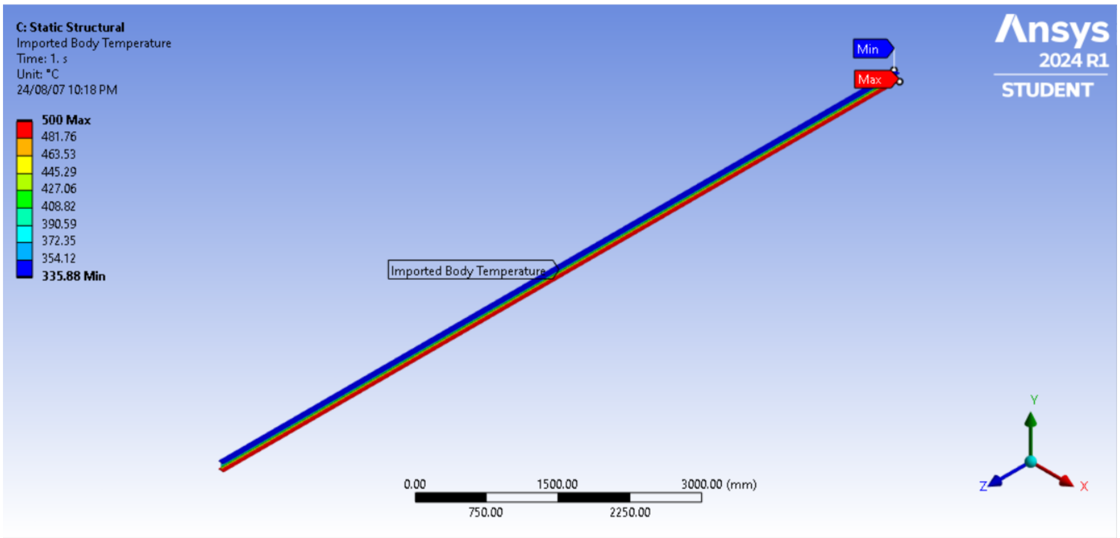


Figure 26. Thermal Load imported to the model

After getting the results of the deformation and equivalent stress from the model, these results are the behavior under temperature only. There is a need to determine how much load the beam can bear under this high temperature. To get the load capacity of a beam the nodal displacement was applied on the model. As the nodal displacement can only be applied to the named selection (named as loading pad), the node selection was defined using the worksheet according to Figure 27 by adding two ranges of mesh nodes from 2550 mm to 2600 mm and 7400 mm to 7450 mm. Then the filtering was defined to only keep the nodes on the top surface of the beam.

