© July 2025 | IJIRT | Volume 12 Issue 2| ISSN: 2349-6002

Static Structural Analysis of a Hollow Rectangular Cantilever Beam Using ANSYS

Piyush Dutta
Summer Intern, Talisha Aerospace, Durgapur, West Bengal, India

***Abstract*—It is a static structural analysis study on hollow rectangular cantilever beam using ANSYS and with the aim of viewing its mechanical behavior under loading with a point load. The beam was considered to be made of structural steel with an inner dimension of 25 mm × 15 mm and with equal thickness at walls of 2 mm and thereby having an outside cross-sectional dimension of 29 mm × 19 mm. The total length of the length of beam is 182.88 mm. It was given fixed support on one end and vertical point load of 82 N at the free end on a standard cantilever load case. Finite element analysis was conducted on ANSYS for determining parameters such as overall deformation, equivalent (von Mises) distribution and strain distribution. The FEA results were compared with analytically determined values using classical beam bending theory to verify of model. There was good correlation between the results from analytical and FEA which ensures ANSYS is ideal for structural analysis. This study demonstrates the credibility of simulation software in finding and optimizing structural behavior particularly for components with localized load. Index Terms—Hollow Rectangular Cantilever Beam, Point Load, FEA(Ansys), Catia v5**

***Index Terms*—Hollow Rectangular Cantilever Beam, Point Load, FEA(Ansys), Catia v5**

I. INTRODUCTION

Cantilever beams are simple structural elements that find extensive applications in the field of engineering such as bridges, building structures, and mechanical equipment with a single anchored end and the other end that has no support to bear force applications. Accurate analysis of their mechanical behavior under various loading conditions is critical for their efficiency and dependability. Finite element analysis (FEA) has found widespread applications in today's engineering designs as a highly effective tool utilized for simulating and forecasting complicated structures at high accuracy levels.

In this case, we are interested in the static deflection of a hollow rectangular cantilever beam made of structural steel and loaded by a point load applied on its free end. With the assistance of ANSYS, a renowned FEA program, the work endeavors to examine significant mechanical responses such as overall deformation, equivalent (von Mises) stress, and strain distribution. With its hollow rectangular crosssectional structure and fixed-free boundary condition, such a
beam configuration constitutes a rather common design used in structural work and thus provides such work with practical implications and application.
To confirm the numerical model, simulation response is verified with theory-based expectations of conventional beam bending theory. The compatibility between analytical and
computational solutions not only confirm the effectiveness of FEA methods but also demonstrates the effectiveness of simulation tools for improving design on a structural level. This study adds depth to an overall understanding of cantilever response for solid loading and concentrated loading and solidifies computational resources' application in engineering analysis.

II. LITERATURE REVIEW

There have also been several research works on the behavior of cantilever beams through analytical solutions and numerical modeling. Pravin et al. (2017) carried out a finite element study on statically loaded cantilever beams and found that accurate distribution of stresses can only be achieved through careful refinement of mesh. However, their work only used solid rectangular shapes and didn't include how hollow geometries influence concentrations of stresses and deformation. Kumar and Reddy (2019) studied static and modal characteristics of hollow rectangular beams. They revealed that natural frequency and stiffness of beams significantly rely on crosssectional geometry. But their study was focused on modal behavior and didn't include an explicit comparison between theoretical and simulation-based static behavior. Singh and Patel (2021) carried out a comparison between beams of various structural materials utilizing ANSYS. Though their work shed light on material-dependent responses of stress, their work didn't extensively consider different directions of load or boundary conditions and didn't compare deflection precision through theoretical models. It was highlighted by Rajasekaran et al. (2020) how CATIA V5 and ANSYS can be integrated together for convenient mechanical modeling and simulation. Workflow effectiveness was demonstrated through their work, but there was no consideration for issues like meshing errors, element quality checks, or non-linear behavior for larger loading cases. Limitations Found:

- Limited focus on hollow beam sections in theoretical validation.
- Leaving out vertical point load effects from the majority of previous work.
- Lack of convergence studies and sensitivity analysis on mesh size.
- Lack of complete comparison between FEA solutions and classical beam theory.

Most of these papers took account of static load in a horizontal direction, but practical applications experience vertical or off-axis loading.

