ANALYSIS AND SIMULATION OF STEEL MEMBERS IN ANSYS AT DIFFERENT TEMPERATURES

[a]Raja Wajahat Zahoor Khan*, [b]Raja Shujahat Zahoor Khan, [c]Muhammad Yaqub

a: Department of Civil Engineering, University of Engineering and Technology, Taxila, Pakistan. wajahatzahoor12@gmail.com

b: Department of Civil Engineering, University of Engineering and Technology, Taxila, Pakistan. shujahatkhan277@gmail.com

c: Department of Civil Engineering, University of Engineering and Technology, Taxila, Pakistan. yaqub_structure@yahoo.com

* Corresponding author

Abstract- The main objective of the project is to analyze and simulate a steel frame using ANSYS software to determine how stresses and deformation vary with temperature. An approximation method is used to accurately represent the non-uniform temperature distribution within a portion of a 3d beam structural finite element. With this method, the impact of temperature fluctuations on the structural behavior of the 3D members may be represented more accurately. The study aims to analyze the behavior of steel members under various heat conditions by examining the stress distribution and deformation patterns. The study showed that the deformation of the steel column at 500 °C was 0.0568m, while at 1000 °C, the maximum deformation was 0.1086 m. The maximum deformation of 0.1240m occurred at 1500 °C. When beams were exposed to 500 °C, 1000 °C, and 1000 °C, they also showed maximum deformation up to 0.066m, 0.1086m, and 0.2351 respectively. According to our modelling results, steel beams went through deformation, cracking, fracture, and ultimate failure as the temperature increased, which ultimately affected their load-bearing capacity. This study also establishes that the span length, loading magnitude, and cross-sectional properties of structural members influence fire resistance behavior.

Keywords- ANSYS, Steel Frame, Temperature

1 Introduction

If the fire is not properly controlled, there is a significant risk to human life. Losses of life and human resources could result from it. Nowadays, the use of structural steel is very common to construct large-span industrial structures and community centres. Before designing the steel structures, it is essential to study steel behaviour under different conditions of fire. Due to their high thermal conductivity, steel structures do not perform very well in fire, which causes shrinkage and deformation of structural members. The behaviour of steel structures to fire or high temperatures is determined by the steel's inherent properties, including stiffness, strength, thermal conductivity, thickness, thermal gradient, tensile property, and thermal expansion. These strength parameters of steel are affected due to stresses induced by thermal action and the lateral buckling of steel, which occurs at high temperatures or in the presence of fire [1].

The mechanical and physical characteristics of steel are completely altered when it is subjected to high temperature and then quickly cooled, so it is not practical to use the fast-cooled steel in the current structure. The strength of steel is unaffected when exposed to below 600 °C; however, at 1100 °C, there is essentially no remaining strength. Steel loses its strength at 2200°C, and its load-bearing capacity completely decreases at 2700°C. Steel beams could fracture if subjected between 650°C and 1100°C. Given that the upper half of the structure rests on the beam and the slab is supported by the beam, the structure can collapse at high temperatures [2].

High temperatures have a notable effect on the performance of steel members and can result in several problems like deformation, fatigue, cracking, spalling, and even failure [3]. To ensure these materials' dependability and safety, it is crucial to understand how they respond to high temperatures [5]. The ANSYS program is a useful tool for modelling and analysing the behaviour of structural elements under various heat conditions. The performance of structural components under high temperatures is examined in this study using ANSYS software. The purpose of this study is to simulate how structural components will behave in high temperatures and to investigate the deformation and stress distribution of structural components at various temperatures.

2 Experimental program

2.1 *Model Preparation in ANSYS*

Several structural elements, such as columns and beams, have a yield strength of 355MPa, were analysed in this study. Fixed, rolling, and pinned supports are considered in our model. Several beams having spans of 5, 15, and 10 meters were considered in this model, as shown in Table 1. The model was subjected to various loads and temperatures, including 25°C, 1000°C, 500°C, and 1500°C, as shown in Table 2. ANSYS software was used to model the 4-meter-long steel beams and columns. The steel used in this study exhibited a Poisson's ratio of 0.3, Young's modulus of 2 × 105 N/mm^2, a density of 7850 kg/m^3, a value of K as 45 W/mK, and a thermal expansion coefficient of 12 x 10^{-6} /°C. The model was exposed to the desired temperatures as part of the analysis's first step, a thermal analysis. For the succeeding structural analysis, the thermal study's findings were subsequently imported.

Table 1. Cross-section properties of members.

Parameter	h (mm)	b (mm)	d (mm)	t_w (mm)	t_f (mm)
Beam	635.8	311.4	540	18.4	31.4
Column	352.5	318.4	246.7	23	37.7

2.2 *Applied thermal loads and boundary conditions.*

Table 2. Applied thermal load on the beam.