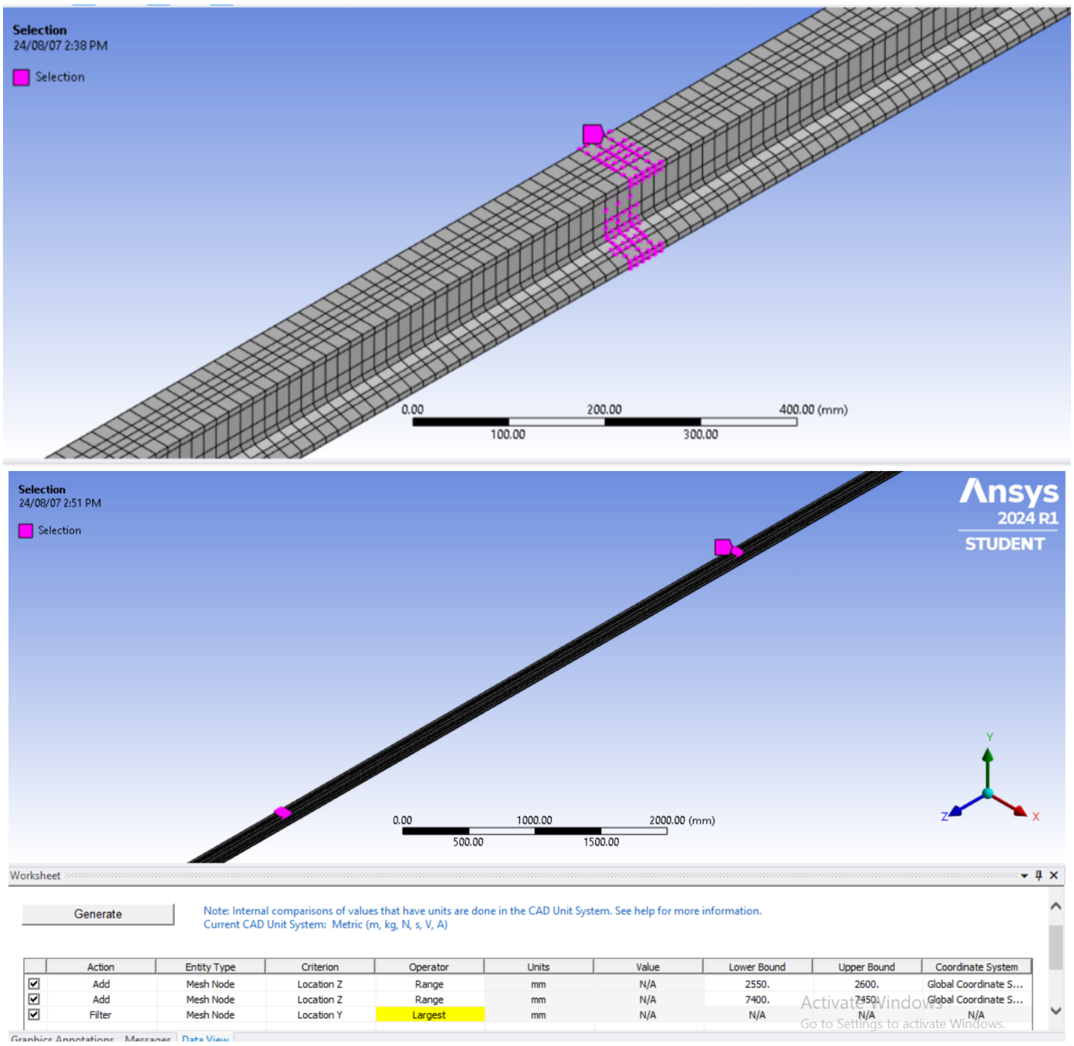


Figure 27. Node selection of the beam

Then a remote point was created on those nodes as it is going to control the boundary conditions later on. According to Figure 28, you can easily see that the remote point is created right in between the loading pads. Then the remote displacement was applied to the remote point. To get better data, 3 time steps were created to monitor 3 steps of results. First time step, is heat transfer only, second time step, holding the model to stabilize, and the last and third time step is displacement and pressing the beam down by 50 mm.

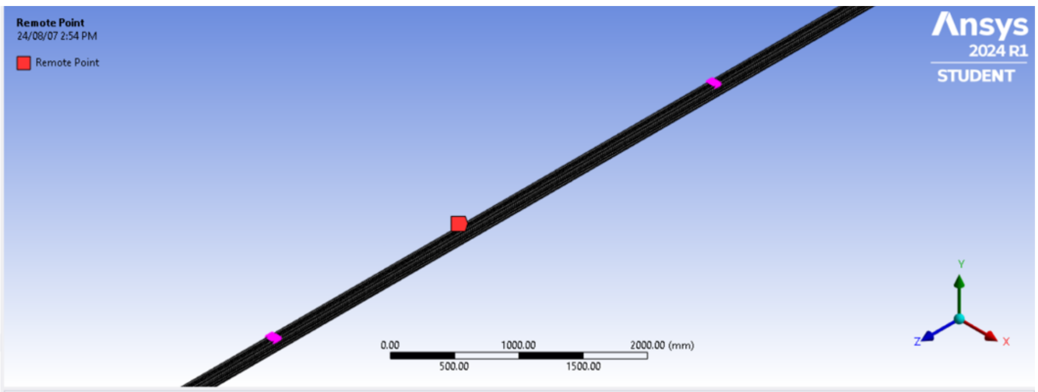


Figure 28. remote point defined for the beam

To get clearer results and to force Ansys to do exactly what we want, some commands were defined to the remote point.

- To remember the pilot name number, “my_pilot= _npilot” was defined.
- Step number 2, “D, my_pilot, all, %_FIX%” was defined to hold the displacement in the second step.
- Step number 3, “D, my_pilot,uy,UY(my_pilot)-50” was defined to apply -50 mm displacement in the Y direction in step 3.

Then the results for directional deformation and force reaction have been obtained.

3.2. Mesh Sensitivity Analysis

Like Static Structural Analysis, in order to get an appropriate mesh size, the mesh sensitivity analysis was carried out to get more reliable and better data from the Ansys model. Mapped Face meshing was applied to the beam, using the quadratic method, to get a better quality of the mesh by maintaining the same shape of the finite elements and to have uniformly distributed mesh.

For thermal analysis, different mesh sizes from 5000 mm to 250 mm were tried to get the results for deformation, find the most converging one, and stop changing the mesh size. In Table 7 You will be able to see the total deformation that has been achieved using each mesh. Moreover, the time that Ansys needs to run the model for the following mesh size can be found in the table which will affect our choice of mesh size. According to the max deformation and the running time the most suitable mesh size for this model has been decided to be 250 mm.

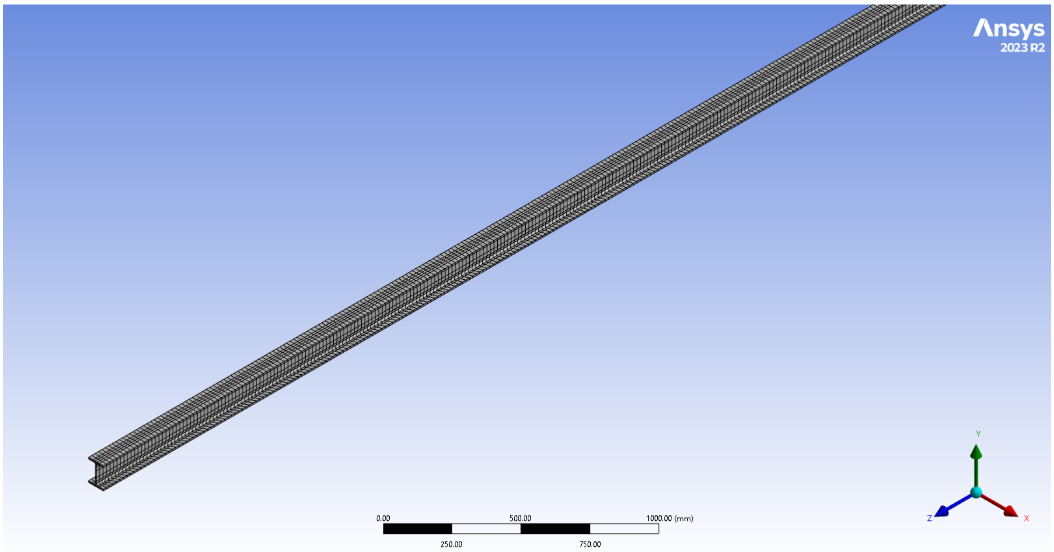


Figure 29. Mesh and face meshing of the worst loaded beam

Table 7. Mesh sensitivity analysis

Mesh size (mm)	Max deformation (mm)	Running time (s)
5000	243.78	51
2000	239.22	67
1000	237.22	92
500	235.62	123
250	235.53	152
100	235.52	161
50	235.49	183