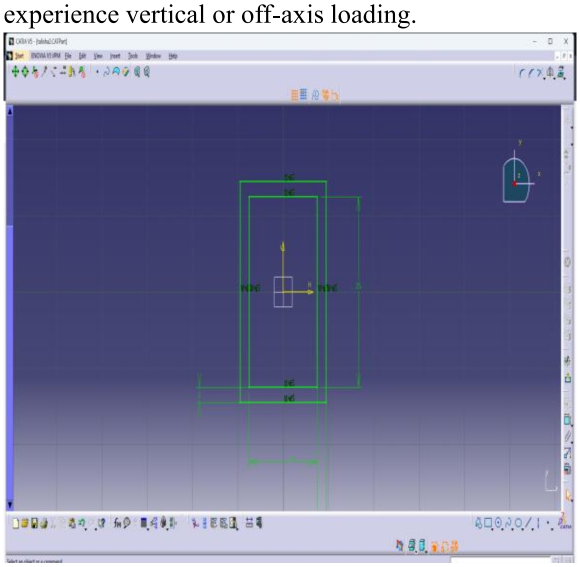

Fig. 1. Cross Section of Hollow Rectangular Beam with Dimensions

Such gaps highlight the demand for research work involving CAD-based geometry, FEA simulation, and theoretical validation for hollow cantilever beams under vertical point load—discussed in this work.

III. METHODOLOGY

The hollow rectangular cantilever beam discussed above was created using CATIA V5, a powerful 3D computer-aided design (CAD) software provided by Dassault Systemes. CA-` TIA is widely used in engineering disciplines and particularly used for aerospace engineering due to its precision, parametric control, and integration.

A. Modeling Process

The beam model was created using CATIA V5's Part Design module by these steps:

- A 2D model was generated on the XY plane and consisted of two concentric rectangles:
- The external rectangle was 29 mm × 19 mm and was thus the overall cross-sectional dimension.
- The inner rectangle was 15 mm × 25 mm and was a hollow section with an even wall thickness of 2 mm.
- The hollow profile was extracted from the area between the rectangles.
- Then the sketch was extruded (by utilizing the feature Pad) depth of 182.88 mm and generate final 3D hollow beam configuration.

The model was exported and saved as STEP (.stp) files to be used with ANSYS for further study.

B. Advantages of Utilizing CATIA V5

CATIA offers several advantages for engineering design work, particularly for high-precision work like aerospace:

- Parametric Flexibility: Designers can instantly change dimensions such as thickness, length, or cross-section without re-creating their model, making them more efficient.
- Accuracy and Reliability: CATIA offers very high modeling accuracy, which is critical for simulations wherein geometric fidelity impacts outcomes.
- Integration with Simulation Tools: The exported geometry is correct and seamlessly interoperable with simulation software packages like ANSYS.

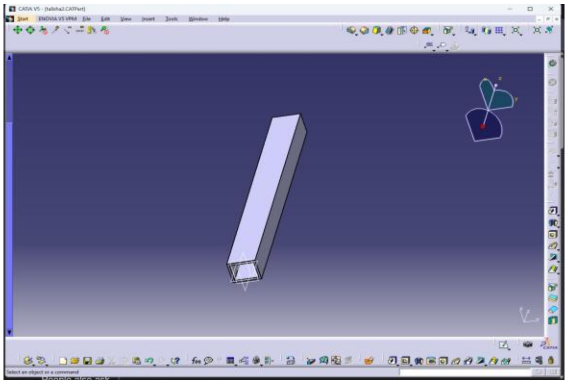

Fig. 2. Hollow Rectangular Beam

- Advanced Visualization: CATIA provides real-time 3D visualization and engineers can detect design flaws and inconsistencies even prior to prototype building.
- Lightweight and Complex Shape Moulding: Particularly suitable for aerospace projects, CATIA can handle optimized hollow forms and intricate geometries essential for weight loss.

C. Relevance in the Aerospace Industry

CATIA is regarded as an industry standard for the aerospace industry because it can efficiently work on large assemblies, complex shapes, and multi-disciplinary integration. Top aerospace industry leaders like Airbus, Boeing, and ISRO use CATIA for generating designs for components such as fuselage frames and wing spars and internal support structure. It can manufacture structurally optimal and lightweight components with high accuracy, as required by industry standards for high performance and cost effectiveness and safety. Utilization of CATIA V5 employed herein ensures that beam design sticks tightly reflect real-world applications and provides a strong foundation for accurate finite element analysis at subsequent stages of this research.

D. Finite Element Solution by ANSYS

Through ANSYS 2025 R1, a hollow rectangular cantilever beam was given static structural analysis. The beam was modeled and meshed with tetrahedral elements. The beam was loaded with a point load of 82 N acting on the free end upwards and fixed on its other end.

