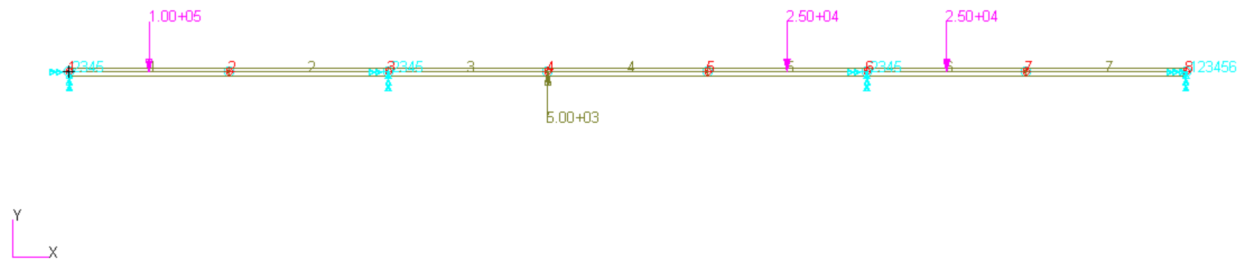
Homework 9 Stress Report



Tiger Sievers

Instructor: Dr.Richard Hale

Department of Aerospace Engineering

May 30, 2025

Executive Summary

This report presents a finite element analysis of a statically loaded beam structure composed of seven prismatic elements, each 1 m in length and featuring a rectangular cross-section. The model was developed in PATRAN, where the structural configuration included roller supports at Nodes 1, 3, and 6, and a fixed support at Node 8 to simulate realistic boundary conditions. Loading conditions involved a 100 kN/m distributed load applied to Element 1, a 25 kN/m distributed load over Elements 5 and 6, and a concentrated point load at Node 4.

The material used in the model was assumed to behave in a linear elastic manner with a Young's Modulus of 70 GPa. No failure or maximum stress criteria were applied, simplifying the analysis to focus on elastic structural response.

The finite element solution provided reaction forces at all constrained nodes, validated through equilibrium equations. Shear force and bending moment diagrams were derived from the internal load distribution, and these were used to compute both shear and bending stresses across the structure.

Key results from the analysis include:

- Maximum bending moment: $1.9939 \times 10^5\ \mathrm{N \cdot m}$
- Maximum bending stress: $1.098 \times 10^{10}\ \mathrm{N/m^2}$
- Reaction forces at Nodes 1, 3, 6, and 8 matched expected static equilibrium values
- Displacement and internal force results aligned with theoretical behavior of beam elements under mixed loading

This analysis confirms that the modeled beam system can support the given loading conditions within the assumed linear elastic regime. The methodology and outcomes serve as a foundational validation for future enhancements, such as incorporating material nonlinearity, variable cross-sections, or dynamic load analysis.

Contents

List of Figures

List of Tables

1. Introduction

This report presents a finite element analysis of a beam structure composed of seven prismatic elements. Each beam is in length with a rectangular cross-section, as shown in Figure 2.0.0.1 . The structural configuration includes a combination of roller and fixed boundary conditions to simulate realistic support conditions. The structure is subjected to both distributed and concentrated loads, and the resulting internal forces and stresses are analyzed.

The primary objective of this analysis is to determine the internal shear forces, bending moments, and corresponding stress distributions across the structure. Using principles from classical beam theory, shear and moment diagrams are developed, and analytical stress evaluations are performed. This investigation provides a foundational understanding of the beam's mechanical behavior under prescribed loading scenarios.