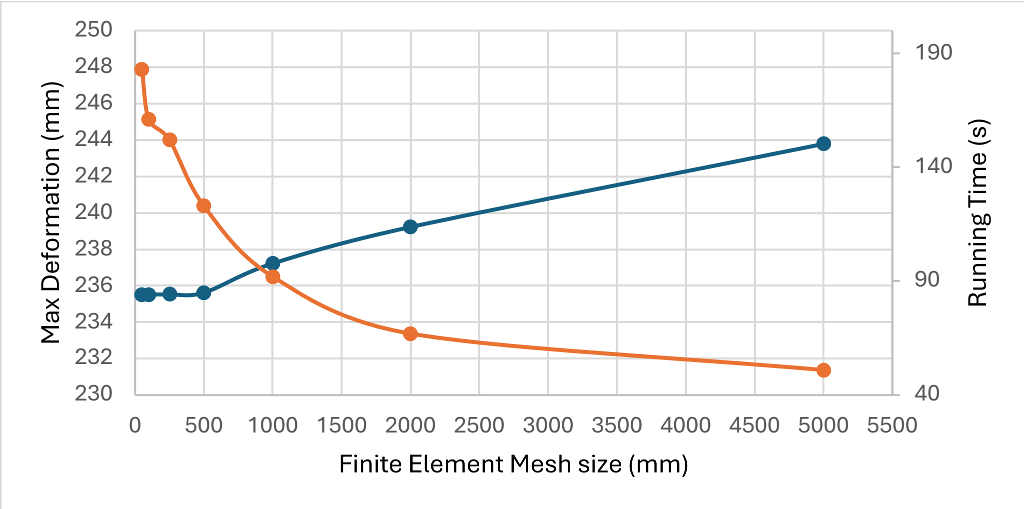


Figure 30. Mesh Sensitivity Analysis

As it's obvious from the graphs from Figure 30, all the results of the maximum deformation from the mesh size 500 mm to 50 mm are convergence so due to the high running time of the 50 mm mesh size, the 250 mm mesh size is a good choice for this model.

3.3. Results and Discussion

3.3.1. Transient Thermal Analysis

However, the client didn't ask us to undertake transient thermal analysis, but it's done to get the best overview of temperature distribution over time in the beam. Figure 31 highlights the results of this analysis. As it's apparent at the beginning the bottom is 500°C and the other parts are 22°C. After sometimes passing the heat transfers to the web and then gradually goes to the flange. As you can see the temperature stabilized around 7500 seconds. The top surface's temperature is 335.88°C and the bottom flange which is exposed to the highest degree is 500°C.

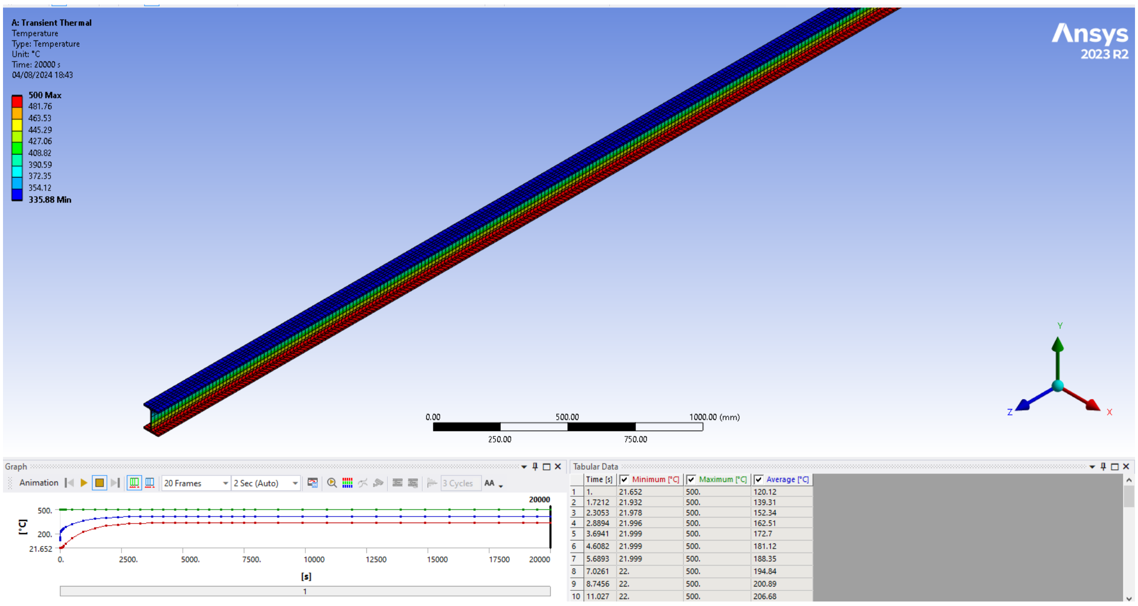


Figure 31. Transient thermal analysis results of the worst-loaded beam

3.3.2. Steady State Thermal Analysis

The steady-state analysis results illustrated in Figure 32, are the temperature distribution once the heat transfer is stabilized. As is visible from the figure in this analysis, we just get one result for minimum, average, and maximum temperature.