The following key results were obtained: Total Deformation:

Maximum Deformation = 4.6352×10^{-5} m = 0.04635 mm

(1)

Equivalent (von Mises) Stress:

Maximum Stress = 1.3642×10^{7} Pa = 13.64 MPa (2)

Directional Deformation (X-axis):

$-1.6446 \times 10^{-7}\ \text{m} \le u_x \le 1.9368 \times 10^{-7}\ \text{m}$ (3)

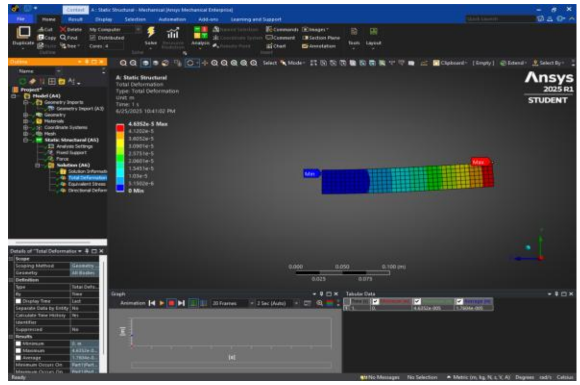

Fig. 3. Total Deformation Using Ansys

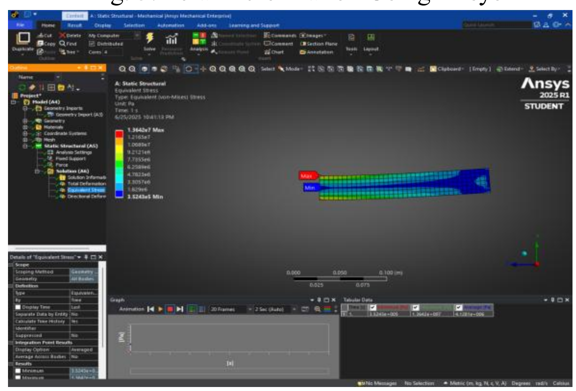

Fig. 4. Equivalent Stress Using Ansys

Indicates slight lateral deformation due to mesh and structural stiffness

These displacements exhibit stable and predictable behavior for applied load, characteristic for steel cantilevered configurations.

E. Comparison with Theoretical Calculations and Error Analysis

To confirm the ANSYS outcomes, we carried out theoretical calculations through Euler-Bernoulli beam theory using the

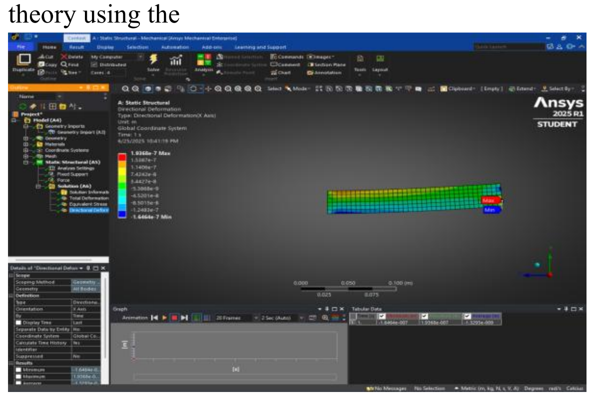

Fig. 5. Directional Deformation using Ansys

following formula for a cantilever beam's tip deflection under a point load:

$$\delta = \frac{PL^3}{3EI} \quad (4)$$

Where:

- P = 82 N
- L = 182.88 mm = 0.18288 m
- $E = 200 \times 10^9$ Pa (Structural Steel)
- $I = \frac{bh \; - b_i h_i}{12}$ (Moment of inertia for hollow section)

Moment of Inertia Calculation

- Outer dimensions: b = 29 mm, h = 19 mm
- Inner dimensions: b_i = 25 mm, h_i = 15 mm

$$I = \frac{29 \cdot 19^3 - 25 \cdot 15^3}{12}$$

$$= \frac{29 \cdot 6859 - 25 \cdot 3375}{12}$$

$$= \frac{198911 - 84375}{12}$$

$$= \frac{114536}{12} = 9544.67 \text{ mm}^4 = 9.5447 \times 10^{-9} \text{ m}^4$$

Theoretical Deflection

$$\delta = \frac{82 \cdot (0.18288)^3}{3 \cdot 200 \times 10^9 \cdot 9.5447 \times 10^{-9}}$$

≈ 0.04513 mm

Theoretical Calculation of Maximum Bending Stress

$$\sigma_{\max} = \frac{M_{\max} \cdot c}{I} \quad (5)$$

Where:

- $M_{max} = P \cdot L$ (Maximum bending moment)
- c = Distance from the neutral axis to the outermost fiber
- I = Moment of inertia of the cross-section