2. Geometry

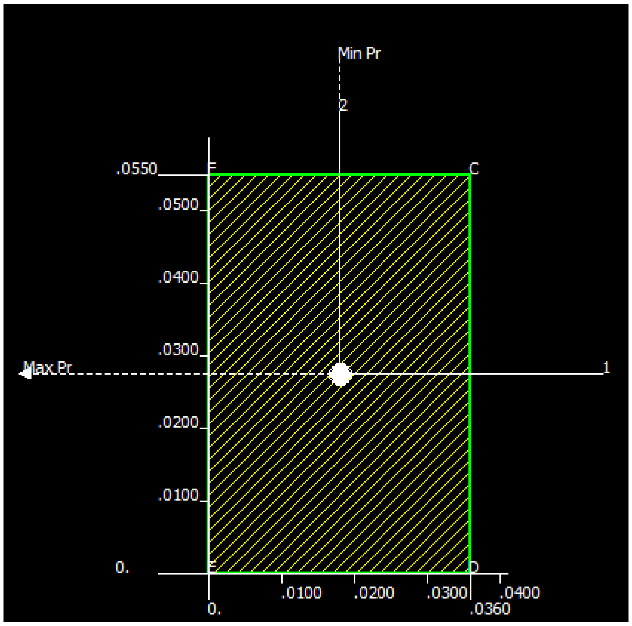


Fig. 2.0.0.1 Problem 10.16 Geometry

The structural model is composed of seven beam elements, each 1 m in length. All elements are rectangular cross-section beams, with the geometric dimensions illustrated in Figure 2.0.0.1 .

3. Loading and Boundary Conditions

The model incorporates a combination of distributed loads, point loads, roller supports, and a fixed support to simulate realistic structural behavior. Distributed loads are used to represent continuous forces along the length of a beam element; in this case, a 100 kN/m load is applied to Element 1 and a 25 kN/m load is applied across Elements 5 and 6. A concentrated point load is applied at Node 4 to simulate a localized force input. Roller supports are applied at Nodes 1, 3, and 6, allowing horizontal translation and rotation while restricting vertical displacement. These are used to model simple supports that prevent vertical movement without over-constraining the system. A fixed support is applied at Node 8, fully restraining translation and rotation, thereby anchoring the structure. These boundary and loading conditions define the constraints and external influences acting on the system throughout the analysis.

4. Materials

The structure is made of an unknown material with a Young's Modulus of 70 GPa. Material was assumed to behave in a linear elastic manner and has no maximum stress.

5. Finite Element Model

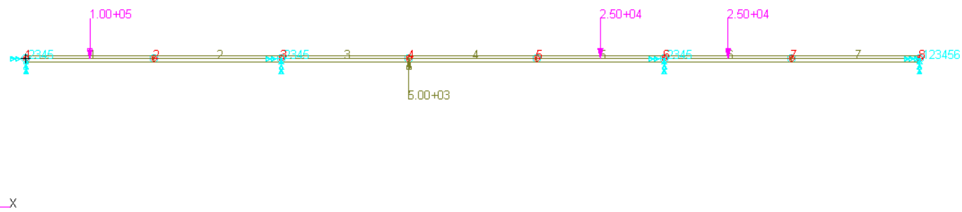


Fig. 5.0.0.1 PATRAN Model

The structural model consists of seven beam elements, each measuring 1 m in length. Roller displacement constraints are applied at Nodes 1, 3, and 6, while a fixed boundary condition is imposed at Node 8. A distributed load of 100 kN/m is applied to Element 1, and an additional 25 kN/m distributed load is applied across Elements 5 and 6. A concentrated point load is also applied at Node 4.

The results from the Finite Element Model are shown in Figure 5.0.0.2 showing that the maximum normal stress from bending is shown in Element 1 with 1.109379×10^9 Pa.

S T R E S S   D I S T R I B U T I O N   I N   B A R   E L E M E N T S   (C B A R)