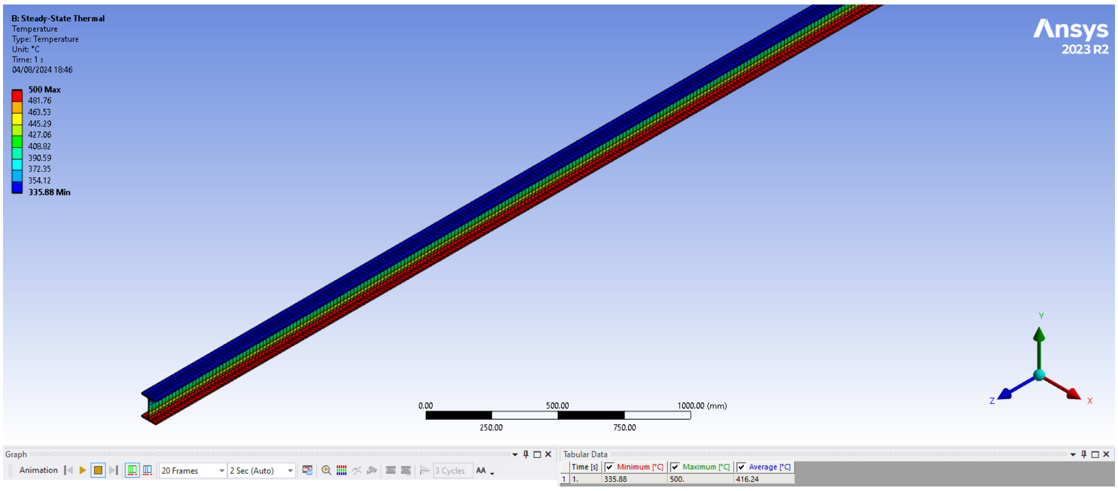
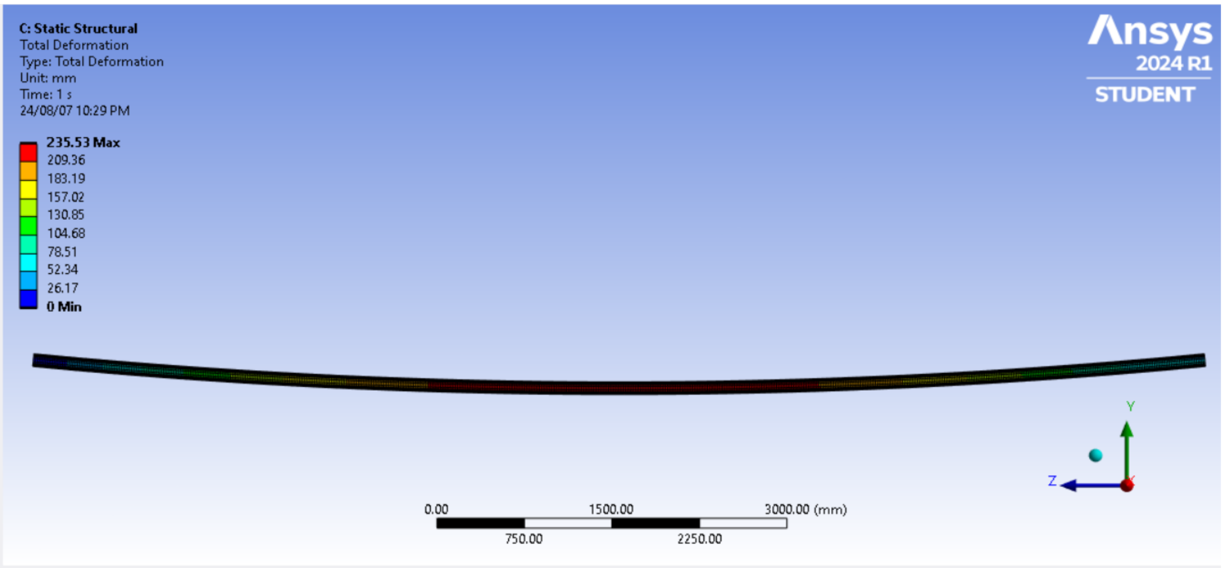


Figure 32. Steady-state thermal analysis results of the worst-loaded beam

3.3.3. Thermal Mechanical Coupled Analysis

The results of total deformation and the equivalent stress have been obtained from Ansys while doing the static structural analysis under Steady-State thermal analysis and with thermal load imported from Steady-State analysis. Figure 33 represents how the beam deforms under temperature. The top and bottom flanges expand differently as top and bottom temperatures are different.