TABLE I

COMPARISON BETWEEN THEORETICAL AND ANSYS RESULTS

Parameter	Theoretical	ANSYS	Error (%)
Max Deformation (mm)	0.04513	0.04635	≈ 2.7%
Max von Mises Stress (MPa)	14.93	13.64	≈ 8.6%

Known Values

- Applied Load: P = 82 N
- Length: L = 182.88 mm = 0.18288 m
- Outer Height: h = 19 mm $\Rightarrow c = \frac{h}{2} = 9.5$ mm = 0.0095 m
- Moment of Inertia: $I = 9.5447 \times 10^{-9}$ m^4

Step-by-Step Calculation Moment:

M = P · L = 82 · 0.18288 = 15.00016 Nm

Stress:

$$\sigma_{\max} = \frac{M \cdot c}{I} = \frac{15.00016 \cdot 0.0095}{9.5447 \times 10^{-9}}$$

$$= \frac{0.1425}{9.5447 \times 10^{-9}}$$

≈ 1.493×10^7 Pa = 14.93 MPa

F. Conclusion

The theoretical bending stress is 14.93 MPa, and the ANSYS program gives a result of 13.64 MPa.

The error thus obtained is approximately 8.6%, which lies within an acceptable range for finite element analysis (FEA).

Sources of Discrepancy

- ANSYS von Mises stress accounts for a triaxial stress state but not purely in bending.
- Numerical interpolation and mesh resolution can influence localized stress readings.
- These are very approximate calculations based on extremely simple assumptions, which do not include shear deformation or zones of stress concentration.

This comparison gives evidence that ANSYS's outcomes strongly agree with classical beam theory and confirms the correctness of the finite element model and proves the effectiveness of the whole simulation method.

IV. FUTURE SCOPE

It provides a basic insight into how hollow rectangular cantilever beams behave when static load is imparted through finite element simulation. But there are also certain areas which offer scope for experimentation and development:

- Materials Variation: For future projects as well, there can also be tried experiments with newer lightweight materials such as composites, titanium alloys, or carbon fiberreinforced polymers (CFRPs) which are greatly utilized in aerospace engineering due to their very large strengthto-weight ratios.
- Dynamic and Fatigue Analysis: Incorporating timedependent loading conditions, vibrations, or fatigue cycles can offer a more accurate estimate on how the beam would react when subjected to

operational conditions typical in real-case applications.

- Coupling Thermo-Structural: One can actually simulate thermal effects on thermo-structural behavior of the beam and potentially could represent unfavorable thermal environments typical to aerospace components and thereby enhance the reality of the analysis.
- Topology Optimization: Topology and shape optimization methods can be used to reduce weight and maximize strength and stiffness—important considerations for aerospace and automotive applications.
- Experimental Verification: It is possible to develop and experimentally test a hollow beam prototype and use such experiments for further verification and adjustment of simulation outcomes.
- Complex Boundary Conditions: Practical constraints such as partial fixity, load eccentricity, and multi-point loading can be modeled to more nearly replicate field usage conditions.
- Integration with Additive Manufacturing: The future can also be directed towards investigating manufacturability with hollow geometries optimized using 3D printing, thus facilitating internal features and complex geometries.

REFERENCES

[1] Pravin, A., Baskar, P. (2017). "Finite Element Analysis of Cantilever Beam Using ANSYS". International Journal of Engineering Research and Applications, 7(4), 45–49.

[2] Kumar, N., Reddy, P. S. (2019). "Modal and Static Analysis of Cantilever Beam with Different Cross Sections Using ANSYS". International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), 9(3), 1039–1050.

[3] Singh, G., Patel, M. (2021). "Comparative Static Structural Analysis of Cantilever Beam of Different Materials using ANSYS". Materials Today: Proceedings, 44, 3455–3460.

[4] Rajasekaran, R., Murugan, S. (2020). "Integration of CAD and FEA for Mechanical Component Analysis Using CATIA and ANSYS". Journal of Engineering and Applied Sciences, 15(7), 1751–1756.

[5] Dey, S., Patel, B. (2020). "Structural Analysis of Hollow Beam Sections with Different Geometries". International Research Journal of Engineering and Technology (IRJET), 7(5), 1052–1056.

[6] Mishra, S., Tiwari, S. (2018). "Finite Element Analysis of Beams for Static and Dynamic Loading". International Journal of Engineering Trends and Technology (IJETT), 60(1), 34–39.

[7] Bhat, R. R., Alti, A. R. (2015). "Effect of Thickness on Stress and Deflection of Hollow Beams using Finite Element Analysis". International Journal of Emerging Technology and Advanced Engineering, 5(1), 257–262.