0 ELEMENT ID.	STATION (PCT)	SXC	SXD	SXE	SXF	AXIAL	S-MAX	S-MIN	M.S.-T M.S.-C
1	0.000	5.364418E-07	-5.364418E-07	-5.364418E-07	5.364418E-07	0.0	5.364418E-07	-5.364418E-07	
1	1.000	-1.109379E+09	1.109379E+09	1.109379E+09	-1.109379E+09	0.0	1.109379E+09	-1.109379E+09	
2	0.000	-1.109379E+09	1.109379E+09	1.109379E+09	-1.109379E+09	0.0	1.109379E+09	-1.109379E+09	
2	1.000	5.360628E+08	-5.360628E+08	-5.360628E+08	5.360628E+08	0.0	5.360628E+08	-5.360628E+08	
3	0.000	5.360628E+08	-5.360628E+08	-5.360628E+08	5.360628E+08	0.0	5.360628E+08	-5.360628E+08	
3	1.000	3.660745E+08	-3.660745E+08	-3.660745E+08	3.660745E+08	0.0	3.660745E+08	-3.660745E+08	
4	0.000	3.660745E+08	-3.660745E+08	-3.660745E+08	3.660745E+08	0.0	3.660745E+08	-3.660745E+08	
4	1.000	-7.939598E+07	7.939598E+07	7.939598E+07	-7.939598E+07	0.0	7.939598E+07	-7.939598E+07	
5	0.000	-7.939598E+07	7.939598E+07	7.939598E+07	-7.939598E+07	0.0	7.939598E+07	-7.939598E+07	
5	1.000	1.638389E+08	-1.638389E+08	-1.638389E+08	1.638389E+08	0.0	1.638389E+08	-1.638389E+08	
6	0.000	1.638389E+08	-1.638389E+08	-1.638389E+08	1.638389E+08	0.0	1.638389E+08	-1.638389E+08	
6	1.000	-1.527550E+08	1.527550E+08	1.527550E+08	-1.527550E+08	0.0	1.527550E+08	-1.527550E+08	
7	0.000	-1.527550E+08	1.527550E+08	1.527550E+08	-1.527550E+08	0.0	1.527550E+08	-1.527550E+08	
7	1.000	2.193564E+08	-2.193564E+08	-2.193564E+08	2.193564E+08	0.0	2.193564E+08	-2.193564E+08	

Fig. 5.0.0.2 Stress Distribution in Finite Element Model

6. Analysis

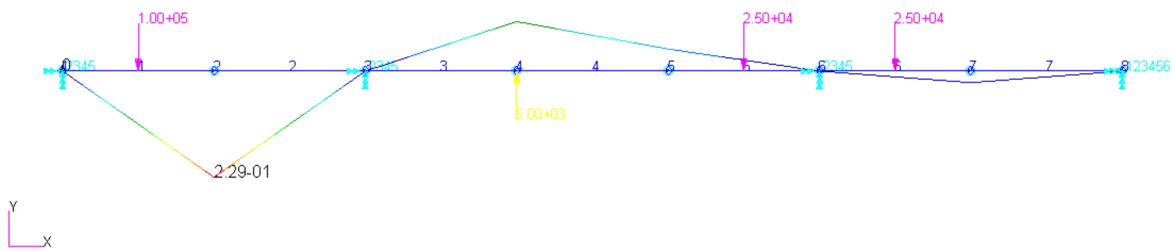


Fig. 6.0.0.1 PATRAN Model Displacements

6.1 Equilibrium Equations

To solve for the support reactions, apply the static equilibrium conditions:

$$\sum F_x = 0 \quad \text{(Sum of horizontal forces)}$$
$$\sum F_y = 0 \quad \text{(Sum of vertical forces)}$$
$$\sum M = 0 \quad \text{(Sum of moments about any point)}$$

Since there are no forces in the X-direction, the equilibrium equation can be ignored. The resulting reaction forces found in Table 6.1.1 were calculated in PATRAN [1].

Node	Reaction Force (N)
1	7.013523×10^4
3	3.295006×10^4
6	3.516089×10^4
8	6.753822×10^3

Table 6.1.1 Support Rection Forces

6.2 Shear Force Equations

The internal shear force $V(x)$ is related to the distributed load $w(x)$ by the differential equation:

$$\frac{dV(x)}{dx} = -w(x)$$

To find $V(x)$, integrate the load function:

$$V(x) = -\int w(x)\, dx + C_1$$

where C_1 is an integration constant determined from boundary conditions or continuity. The results are shown in Figure 6.2.0.1 .