	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	0.2	0.	47.221	32.098
2	0.4	0.	94.388	64.162
3	0.7	0.	165.05	112.2
4	0.835	0.	196.8	133.78
5	0.9175	0.	216.17	146.95
6	1.	0.	235.53	160.11

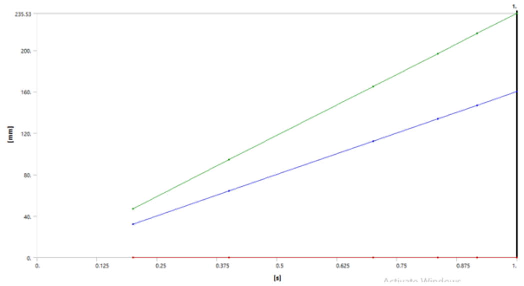
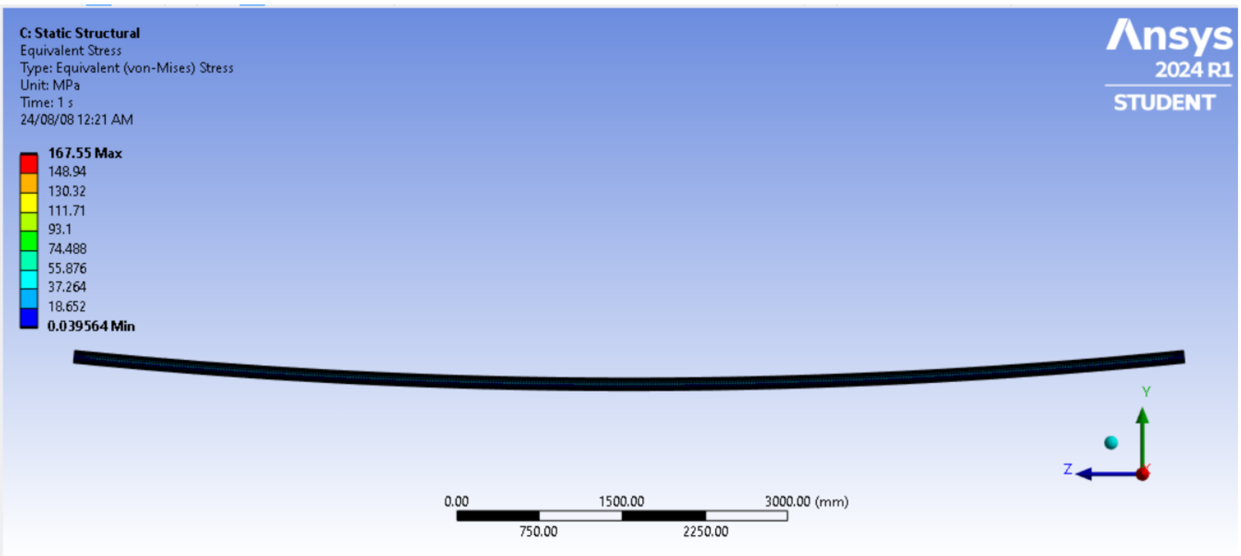


Figure 33. total deformation of thermal-mechanical analysis

As shown in Figure 33, the maximum deformation of the beam under a steady-state temperature load is 235.53 mm. This means that if the bottom of the beam is exposed to a temperature of 500°C due to fire, the beam will only deform by 235.5 mm. According to the following results of equivalent stress under a given thermal load, the maximum stress experienced by the beam under thermal load is 167.55 MPa, representing the highest stress.



	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	0.2	1.1039e-002	508.48	3.7247
2	0.4	2.2607e-002	248.57	6.6242
3	0.7	3.364e-002	193.01	9.9519
4	0.835	4.015e-002	184.51	11.056
5	0.9175	4.1566e-002	176.88	11.627
6	1.	3.9564e-002	167.55	12.108

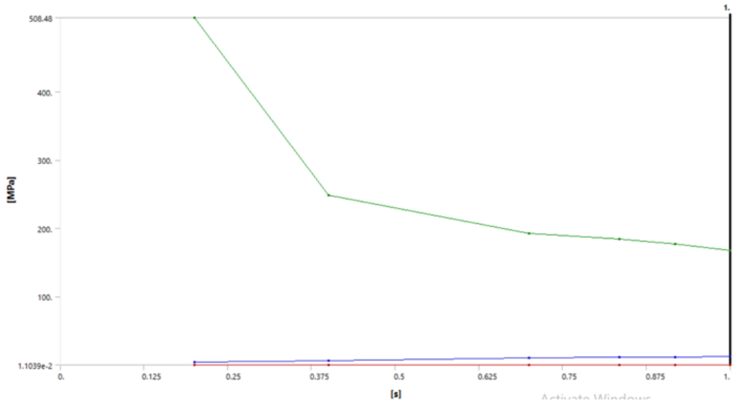


Figure 34. Equivalent Stress of thermal-mechanical analysis

while the remote displacement was applied to the beam model in 3 steps, according to the results from the directional deformation, as is obvious in Figure 35 that the deformation happens at 3 steps as we asked Ansys to do.