Case	**Support Conditions**	**Thermal Load**
Case 1	Fixed Supports	25ºC, 1000ºC, 500ºC, 1500ºC
Case 2	Pinned Supports	25ºC, 1000ºC, 500ºC, 1500ºC

2.3 *Meshing and Simulation*

The meshing of the structure was performed as shown in Figure 2. There are two specifications: a maximum and a minimum level, for meshing analysis. Maximum stress and pressure versus mesh density and other variables were incorporated in the model. We simulated our model by using a linear analysis method with various mesh densities that allowed us to determine which mesh density is most appropriate for each circumstance in the finite element model. The analysis takes longer to run on the software when the mesh density is increased, but the results are more accurate [4]. For FEM models used in the current analysis, the mesh value was set to 5mm.

3 Results

3.1 *Impact of Temperature on Deformation*

Our research shows that a steel beam's behaviour is highly dependent on the temperature to which it is exposed. When the temperature didn't increase, there was no noticeable deformation in the steel beam sections as shown in Figure 1. This is expected because steel is a hard material that can withstand static and dynamic loads without suffering significant deformation. But as the temperature increased, cracking in the beam sections caused deformation.

Compression loads applied to the members caused bending, leading to an abrupt, catastrophic failure condition when the applied load exceeds the critical buckling load of a thin structural element. Analysis shows that as temperature increased, a critical buckling resistance to load decreased. The deformation of the steel column at 500 °C was 0.0568, as shown in Figure 2, while at 1000 °C the maximum deformation was 0.1086m, as shown in Figure 3, and at 1500 °C it was 0.1240m, as shown in Figure 5.

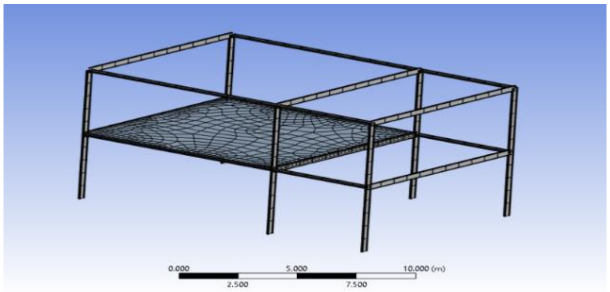

Figure 1. Meshing of beams, slab, and columns.

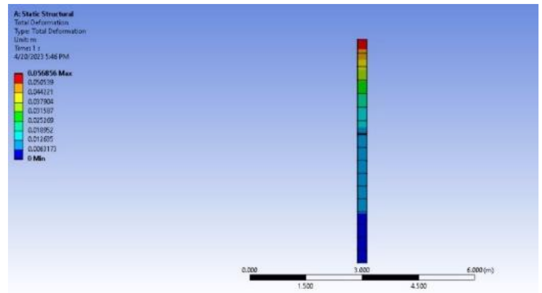

Figure 2. Temperature-induced deformation at 500 °C.

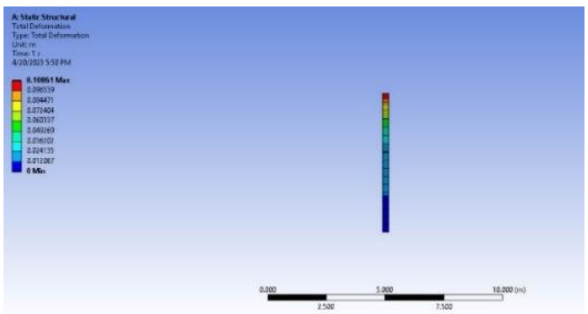

Figure 3. Deformation in a steel column at 1000 °C.

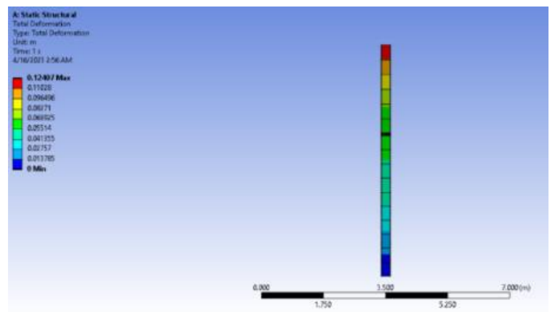

Figure 4. Deformation in a steel column at 1500 °C.