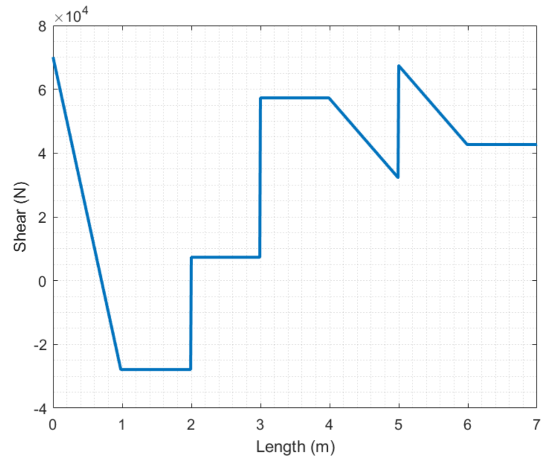


Fig. 6.2.0.1 Shear Over Structure Length

6.3 Bending Moment Equation

The bending moment $M(x)$ is obtained by integrating the shear force:

$$\frac{dM(x)}{dx} = V(x)$$

Thus, the internal moment is:

$$M(x) = \int V(x)\,dx + C_2$$

where C_2 is determined by moment boundary conditions. The results are shown in Figure 6.3.0.1 .

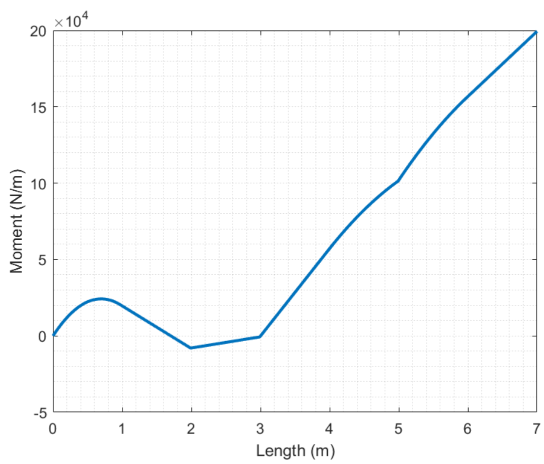


Fig. 6.3.0.1 Moment Over Structure Length

6.4 Shear Stress Calculation

For a rectangular cross-section, the internal shear stress τ is given by:

$$\tau = \frac{VQ}{Ib}$$

where:

- V is the internal shear force at the section,
- Q is the first moment of area about the neutral axis for the area above (or below) the point,
- I is the second moment of area of the entire cross-section,
- b is the width at the point of interest.

6.5 Bending Stress Calculation

The normal stress due to bending is calculated using the flexure formula:

$$\sigma = \frac{M(x)\,c}{I}$$

where:

- σ is the bending stress
- $M(x)$ is the internal moment
- c is the distance from the neutral axis to the outermost edge
- I is the second moment of area

From Figure 6.3.0.1 , the maximum moment is 1.9939×10^5 N-m. Adding this into the bending stress equation above, the maximum bending stress results in: 1.098×10^{10} N/m

7. Summary

The finite element analysis successfully modeled the structural response of a seven-element beam system under mixed loading and boundary conditions. Displacement results, as illustrated in Figure 6.0.0.1 , and internal force distributions were obtained using PATRAN. Reaction forces calculated at the supports aligned with equilibrium requirements, and shear and moment diagrams revealed expected behavior consistent with theoretical predictions.

Stress analysis was conducted using the computed shear forces and bending moments. The maximum bending moment was found to be , resulting in a peak bending stress of . The material, with a Young's Modulus of , was assumed to behave linearly elastically throughout the analysis.

In conclusion, the analysis demonstrates the structure's capability to carry the applied loads within the assumed linear elastic range. The modeling process and resulting stress evaluations provide a solid basis for further structural optimization or validation against experimental data. Future work may incorporate material nonlinearities or dynamic effects to enhance the fidelity of the analysis.

References

[1] "MSC Patran," , 2024.