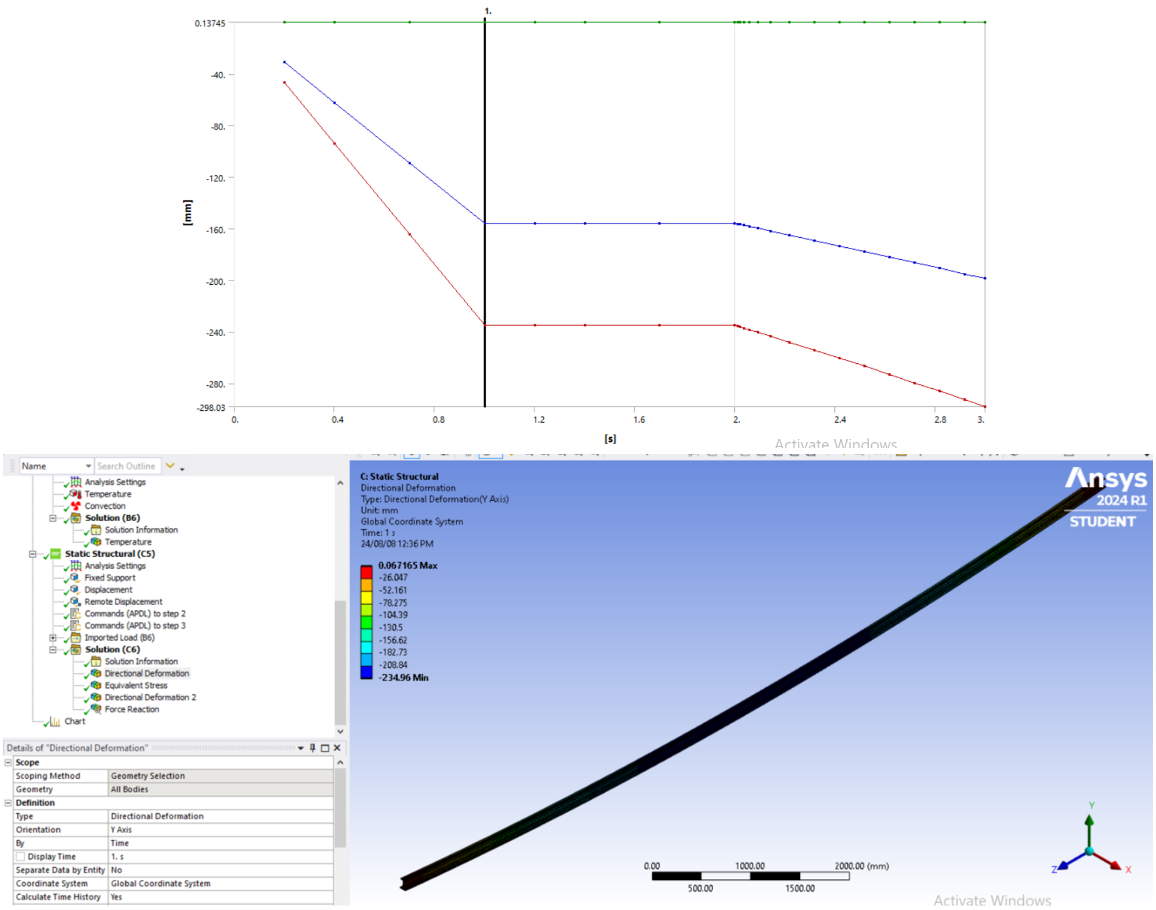


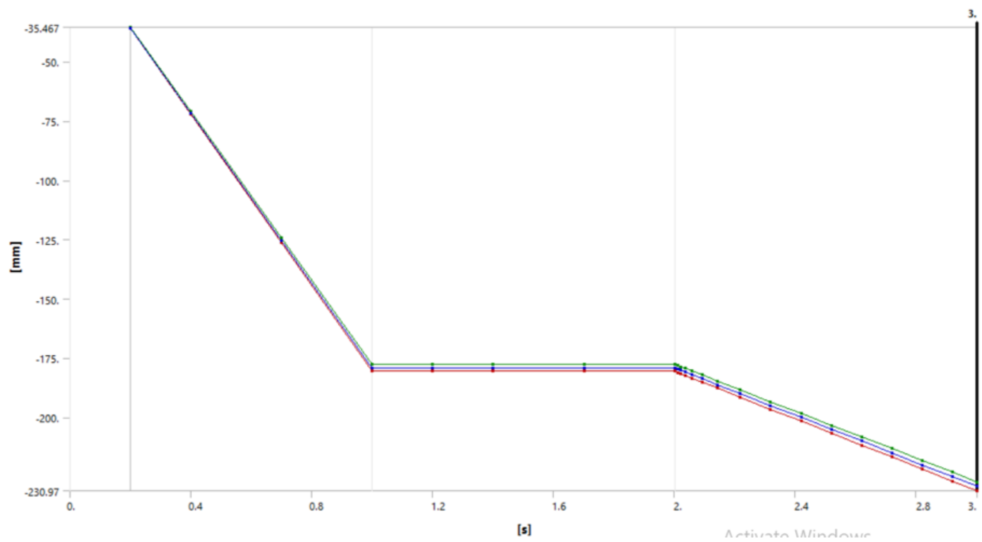
Figure 35. The values of directional deformation of the Y-axis

from the values in Figure 36, we can obvious that at the beginning of load step 3, it starts going down.