Similarly, the deformation of the steel beam at 1000 °C was 0.1086, as shown in Figure 5. Similarly, when beams were subjected to 1500 °C, the maximum deformation was 0.2351m, as shown in Figure 6. The reduced stiffness and strength of a material at high temperatures are caused by the thermal softening effect. When a steel beam is subjected to lower compressive stresses, this effect renders it more prone to buckling and deformation. The steel beam segment begins to show signs of cracking around 1000°C. At 1500°C, the steel beam underwent permanent deformation, thereby increasing the likelihood of structural collapse. The beam could not bear the applied stresses due to the thermal softening effect, which has reduced the beam's strength and stiffness [4]. Similar results of stress distribution and deformation patterns were found by the study conducted by [6]; however, there are limited studies that focused on the evaluation of stress variation in steel members in ANSYS at elevated temperatures. It is crucial to remember that steel has a melting point of roughly 1370°C and enters a liquid form at a temperature of about 1500°C. As a result, the steel beam loses its ability to support weight, ultimately leading to the building's collapse. The study conducted by Kodur et al. [3] determined that the yield strength and elastic modulus of steel can decrease significantly as temperature exceeds 400°C based on idealized stress-strain curves of structural steel at various temperatures. Similarly, steel may lose up to 50% of its yield strength as temperature increases up to 800°C. The results obtained in our study show a similar pattern of stress degradation and plastic deformation. The fire resistance of steel members can be improved by using insulation materials and introducing structural redundancy for the redistribution of loads and deformation, especially at joints [7].

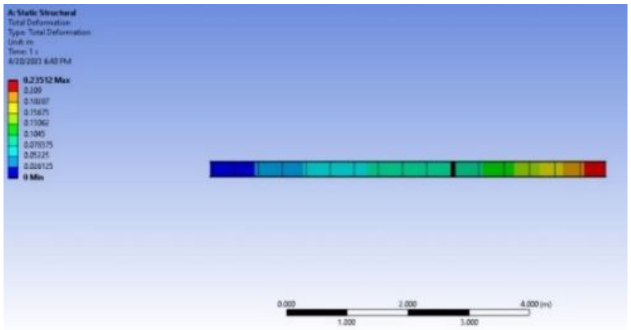

Figure 5. Deformation in the beam at 500 °C.

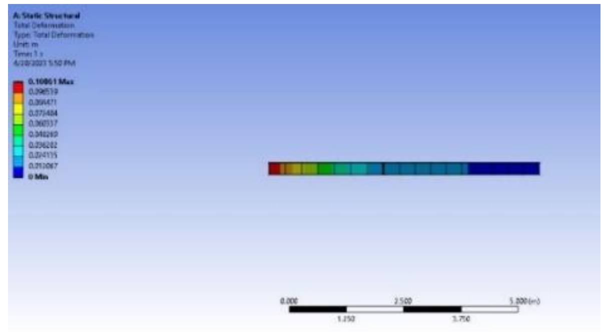

Figure 6. Deformation in the beam at 1000 °C.

4 Conclusion

The main purpose was to analyse the behaviour of structural components at different temperatures. The maximum deformation occurred in the steel column and beam at 1500 °C, which was 0.1240m and 0.2351m, respectively. The reduction in stiffness and strength resulted in increased deformation and the initiation of failure mechanisms due to the formation of plastic hinges and local buckling. This research validates the use of FEM methods for performance-based fire design and emphasises the need for additional experimental validation.

Acknowledgment

The authors are thankful to the Department of Civil Engineering, University of Engineering and Technology, Taxila.

References

[1] H. Ban, G. Zhou, H. Yu, Y. Shi, and K. Liu, "Mechanical properties and modelling of superior high-performance steel at elevated temperatures," *J. Constr. Steel Res.*, 2021, doi: 10.1016/j.jcsr.2020.106407.

[2] J. Chen, B. Young, and B. Uy, "Behavior of High Strength Structural Steel at Elevated Temperatures," *J. Struct. Eng.*, 2006, doi: 10.1061/(asce)0733-9445(2006)132:12(1948).

[3] V. Kodur, M. Dwaikat, and R. Fike, "High-Temperature Properties of Steel for Fire Resistance Modeling of Structures," *J. Mater. Civ. Eng.*, 2010, doi: 10.1061/(asce)mt.1943-5533.0000041.

[4] M. Nassar and L. Amleh, 'Predicting Maximum Effective Temperatures and Thermal Gradients for Steel I-Girder in Canadian Climate Regions', *Applied Sciences (Switzerland)*, vol. 13, no. 10, 2023, doi: 10.3390/app13105906.

[5] J. DING and Y. WANG, 'Finite element analysis of concrete filled steel columns in fire', in *Fourth International Conference on Advances in Steel Structures*, 2005. doi: 10.1016/b978-008044637-0/50155-4.

[6] L. M. Lu, G. L. Yuan, Q. J. Shu, and Q. T. Li, "Numerical analysis on the fire behavior of a steel truss structure," in *Life-Cycle Analysis and Assessment in Civil Engineering: Towards an Integrated Vision - Proceedings of the 6th International Symposium on Life-Cycle Civil Engineering, IALCCE 2018*, 2019.

[7] Wajahat, Raja, Zahoor Khan, Muhammad Yaqub, Muhammad Noman, and Rana Muhammad Waqas. 2021. "Evaluation of Mechanical and Microstructural Properties of Mortar Reinforced with Carbon Nanotubes at Elevated Temperatures. "Pakistan Journal of Engineering and Technology 1–6