	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	0.2	-46.917	9.0018e-002	-31.123
2	0.4	-93.884	0.13745	-62.287
3	0.7	-164.42	0.13592	-109.09
4	1.	-234.96	6.7165e-002	-155.91
5	1.2	-234.96	6.7167e-002	-155.91
6	1.4	-234.96	6.7167e-002	-155.91
7	1.7	-234.96	6.7167e-002	-155.91
8	2.	-234.96	6.7167e-002	-155.91
9	2.01	-235.57	6.5758e-002	-156.33
10	2.02	-236.17	6.4345e-002	-156.75
11	2.035	-237.08	6.2216e-002	-157.39
12	2.0575	-238.45	5.9005e-002	-158.34
13	2.0912	-240.49	5.4148e-002	-159.77
14	2.1419	-243.55	4.8133e-002	-161.91
15	2.2178	-248.12	3.973e-002	-165.12
16	2.3178	-254.2	3.0069e-002	-169.35
17	2.4178	-260.34	2.4392e-002	-173.6
18	2.5178	-266.61	2.0134e-002	-177.88
19	2.6178	-273.07	1.5999e-002	-182.19
20	2.7178	-279.56	1.336e-002	-186.51
21	2.8178	-286.1	1.1247e-002	-190.84
22	2.9178	-292.65	9.1029e-003	-195.18
23	3.	-298.03	7.3106e-003	-198.74

Figure 36. directional deformation data of the Y-axis

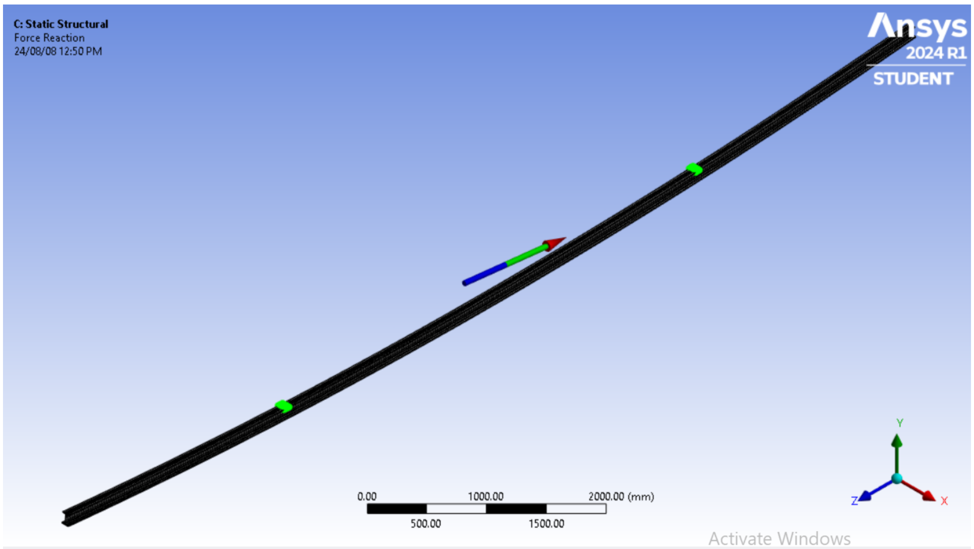
The load-deformation curve shown in Figure 37, it's just representing the Y-axis deformation in loading pads only.

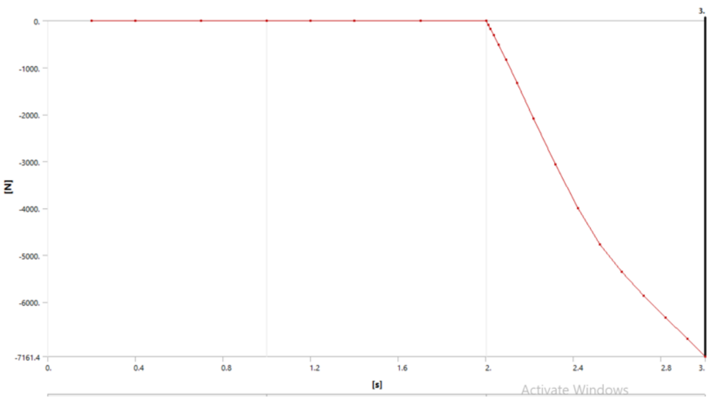


	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	0.2	-36.048	-35.467	-35.761
2	0.4	-72.142	-70.98	-71.57
3	0.7	-126.36	-124.32	-125.36
4	1.	-180.58	-177.67	-179.15
5	1.2	-180.58	-177.67	-179.15
6	1.4	-180.58	-177.67	-179.15
7	1.7	-180.58	-177.67	-179.15
8	2.	-180.58	-177.67	-179.15
9	2.01	-181.08	-178.17	-179.65
10	2.02	-181.59	-178.66	-180.15
11	2.035	-182.34	-179.41	-180.9
12	2.0575	-183.48	-180.52	-182.02
13	2.0912	-185.18	-182.19	-183.71
14	2.1419	-187.73	-184.7	-186.24
15	2.2178	-191.56	-188.46	-190.04
16	2.3178	-196.6	-193.41	-195.04
17	2.4178	-201.64	-198.37	-200.04
18	2.5178	-206.69	-203.32	-205.04
19	2.6178	-211.73	-208.27	-210.04
20	2.7178	-216.77	-213.22	-215.04
21	2.8178	-221.8	-218.18	-220.04
22	2.9178	-226.84	-223.13	-225.04
23	3.	-230.97	-227.2	-229.15

Figure 36. directional deformation data of the loading pads of the Y-axis

Another set of data has been driven out from the modelling which was reaction force at remote displacement in the Y direction. Figure 37 represents these data. Steps 1 and 2 are zero because the first step is heat transfer and step two is holding it and as they are free they show zero.

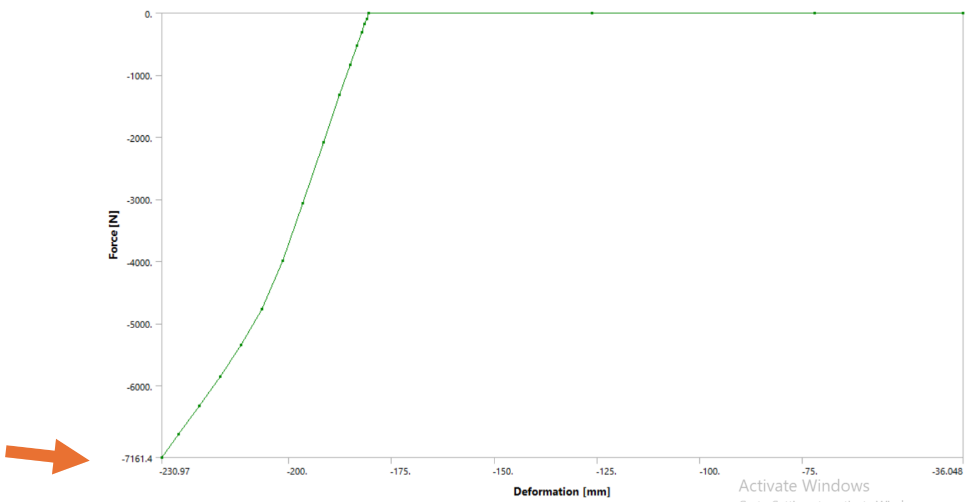




	Time [s]	Force Reaction (Y) [N]
1	0.2	0.
2	0.4	0.
3	0.7	0.
4	1.	0.
5	1.2	-9.3082e-004
6	1.4	-7.114e-004
7	1.7	-6.7458e-004
8	2.	-6.631e-004
9	2.01	-88.125
10	2.02	-177.05
11	2.035	-311.94
12	2.0575	-517.63
13	2.0912	-833.69
14	2.1419	-1323.1
15	2.2178	-2085.2
16	2.3178	-3066.6
17	2.4178	-3990.8
18	2.5178	-4774.1
19	2.6178	-5346.2
20	2.7178	-5863.6
21	2.8178	-6325.4
22	2.9178	-6788.6
23	3.	-7161.4

Figure 37. Reaction force at remote displacement

As you can see in Figure 38, the minimum deformation is plotted against the force reaction. This is the load-deformation curve and from this curve it is understandable **that the maximum load capacity of this beam which is the worst-loaded beam, is 7161.4 N.**



Tabular Data

	Steps	Time [s]	Directional Deformation 2 (Min) [mm]	[B] Force Reaction (Y) [N]
1	1	0.2	-36.048	0.
2	1	0.4	-72.142	0.
3	1	0.7	-126.36	0.
4	1	1.	-180.58	0.
5	2	1.2	-180.58	-9.3082e-004
6	2	1.4	-180.58	-7.114e-004
7	2	1.7	-180.58	-6.7458e-004
8	2	2.	-180.58	-6.631e-004
9	3	2.01	-181.08	-88.125
10	3	2.02	-181.59	-177.05
11	3	2.035	-182.34	-311.94
12	3	2.0575	-183.48	-517.63
13	3	2.0913	-185.18	-833.69
14	3	2.1419	-187.73	-1323.1
15	3	2.2178	-191.56	-2085.2
16	3	2.3178	-196.6	-3066.6
17	3	2.4178	-201.64	-3990.8
18	3	2.5178	-206.69	-4774.1
19	3	2.6178	-211.73	-5346.2
20	3	2.7178	-216.77	-5863.6
21	3	2.8178	-221.8	-6325.4
22	3	2.9178	-226.84	-6788.6
23	3	3.	-230.97	-7161.4

Figure 38. The load-deformation curve and the load capacity value

4. Conclusion

To conclude, the results that were generated from the Ansys simulation are close to the hand-calculated results which demonstrate that static frame structure is safe for construction and use. The result of our validation shows that the model represents the system correctly. Additionally. the mesh sensitivity analysis shows that the answers converge both in static structural analysis and Thermal analysis so that the mesh chosen was a good mesh size. The structural fire safety analysis produced on the worst loaded beam provides valuable insights into the response of the structure to thermal conditions

As it can be observed the loading capacity of the beam is 7161.4 N which shows that the beam is working well.

References

A.U.R. Dogar, Hafiz Muhammad Ubaid Ur Rehman;, T. Tafsirojjaman;, N. Iqbal;, Experimental investigations on inelastic behaviour and modified Gerber joint for double-span steel trapezoidal sheeting, Structures. 24 (2020) 514-525. 10.1016/j.istruc.2020.01.042

Kumar, Pawan & HARSHA, SURAJ. (2020). Modal analysis of functionally graded piezoelectric material plates. Materials today: proceedings. 28. 10.1016/j.matpr.2020.04.825.

Newton, I. (2022) Sir Isaac Newton's mathematical principles of natural philosophy and his system of the world. Univ of California Press.

Sijia Liu, Li Chen, Bin Feng,
A theoretical method for scaling the dynamic response of steel beams subjected to blast loads, International Journal of Impact Engineering, Volume 187,2024,104918, ISSN 0734-743X,https://doi.org/10.1016/j.ijimpeng.2024.104918.
(https://www.sciencedirect.com/science/article/pii/S0734743X24000435)

H. Amrous, N.M. Yossef, M.H. El-Boghdadi,Experimental study and structural analysis of tapered steel beams with cellular openings,Engineering Structures,Volume 288,2023,116212,ISSN 0141-0296,
https://doi.org/10.1016/j.engstruct.2023.116212.
(https://www.sciencedirect.com/science/article/pii/S0141029623006260)